

Introduction to Spice

SPICE Introduction Objectives

- Introduction to circuit simulation
- Installing PSpice on the student's computer
- Creating circuits to simulate
- Performing transient (time domain) simulations
- Performing AC (frequency domain) simulation
- Analyzing the results using GUI tools

**For questions, help, problems with this intro class:
Email to: Wolfgang@TestTechniques.com**

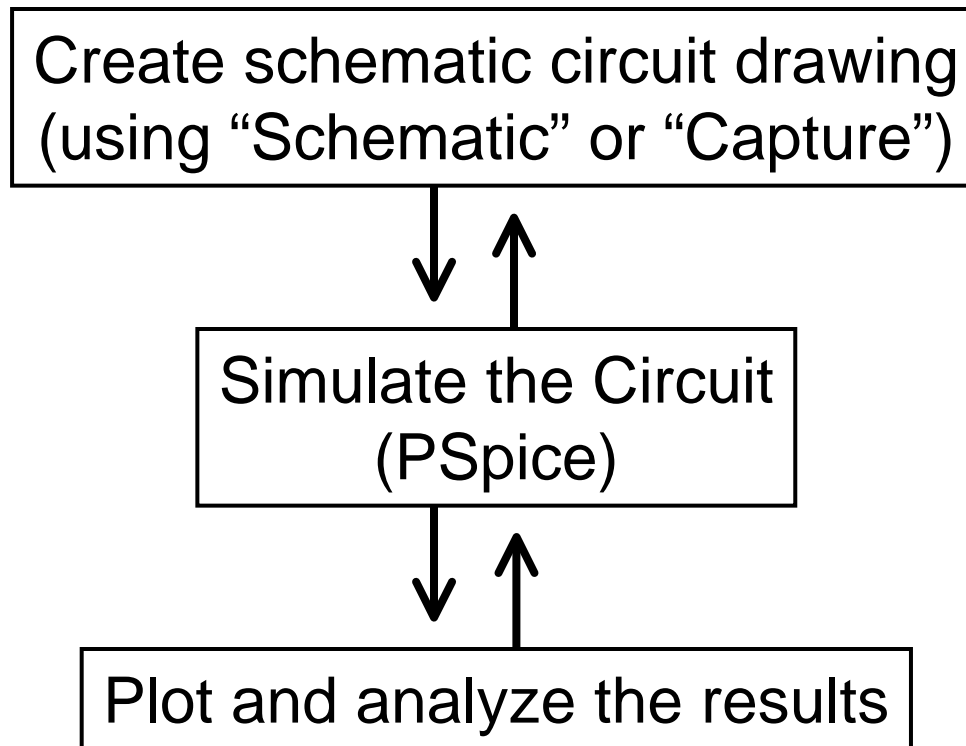
Spice Circuit Simulation

- **Spice = Simulation Program with Integrated Circuit Emphasis**
- Developed at UC Berkeley in the '60's
- Allows simulation of circuits without building physical prototypes.
- Can perform DC level, AC frequency response and transient time domain simulations.

The PSpice System

- The PSpice simulator itself uses “netlists” (text files that contain the circuit description) as input.
- Netlists are rather non-intuitive, and difficult to create, debug, and modify.
- Modern versions of Spice (including this one) have integrated graphical circuit editors, which then produce the netlists automatically.

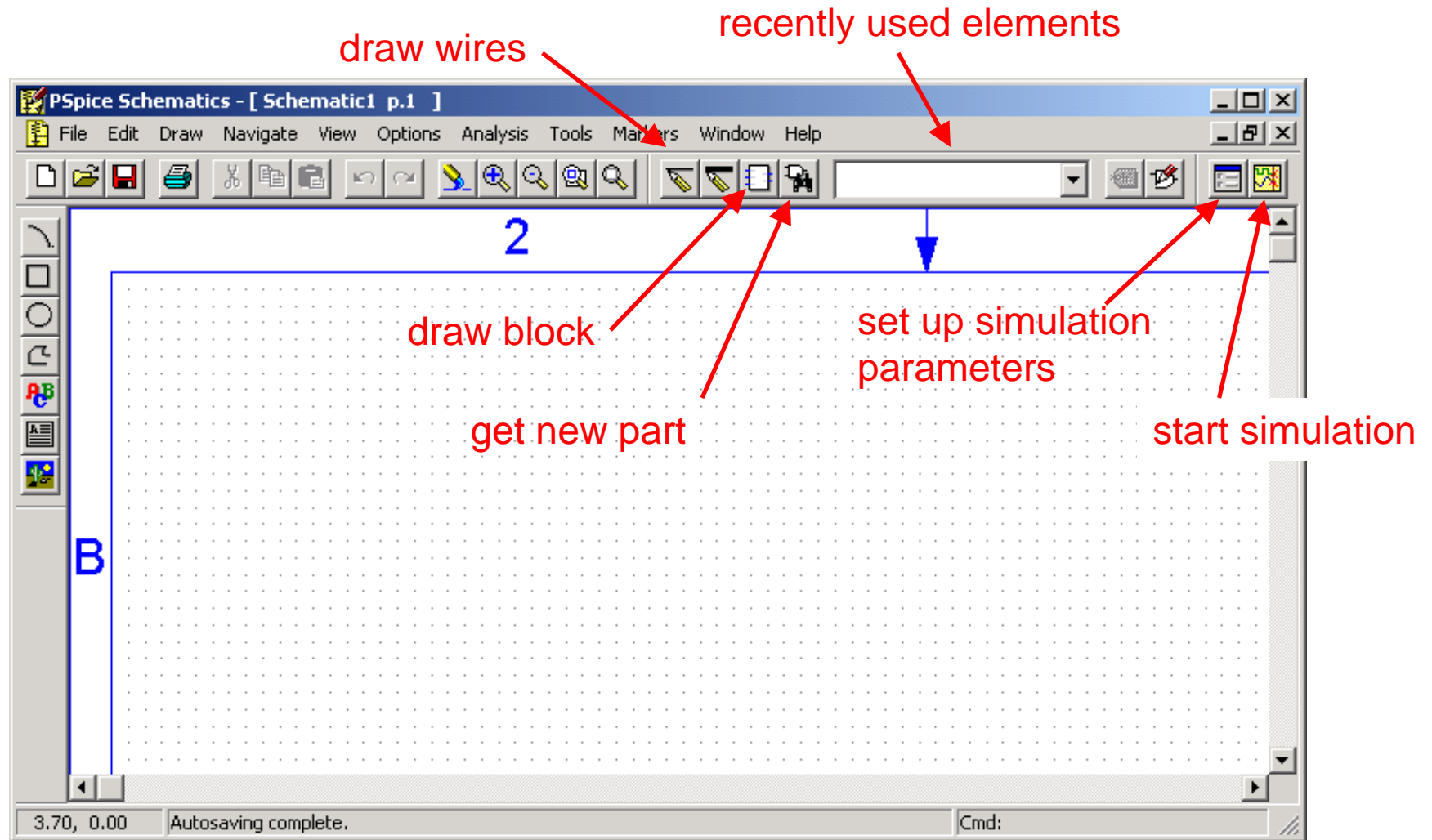
General Simulation Flow



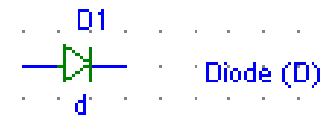
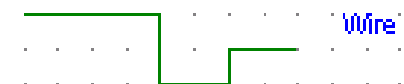
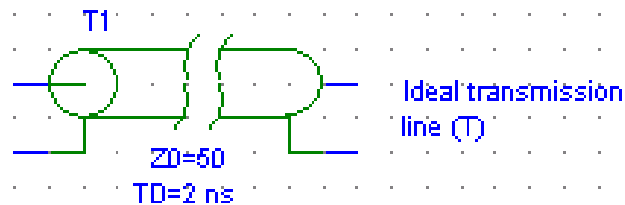
Installing PSpice

- The Spice version used for this class is the Student Version of Orcad (Cadence) PSpice V9.1.
- Can be downloaded and distributed for free from www.testtechniques.com → Free Tools
- Download and double-click on the installer to start the installation.
- When asked, choose “**Schematics**” as circuit editor (not “Capture”).
- Choose “PSpice Student -> Schematics” in the Windows Start menu to start the program.

PSpice Schematics Editor

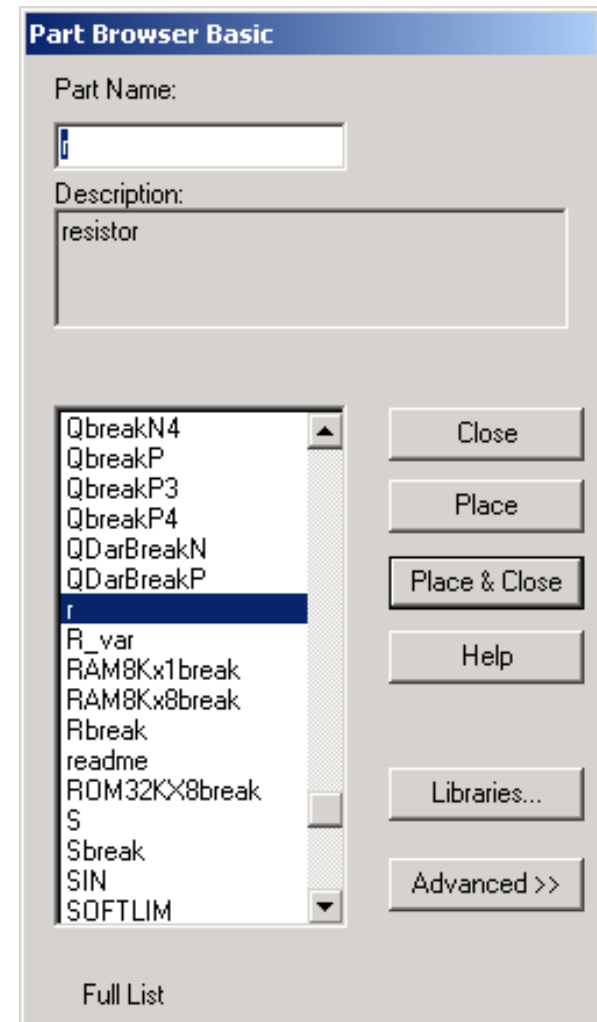


Frequently Used Circuit Elements



Creating a Simple Circuit

- Start Schematics
- Choose Draw / Get New Part (or click on the “Get New Part” icon)
- Select “r” (resistor)
- Click on “Place & Close”
- Place the resistor on the page:
 - move the mouse pointer to the desired position
 - left-click to place the resistor
 - right-click to end end placement of resistors



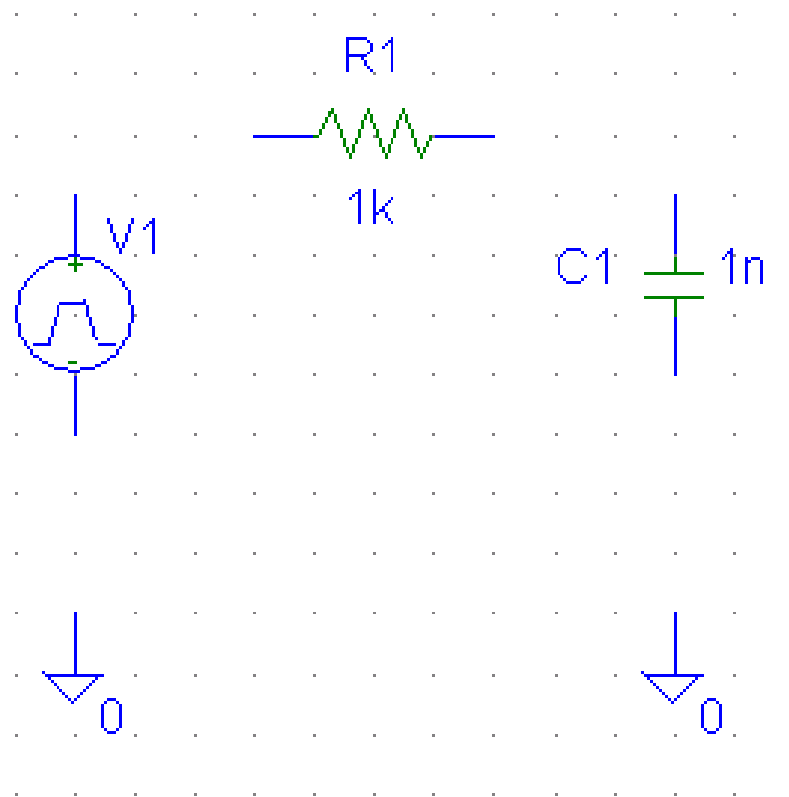
Creating a Simple Circuit (cont.)

Useful hints:

- You can place several elements (of the same type) in a row by choosing their position and the left-clicking
- Right-clicking ends placing (component symbol disappears)
- You can rotate elements by selecting them and pressing <CTRL> + R
- Groups of elements can be selected by clicking on a corner of the area and then dragging the mouse pointer to the opposite corner.
- Selected elements can be moved by clicking on them and then dragging them to their new position
- You delete elements by selecting them and then pressing “Delete”

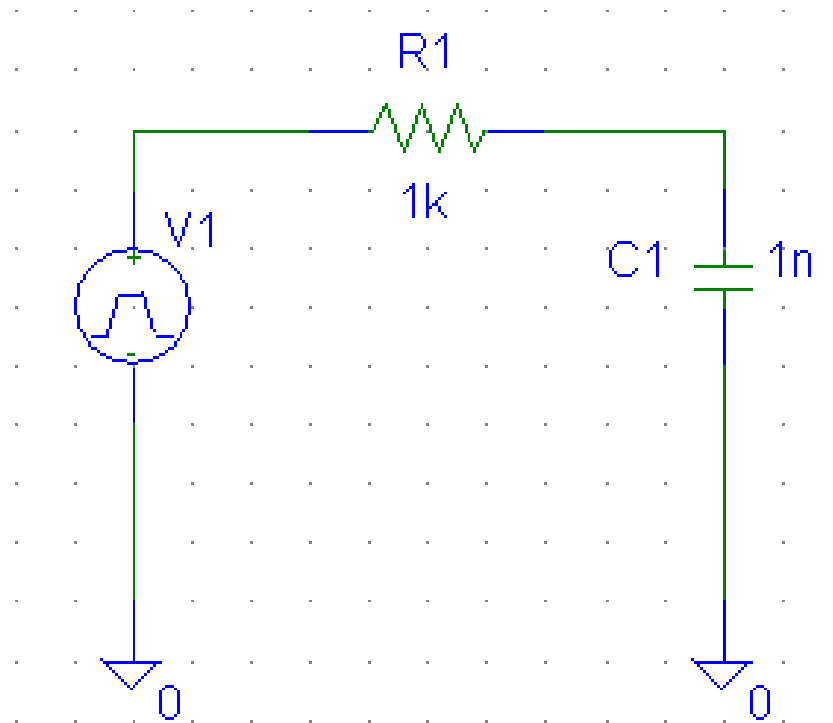
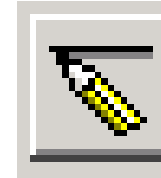
Creating a Simple Circuit (cont.)

- Now also place a capacitance (C), a pulse voltage source (VPULSE) and two analog grounds (AGND) on your schematics:



Creating a Simple Circuit (cont.)

- Click on the “Wire” icon and connect the elements:



Changing Element Properties

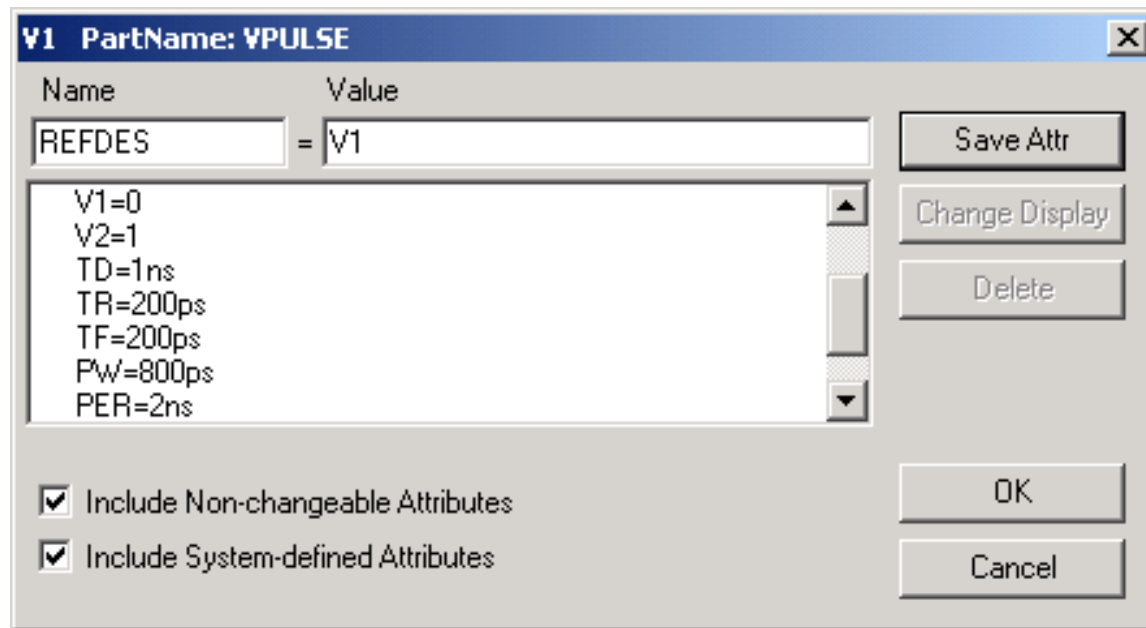
- When creating new elements, they are assigned a default name and default properties.
- To change a displayed property, click on it and enter a new value:
 - change the resistor value to 50 (the unit, Ohm, can be omitted)
 - change the resistor name to R_load (no spaces allowed!)

 - change the capacitance value to 4p
 - change the capacitance name to C_load

 - change the name of the pulse source to V_pulse
- **NOTE:** 'm' as well as 'M' denote "milli" (1e-3), not "mega" (1e6)

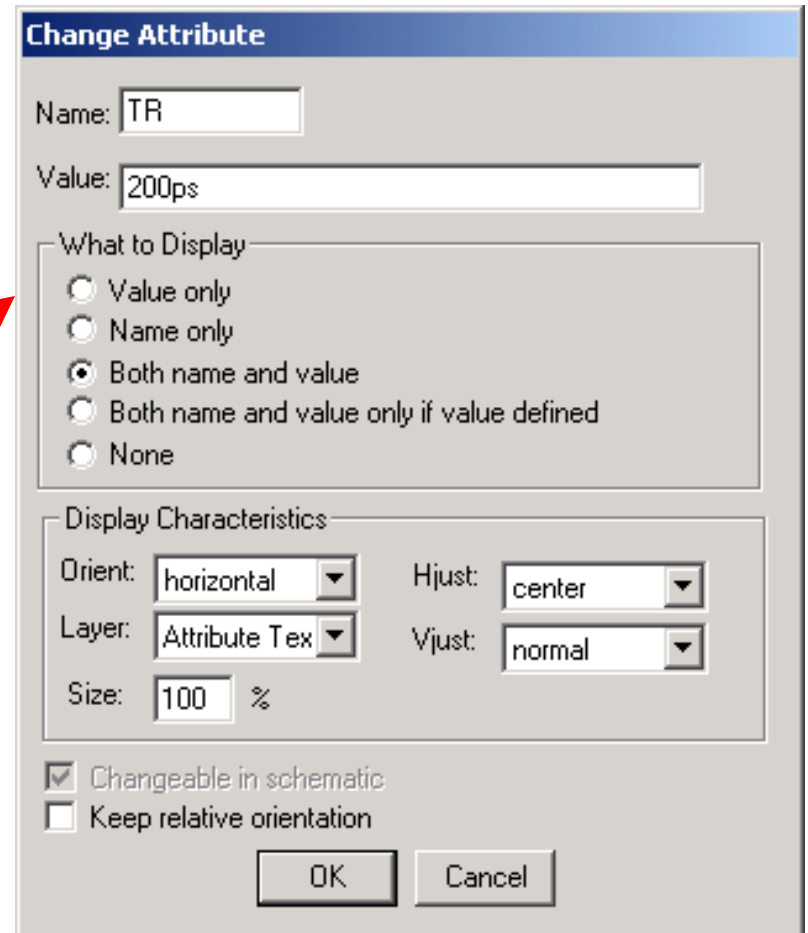
Changing Element Properties (cont.)

- To change properties not displayed directly on the page, you must double-click on the element to open its property display.
- For the voltage source, choose: V1=0V (low voltage), V2=1V (high voltage), TD=1ns (delay to first edge), TR=200ps (rise time), TF=200ps (fall time), PW=800ps (pulse width), PER=2ns (period)
- After entering an attribute value, you need to press “Save Attr”.



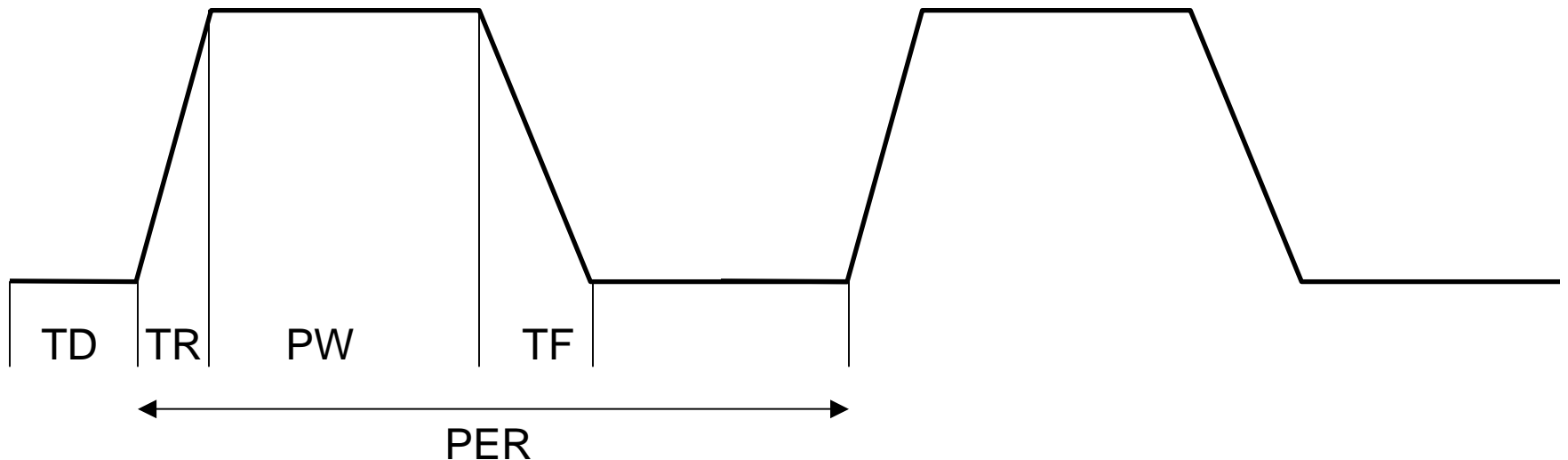
Changing Element Properties (cont.)

- To display additional properties of an element in the circuit drawing, choose the particular property in the element's property box and click on "Change Display".

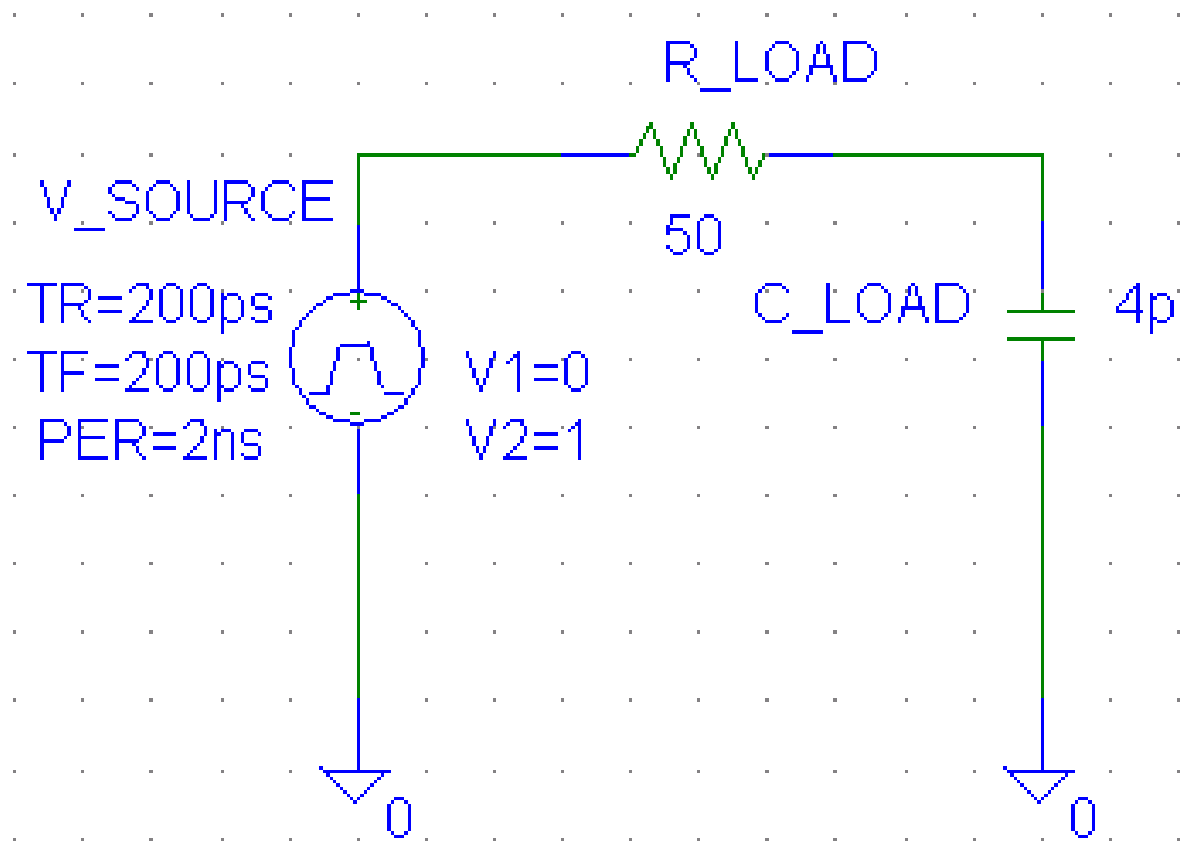


Pulse Source Parameters in PSpice

- The basic pulse source supplied by PSpice uses simple linear ramps. Later we will see how to create more realistic edge shapes.
- The rise time is 0% to 100%. You have to convert from $T_{10/90}$ or $T_{20/80}$.
- Also the definition of the pulse width is somewhat uncommon, as shown in the sketch below:



The finished RC filter

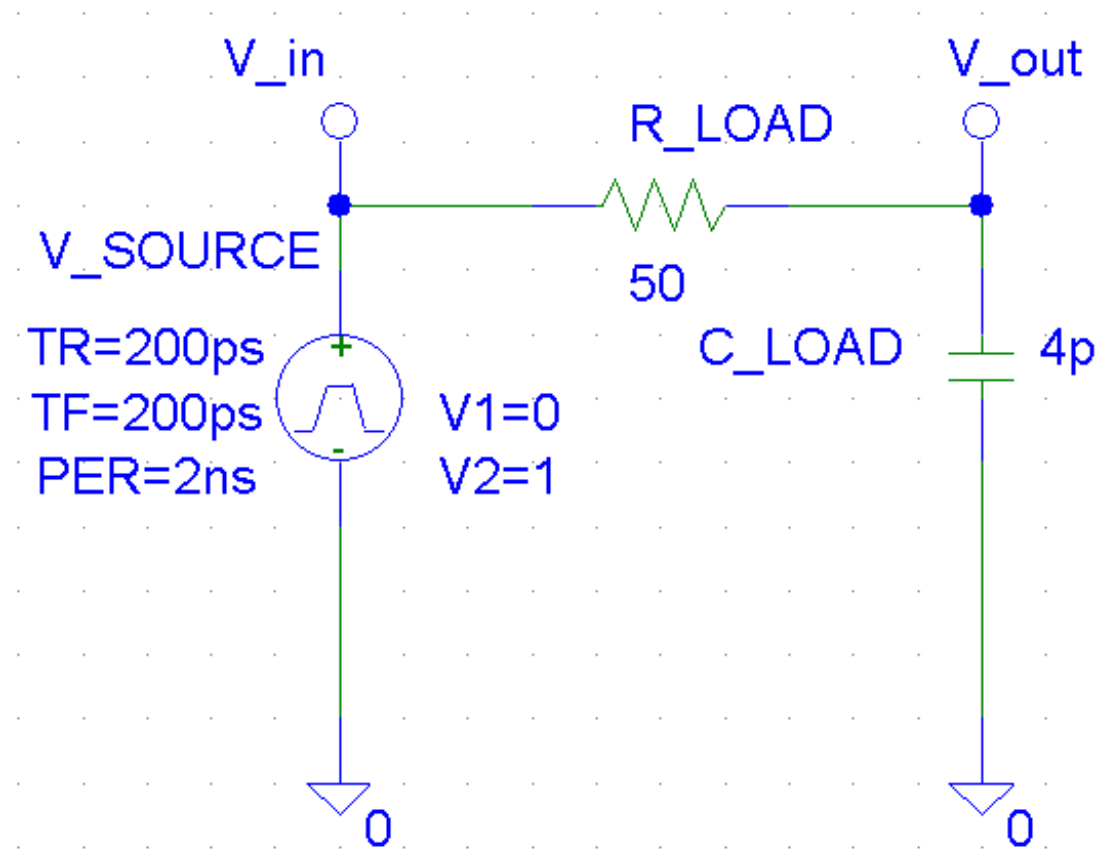


Adding probe points

- The “**Bubble**” elements can be used to add named probe points. It can also be used to provide invisible connection to a single node (e.g. power supply Vcc) by assigning the same name to several bubbles.
- After simulations, the time-resolved voltage on this points can be displayed.
- The bubbles have no default names. Before running any simulations, you must assign a unique label to each of the bubbles.
- As with all other component names, these labels may only contain letters and numbers (no spaces)

Adding probe points

- Add two bubbles into your design, one (labeled “V_in”) at the voltage source, and another one (labeled “V_out”) at the capacitance.



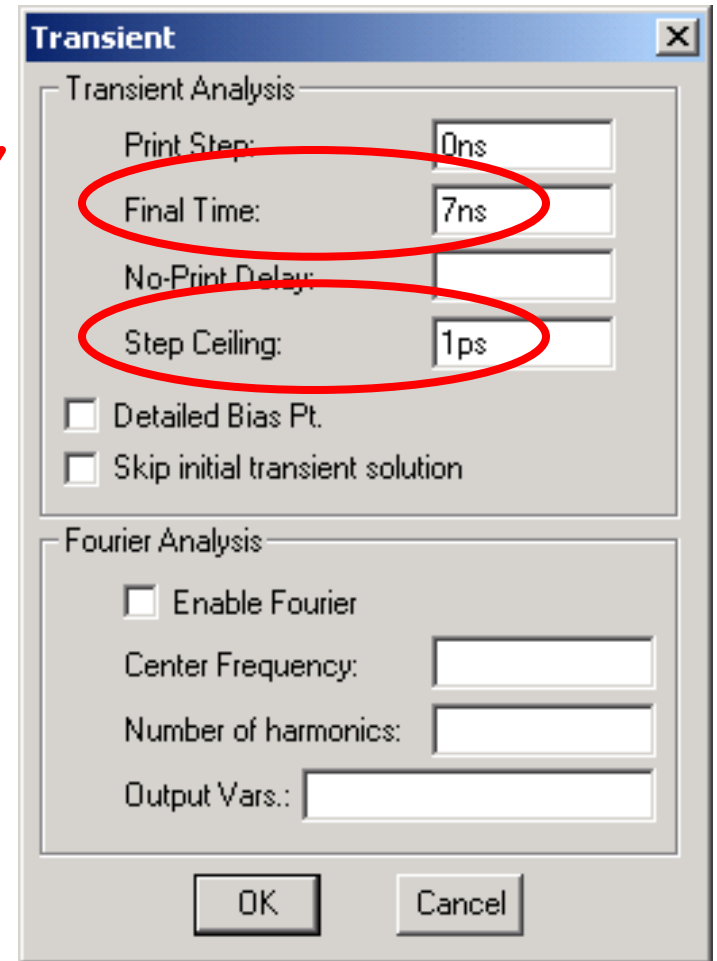
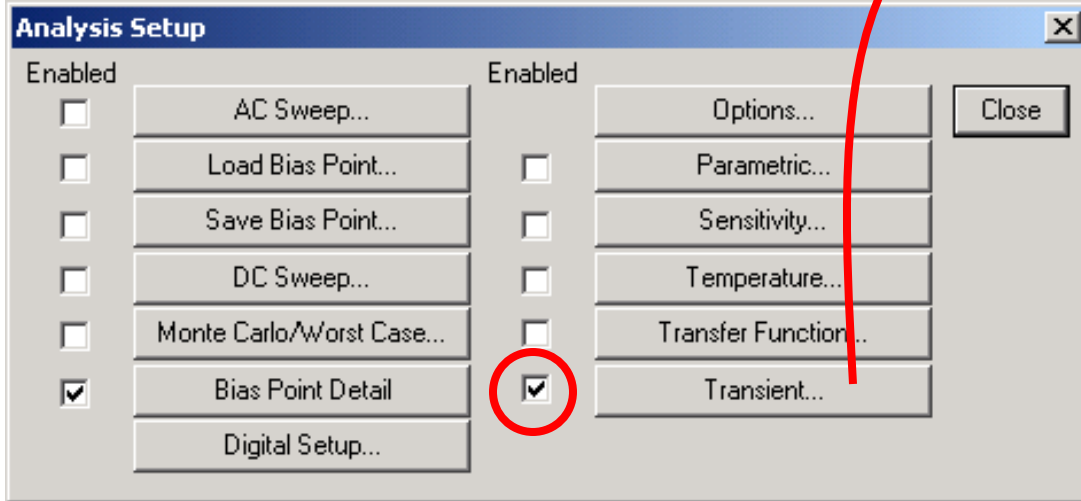
Renaming Elements

- When placing new elements, Schematics assigns unique default names to them (no two elements can have the same name).
 - If you copy an element or a group of elements, the names will get changed automatically.
 - However, when you rename some elements, the program can no longer do this. This results in different elements having the same name, and the simulation will not run (you will get an error if you try to simulate the circuit).
- ➔ Either don't rename elements you want to copy later, or rename the copied elements manually.

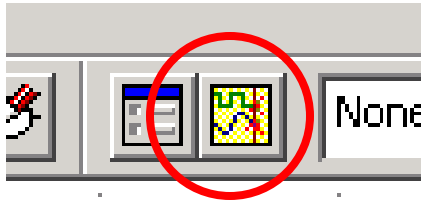
Setting Up a Transient Simulation (Time Domain Analysis)

- In this class (and for digital test simulations in general), the most commonly used simulation mode is “Transient” (time domain).
- Click on the “Analysis Setup Button” (or choose Analysis → Setup from the dropdown menu).
- Click on “Transient...” in the menu that opens.
- Enter “0ns” for Print Step, “7ns” for “Final Time” (start time is always 0), and “1ps” for Step Ceiling, then click on “OK”.
- To improve simulation speed, PSpice will always choose the maximum time step it deems accurate. Sometimes this can degrade accuracy, so if you want to limit this step width, enter the appropriate value (but too small a value will unnecessarily increase simulation time).

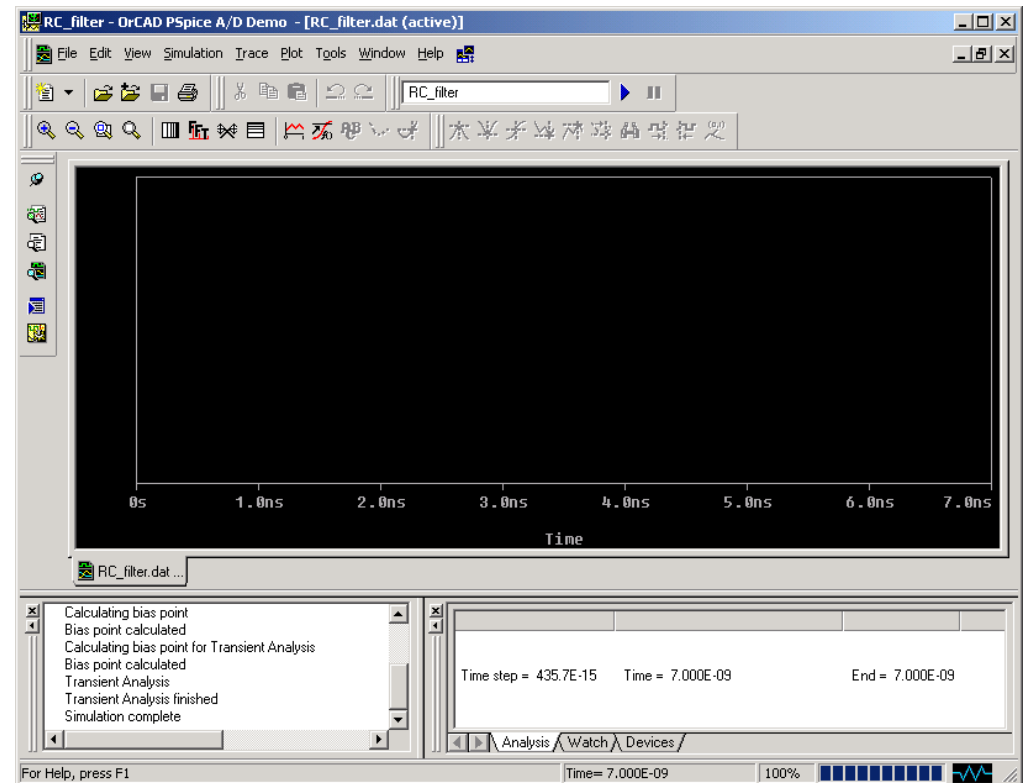
Setting Up a Transient Simulation



Running the Transient Simulation

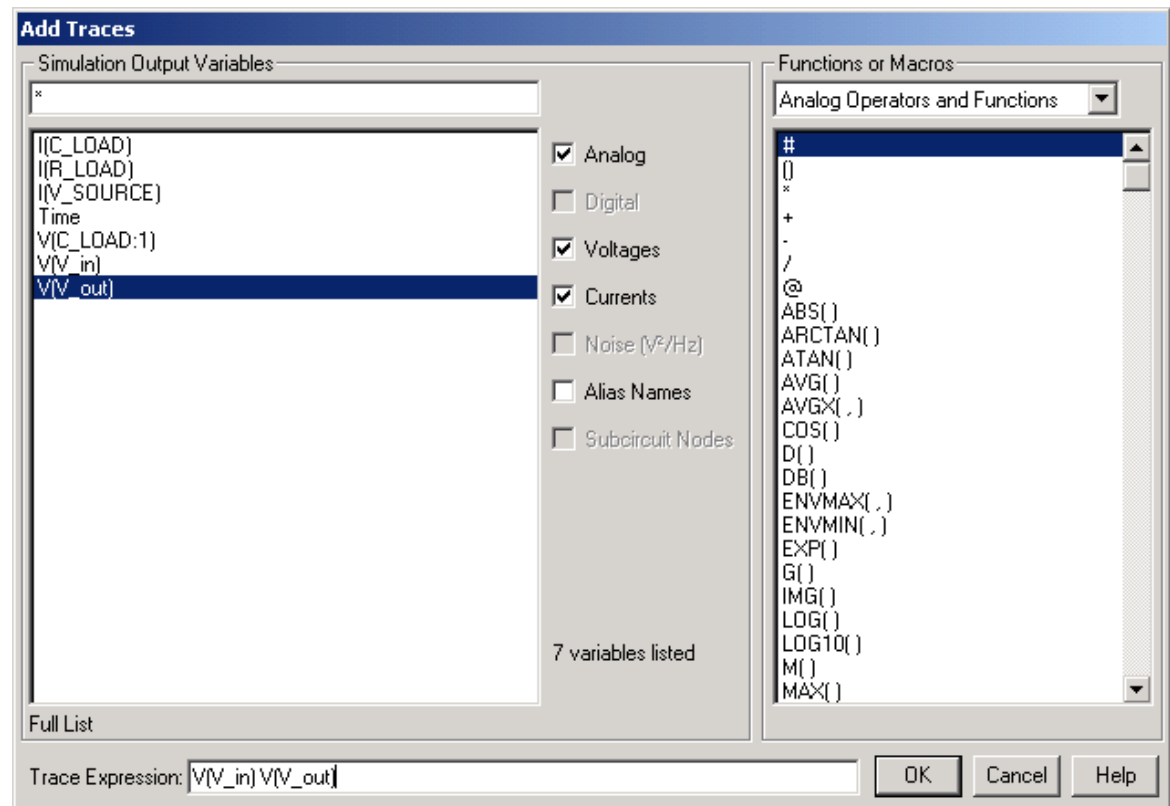


Click on the “Run Simulation” button to start the circuit simulation.

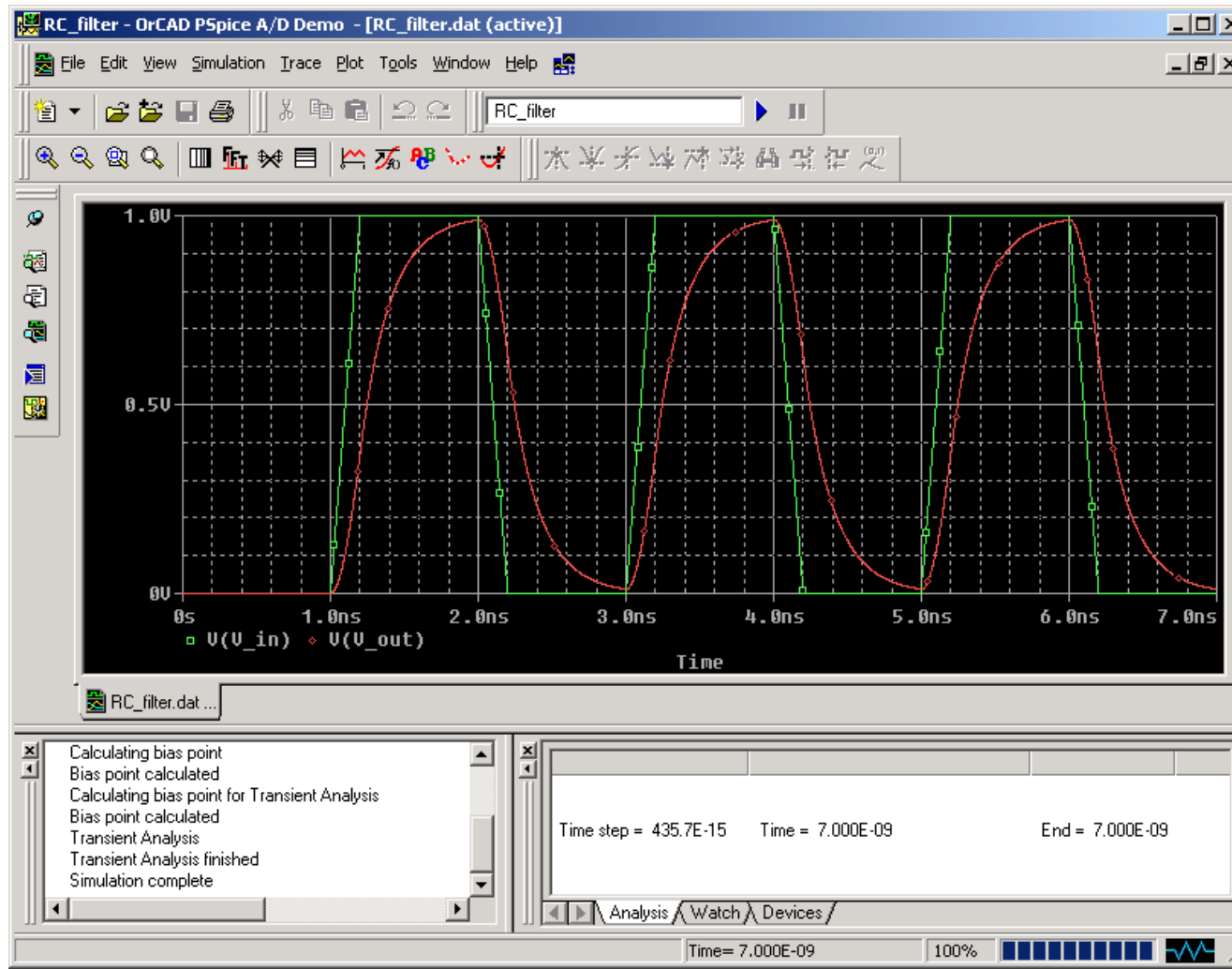


Displaying Simulation Results

- To display simulation results (voltages and or currents versus time at specific probe points), you must add the traces to the display:
- In the PSpice simulation window, go to Trace → Add Traces and choose the nodes (probe points) whose data you want to display.
- Among the available name you will find your “Bubbles” (V_in and V_out). Select them and close the “Add Traces” window.

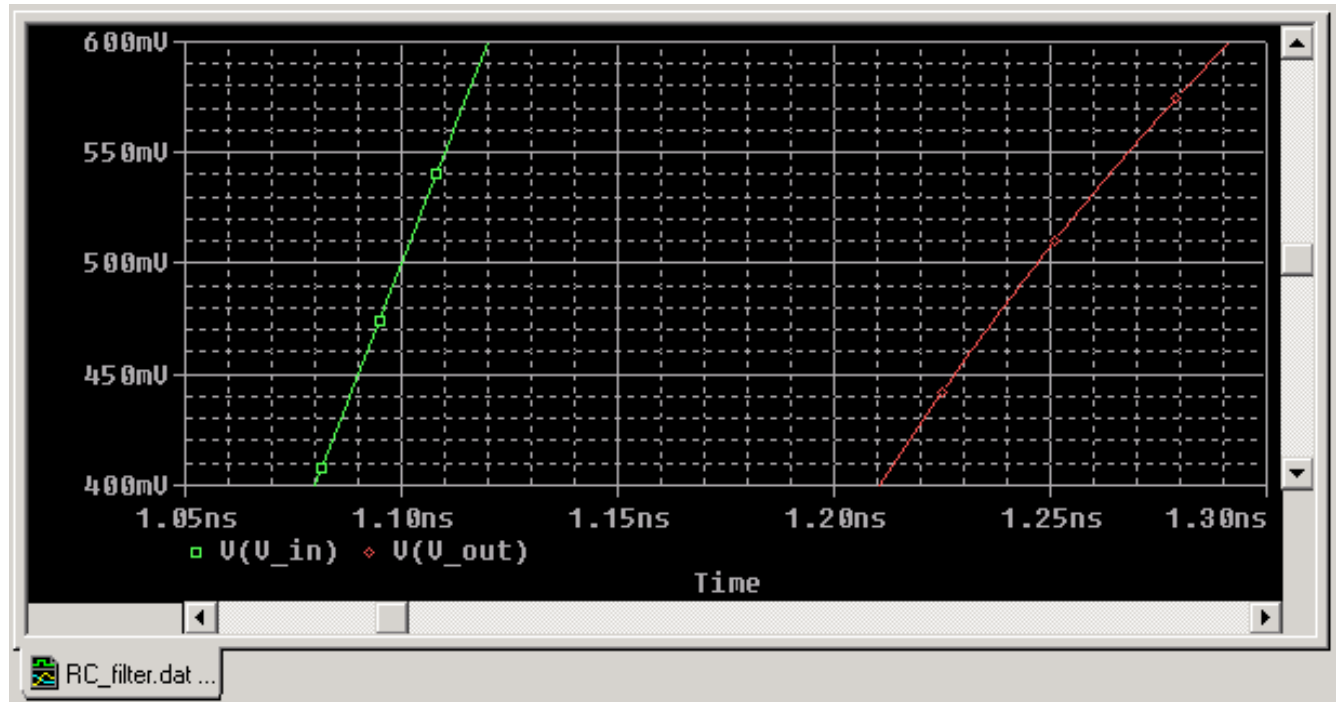


Displaying Simulation Results



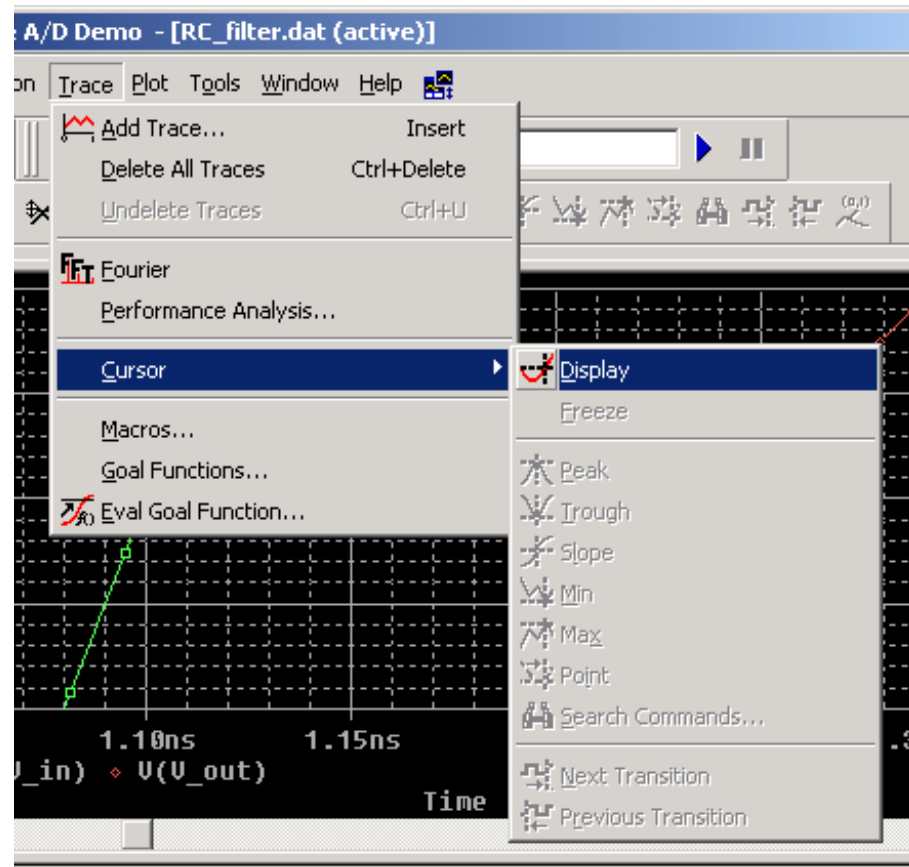
Zooming

- To zoom in onto a specific part of the plot (and to zoom out again), you can use the zoom buttons on the tool bar.
- As an alternative, you can click on the x- and y-axis labels and enter a zoom range numerically.
- Zoom in onto the mid-point (50% point) of the first rising edge in the graph (you want to determine the delay between input and output signal).



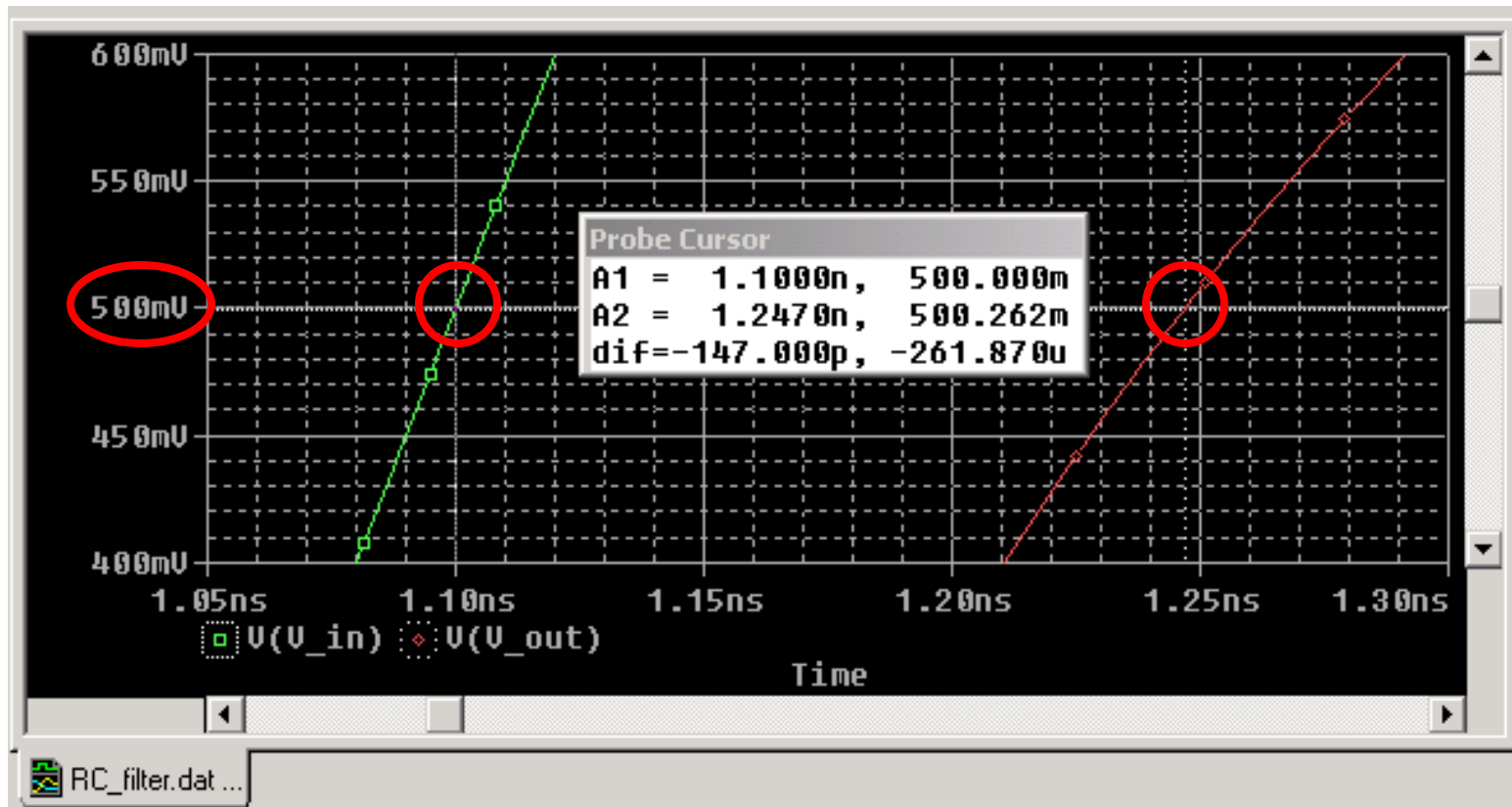
Using Cursors

- Cursors can be used to determine specific locations in the graphs, or to get time and voltage/current differences between two points.
- To display the cursors, choose Trace → Cursor → Display in the menu.
- Press the left mouse button to drag cursor 1, and the right mouse button to drag cursor 2.
- To assign a cursor to a specific trace, click on the legend symbol for that trace (on the bottom of the graph) – again, left button for cursor 1, right button for cursor 2.



Measurements with Cursors

- To measure the 50% delay between input and output, place cursor 1 on the first trace, and cursor 2 to the second, and move the over the 500mV levels.



Measurements with Cursors

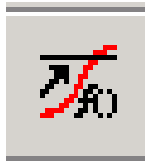
Now use the cursors to determine the following parameters:

- Input rise time (10% to 90%) $T_{rt,in}$
- Output rise time (10% to 90%) $T_{rt,out}$
- Filter delay (at 50% point, in this case at 500 mV)
- Compare the results (delay, increment in rise time) with the theoretical formulas
- If there are differences, where do they come from?

Goal Functions

- PSpice provides a set of functions (called “Goal Functions”) to determine important trace parameters like rise times, delays, etc. They are faster to use than zooming in & cursors if you need exact results.
- Click on the “Eval Goal Function” button (or choose Trace → Eval Goal Function from the menu).
- Then click on the desired function and on the name of the trace it should work with.
- Some functions need more parameters, e.g. XatNthY (determines the X positions (usually time) at which the trace crosses a given threshold the N-th time – very useful for timing measurements)

Goal Functions



Evaluate Goal Function(s)

Simulation Output Variables

Time
V(C_LOAD:1)
V(V_in)
V(V_out)

Analog
 Digital
 Voltages
 Currents
 Noise [V²/Hz]
 Alias Names
 Subcircuit Nodes

4 variables listed

Full List

Trace Expression: `Risetime(V(V_out))`


Functions or Macros

Goal Functions

- Bandwidth(1,db_level)
- BPBW(1,db_level)
- CenterFreq(1,db_level)
- Falltime(1)
- GainMargin(1,2)
- GenFall(1)
- GenRise(1)
- HPBW(1,db_level)
- LPBW(1,db_level)
- Max(1)
- MAXr(1,begin_x,end_x)
- Min(1)
- MINr(1,begin_x,end_x)
- Overshoot(1)
- Peak(1,n_occur)
- Period(1)
- PhaseMargin(1,2)
- Pulsewidth(1)
- Risetime(1)**
- SWINGr(1,begin_x,end_x)
- TPmW2(1,Period)
- XatNthY(1,Y_value,n_occur)
- XatNthYn(1,Y_value,n_occur)
- XatNthYp(1,Y_value,n_occur)

OK Cancel Help

Goal Function Value

 Risetime(V(V_out)) = 4.53152e-010

OK

Goal Functions

- In Schematics, change the capacitance of the filter from 4pF to 6pF.
- Re-Simulate the circuit.
- Using the goal function “XAtNthY” determine the delays between input and output for the first three rising and falling edges.
- Where does the variation in timing come from? What would happen with the second rising edge if one removes the first pulse (i.e. first rising and falling edge)?

(This effect is called “pattern dependent timing error”)

Hierarchical Schematics (Blocks)

- Some sub-circuits appear several times in a circuit, or the same sub-circuit is used in several different circuits. Example: vias in a DIB design, or drivers/comparators in a tester.
- Instead of re-entering these sub-circuits every time (or using cut and paste to duplicate them), one can create them once as a so-called “Block” and then simply include them wherever necessary.
- Connection in and out of the block is done using the “Interface” element.
- Apart from the interface elements, blocks are created and saved like any other schematic circuit drawing.

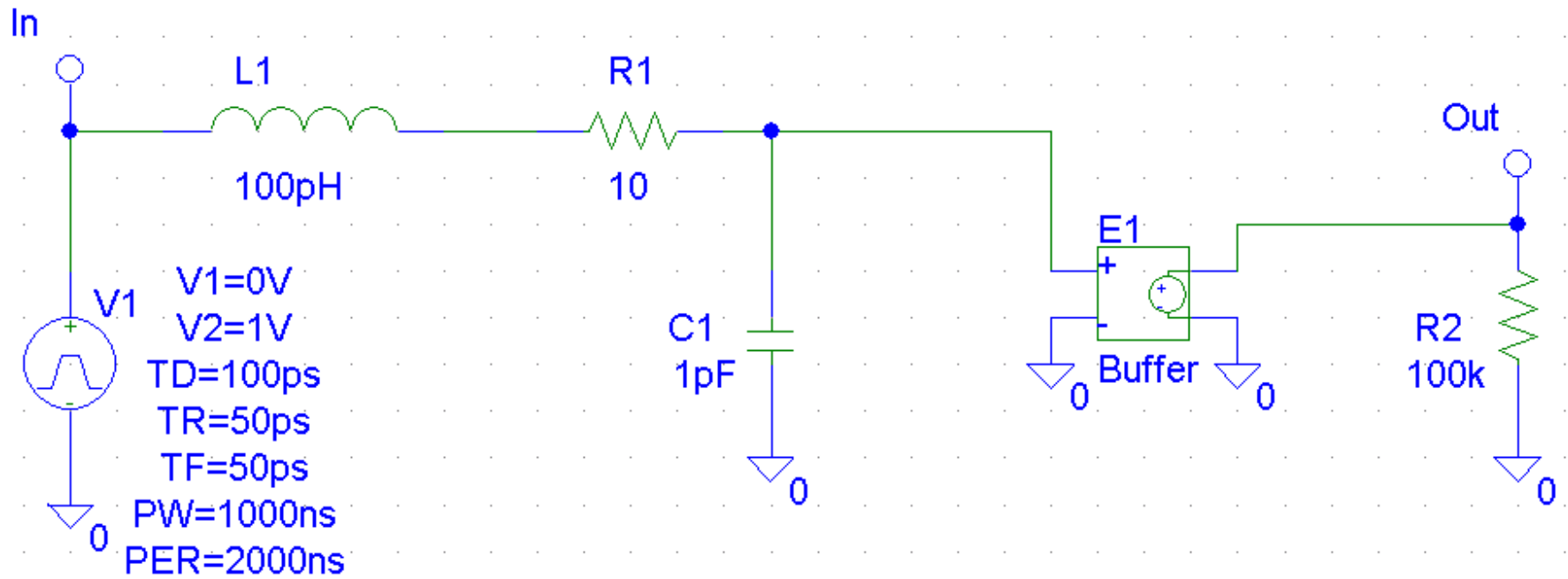
Hierarchical Schematics (cont.)

- Create the circuit shown on the next page (a more realistic edge source that has rounded edges and some overshoot) and simulate it (simulation parameters: 0 to 300 ps, step ceiling 1 ps).

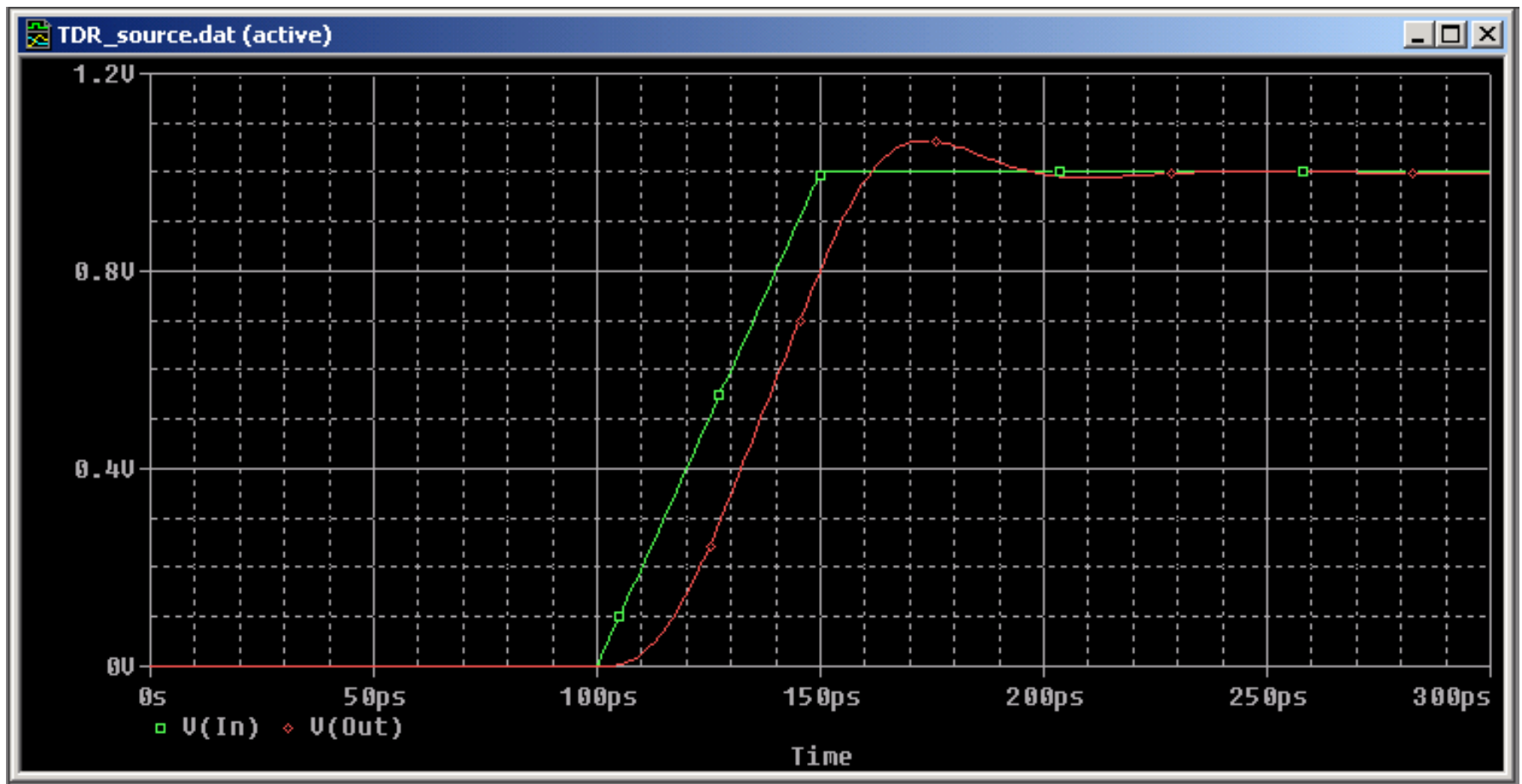
Hints:

- “E” is a voltage controlled voltage source (i.e. an ideal, linear operation amplifier); use gain=1 (default). Here it is used as a buffer to prevent the load circuitry (in this case the load resistor) from influencing the edge generation circuit.
- PSpice does not allow to have open ended circuits (device pins leading nowhere). In this case, place a high (“infinite”) resistor from the particular pin to the ground (the 100k resistor in the schematic).

Hierarchical Schematics (cont.)



Hierarchical Schematics (cont.)

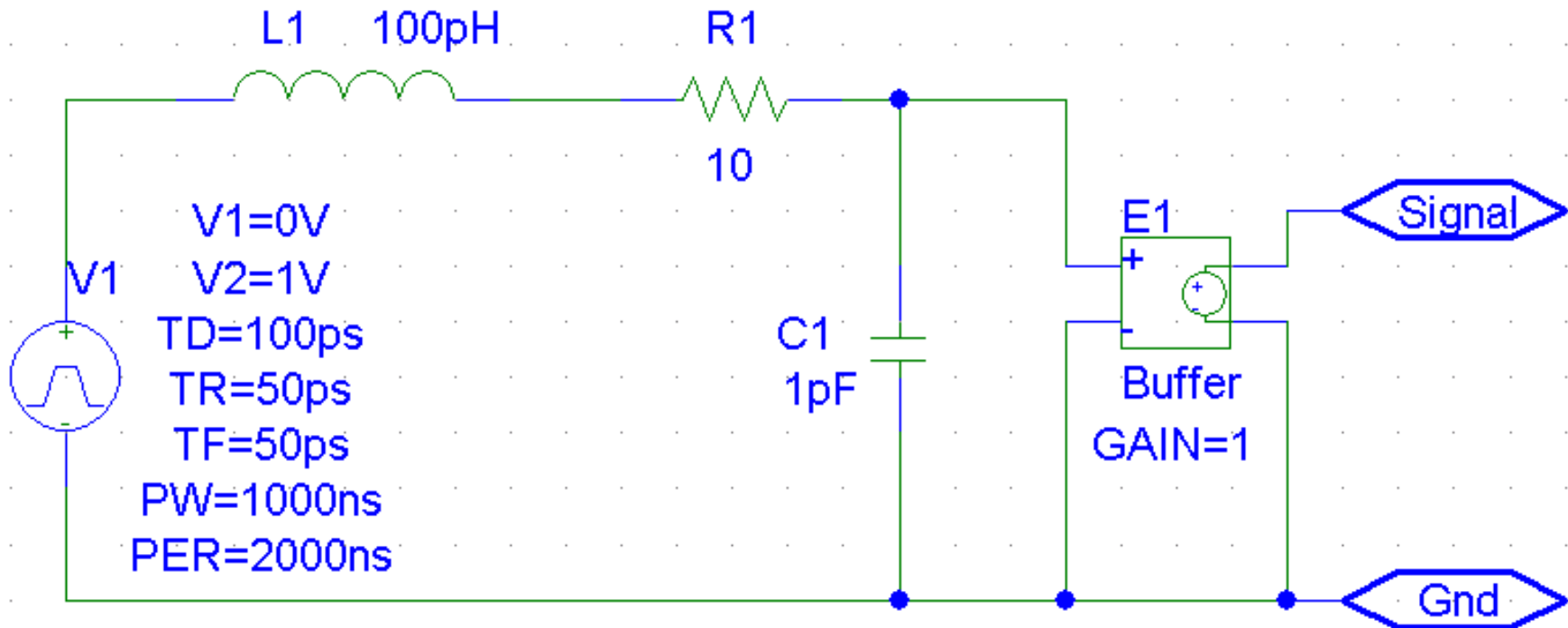


Hierarchical Schematics (cont.)

Modify the circuit as shown on the next page:

- Remove the Bubbles and the dummy load resistor.
- Add two “INTERFACE” Elements, one (named “Out”) instead of the “Out” bubble, and the other one (“Gnd”) instead of the analog ground (this allows to have the circuit floating at some DC level).
- Save the schematic under a new name.

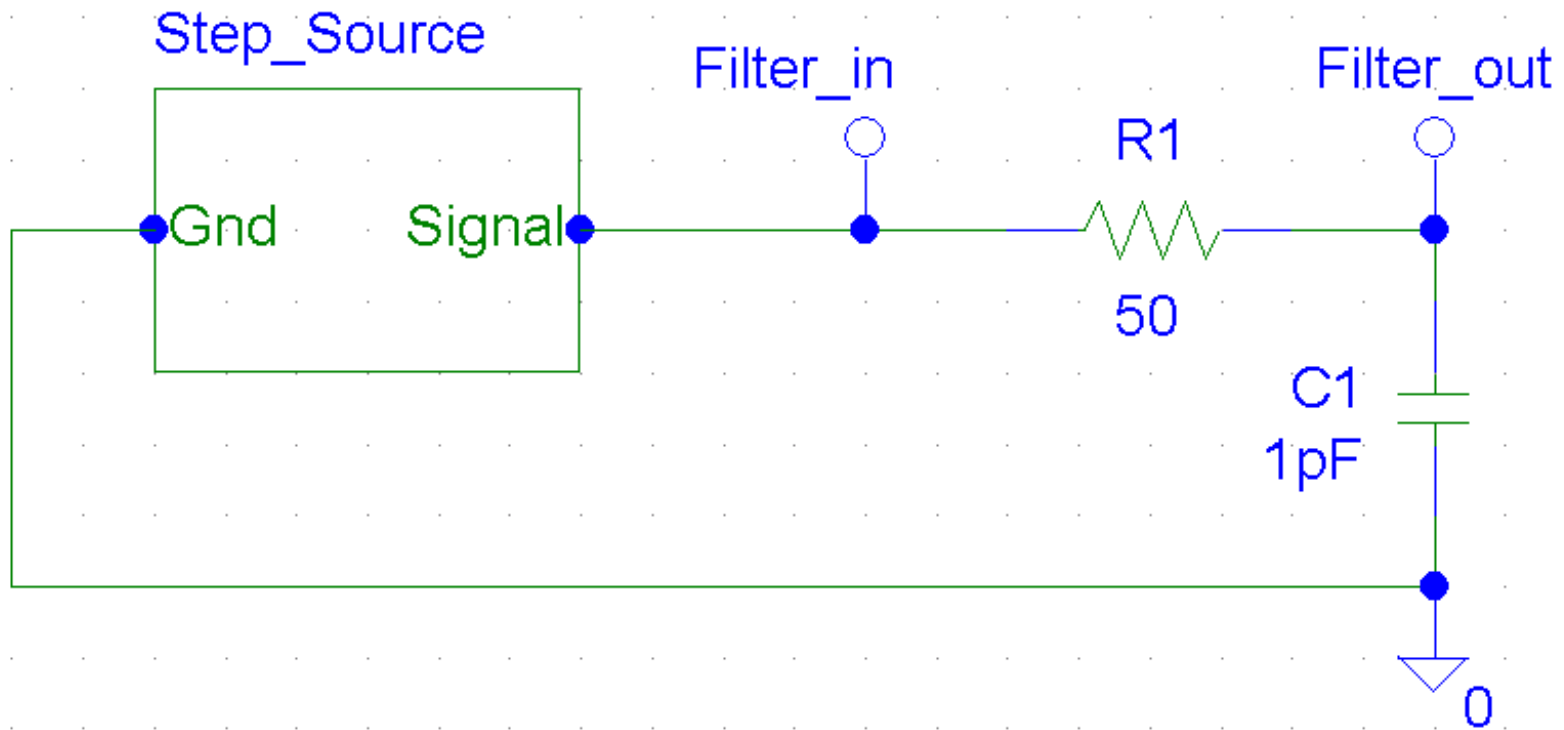
Hierarchical Schematics (cont.)



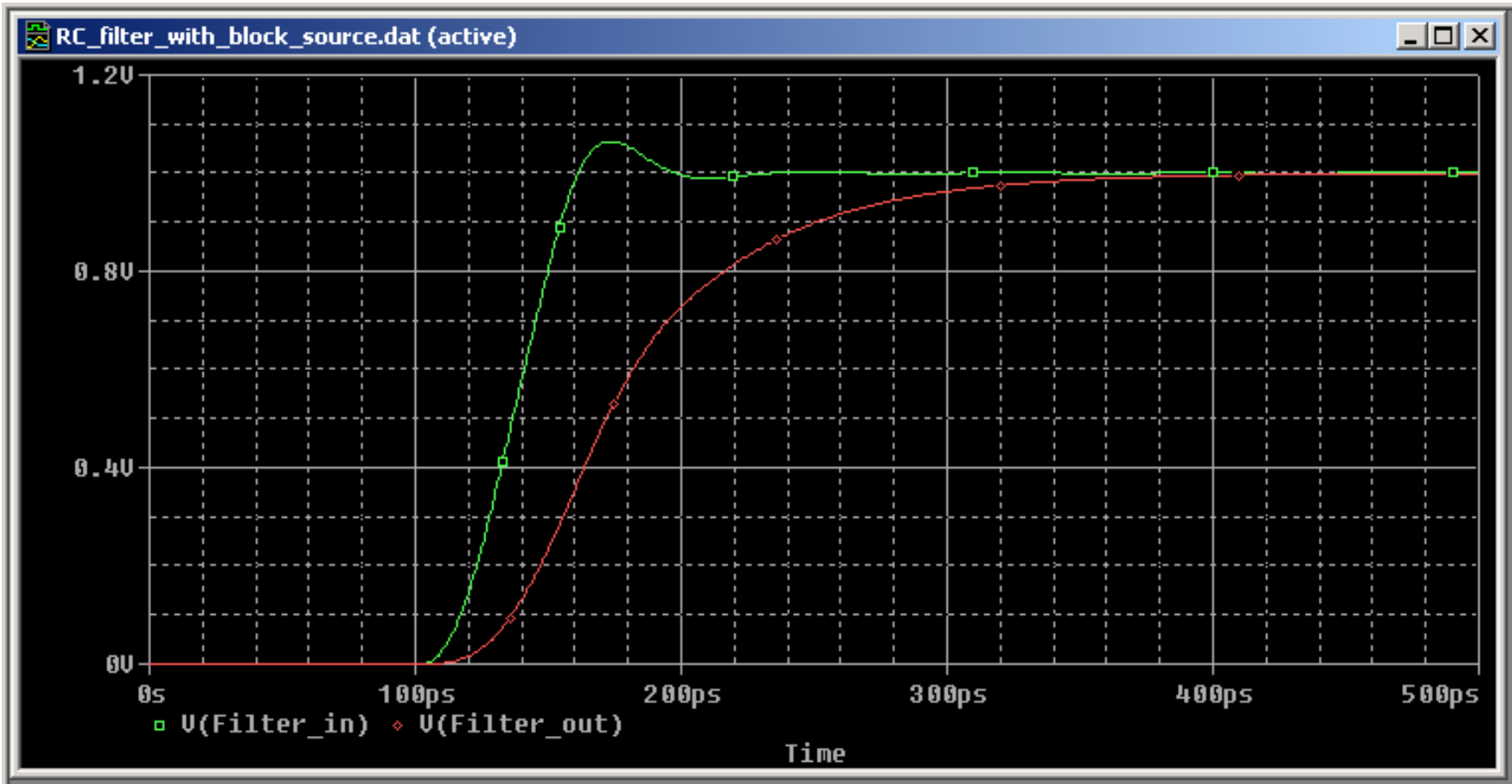
Hierarchical Schematics (cont.)

- Open a new schematic.
- Create an RC filter ($R = 50 \text{ Ohm}$, $C = 1 \text{ pF}$) with bubbles at input and output.
- Add a block (go to “Draw” → “Block”)
- Save the new schematic.
- Double-Click on the block and choose the pulse source file from before as the circuit to use. You will end up in this drawing.
- You can navigate between the filter circuit and the pulse source subcircuit via the “Window” menu.
- Connect the two block interface points to ground and the filter input, respectively.
- Set up a transient simulation (0 to 500 ps, step ceiling 1 ps) and simulate the circuit.

Hierarchical Schematics (cont.)



Hierarchical Schematics (cont.)



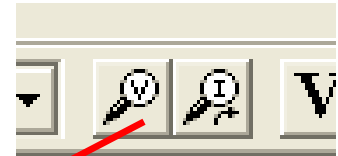
AC analysis

- For digital system simulation, the best view is usually the time domain (edge placements, waveforms, crosstalk, ground bounce, etc.) as it shows directly the resulting signals.
- For general characterization of the quality of transmission paths, for decoupling and filtering, and so on, frequency domain analysis (transmission amplitude/resistance/phase shift versus frequency) often gives a better high-level view.
- From a theoretical point of view, the two methods are equivalent and connected via the Fourier Transformation. Many network analyzers (which do frequency domain measurements) have built-in FFT to display the results either in frequency domain or in time domain (TDR).

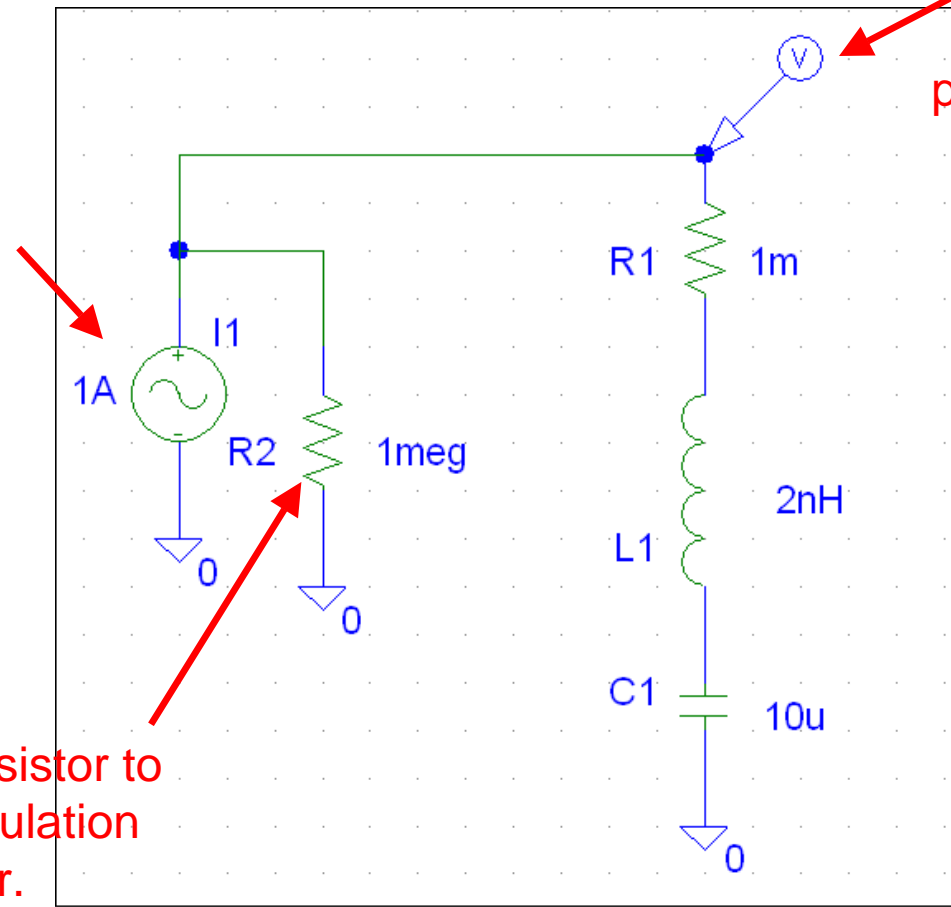
AC analysis – capacitor model

- Real capacitors are not ideal and have parasitic inductance and ohmic resistance in series with the capacitance.
- The ohmic resistance limits the minimum capacitor impedance
- Below the self resonance point, the capacitor acts like a capacitance (i.e. impedance decreases with increasing frequency).
- Above the self resonance point, the capacitor acts like an inductor (i.e. impedance increases with increasing frequency).
- Next slide: PSpice model of realistic capacitor.

AC analysis (cont.)



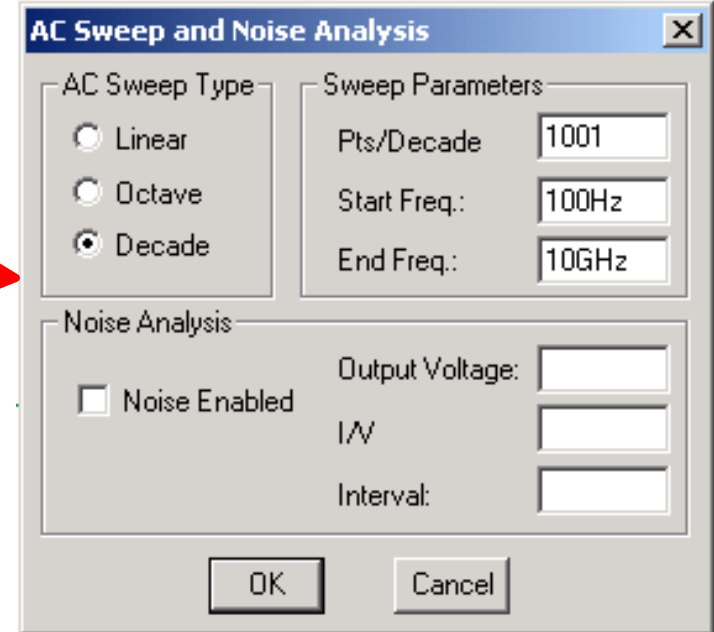
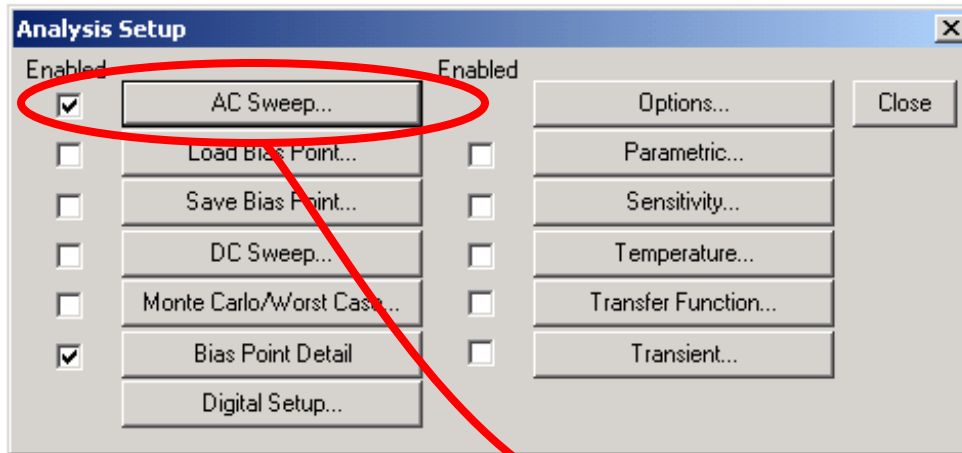
AC current source (IAC). Forcing 1A through the circuit makes the voltage across it identical to its resistance (since $V=I \cdot R$!)



Place a voltage probe to display the signal at a node automatically

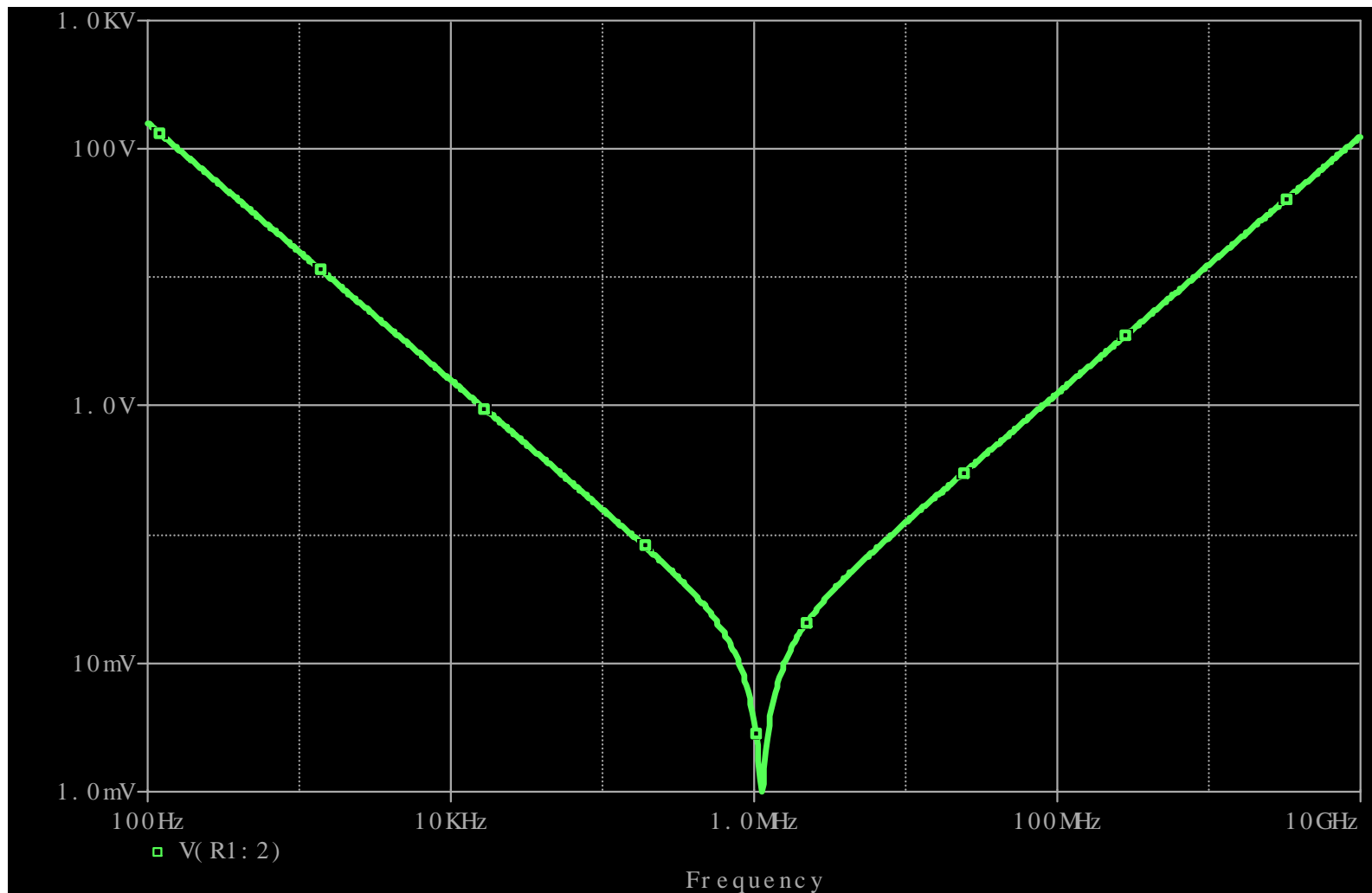
Dummy resistor to avoid simulation error.

AC analysis (cont.)



- Enter the circuit diagram from the previous page.
- Set the output current of IAC to 1A.
- Start simulation. With the voltage probe placed, the graph on the next page should come up automatically.
- In the plot, choose “logarithmic” for the y-axis (double-click on y-axis and select “log”).

AC analysis – capacitor model



That's it for now!

See you at the seminar!