

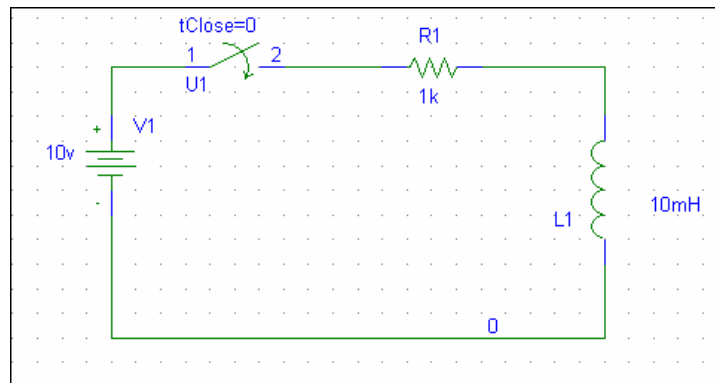
PSpice Tutorial Part Three: RL Series Transient Circuit

By Alex Barnett

IEEE @ The Illinois Institute of Technology Fall 2008

This tutorial may be more useful after reading PSpice Tutorial Part One: DC Voltage Divider and PSpice Tutorial Part Two: AC Low Pass Filter.

Start by opening a new Schematic. The Sw_tclose switch will be needed for the following circuit. Sw_tclose and Sw_topen are the most visually intuitive method of analyzing transient circuits. PSpice also allows transient analysis based on VPulse, which is explained in OrCAD Pspice with Circuit Analysis by Franz J Monssen. Using the text-field or part browser, build the circuit in Schematic 1. Notice that the inductor L1 is oriented with bumps into the circuit. This will be useful for proving a point later.



Schematic 1: Series RL circuit.

Next it is a good idea to examine the parameters of our switch. Double click the switch, and set up the following parameters: ttran=0, tClose=0, Ropen=1Meg, Rclosed=0.01.

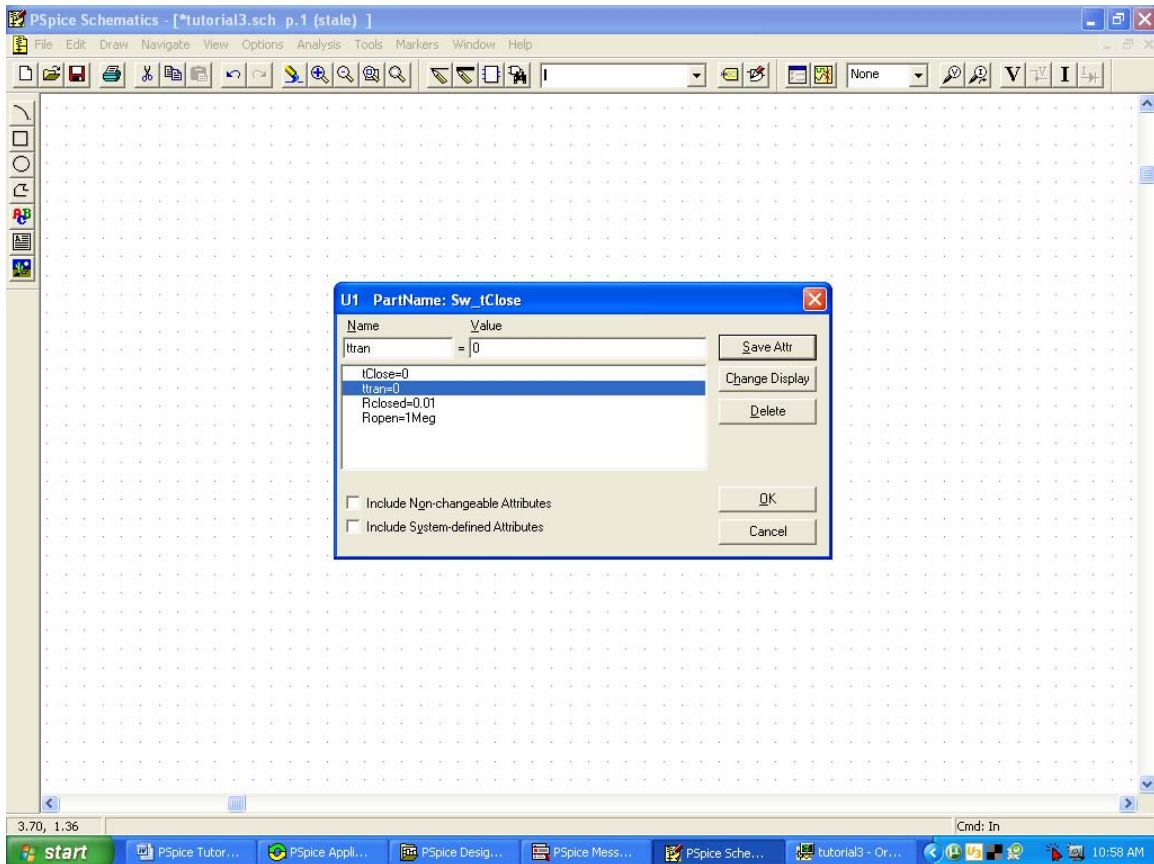


Figure 1: sw_tclose attributes

“Tclose” allows us to close the circuit at any time t . Alternate switch times could prove useful for multiple transient circuits, or to allow an initial steady-state analysis. “ttran” refers to the transitional time between the switches value of R open and R closed. 1Meg Ohm and 0.01 Ohm are sufficiently ideal for our current simulation, but may need to be adjusted for circuits of different resistances.

In setting up our Transient analysis, it is a good idea to at least approximate the time constant τ to get an idea of scale. It is usually a safe bet to have a print step of $1/10^{\text{th}}$ τ and a final time of 50 times τ . For our circuit, τ is easily calculated as $L/R=10$ microseconds.

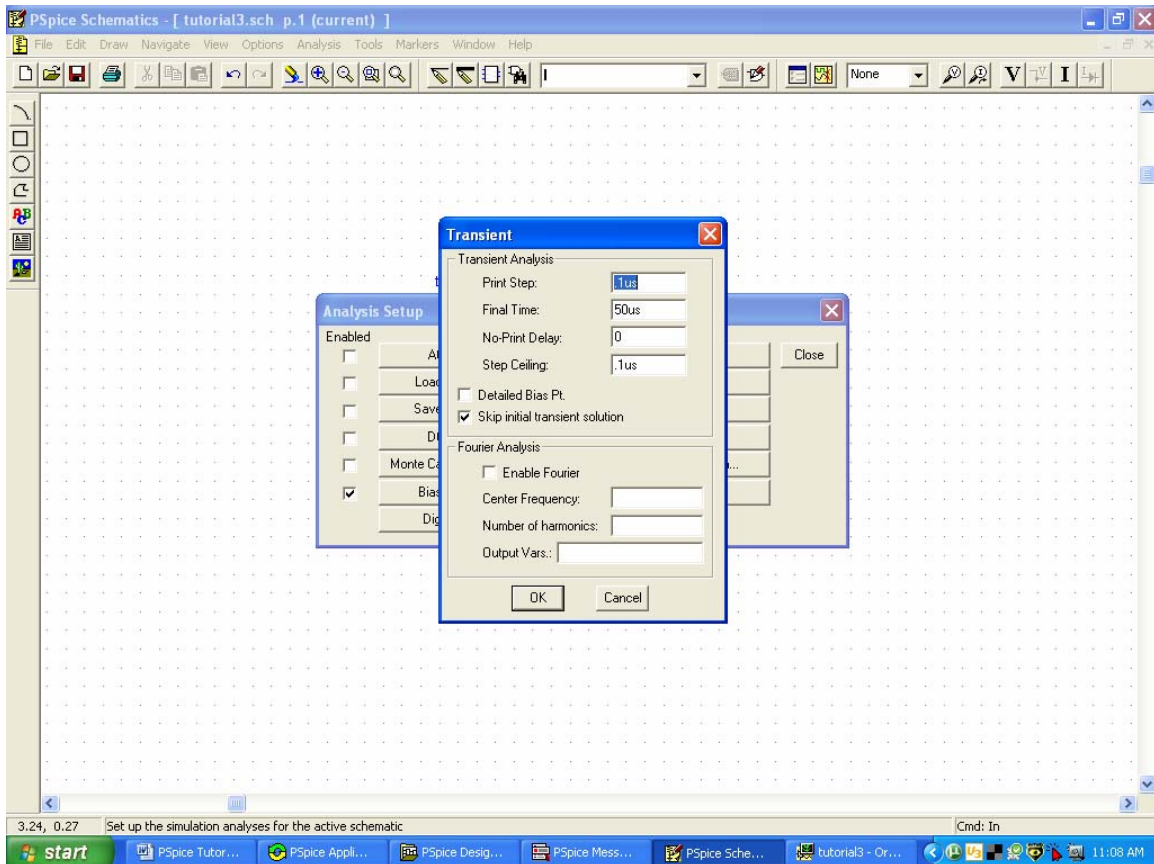


Figure 2: Transient Analysis Setup

Notice that in my setup, I have checked the box “skip initial transient solution.” This has proven necessary for me in capacitor transient circuits. My understanding is that PSpice enters simple initial values for the capacitor if you do not check this box. The choice of value by PSpice seems to be the same as the voltage across the capacitor as t approaches infinity.

Save your circuit and run your simulation. Plot a trace of $I(R1)$ (current through $R1$). To check our time constant, we will need a cursor. Use the *Trace>Cursor>display* to activate the cursor. Drag the cursor to $10\mu\text{s}$ and note the current displayed in the Probe Cursor Window. It should read 6.3212m , or $10(1-1/e)\text{mA}$.

Plot a second trace of $I(L1)$. Notice that $I(L1)$ is equal and opposite of $I(R1)$. This is due to the PSpice current convention. Any 2-pin PSpice component will appear upon selection with its pin 1 on the left and pin 2 on the right. Any current measured though a component will be measured with the direction going from pin 1 to pin 2 taken as the positive. This convention must be kept in mind when analyzing current results.

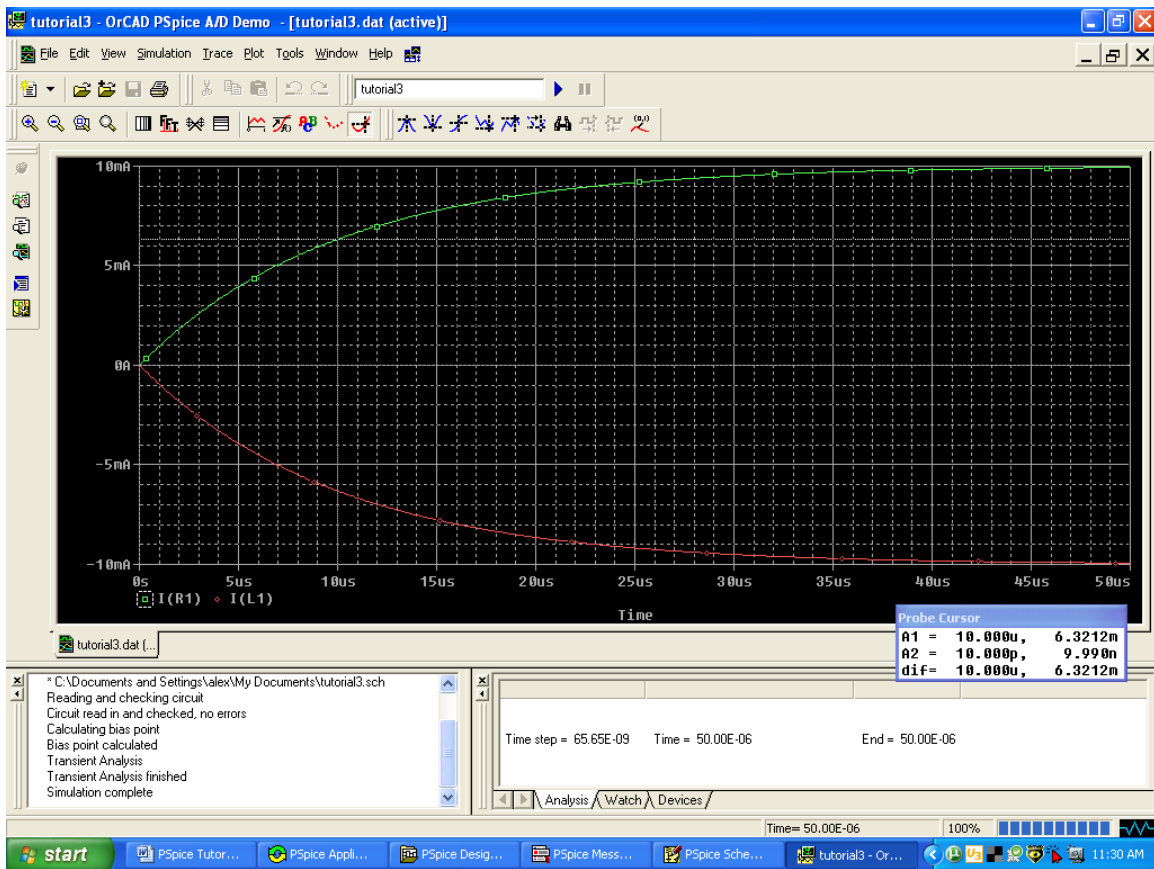


Figure 3: Plots for our Transient Circuit

This concludes my tutorial. I hope you will find PSpice to be a useful learning tool throughout your Academic and Professional Career.