

EXPERIMENT NUMBER 10

TRANSIENT ANALYSIS USING PSPICE

Objective: To learn to use a circuit simulator package for plotting the response of a circuit in the time domain.

Preliminary:

Revise Chap 8 in textbook/lecture notes on transient properties like rise time, fall time etc. The specific preliminary exercises are denoted by P1 and P2 in the notes below.

Background:

Analytically, the transient response of a circuit that contains energy storage elements (capacitance and inductance) to stored energy or to a change in the input signal is determined by the solution of a differential equation. This laboratory experiment uses the computer simulation package PSpice to investigate the transient response of an RC circuit (Part 1) and RLC circuit (Part 2).

Procedure:

Part 1: Transient Response Analysis for a simple RC Circuit

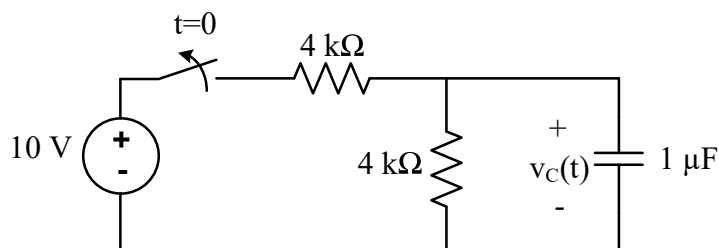





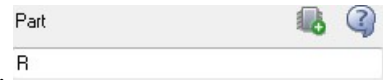
Figure 1: RC Circuit

P1: Assuming the switch in the circuit above has been closed for a long time. The switch is opened at $t=0$. Calculate the initial voltage $v_C(0)$ across the capacitor. Calculate the time constant, $\tau=RC$, for the circuit.

Aim 1: Use PSPICE to obtain the transient response for the voltage across the capacitor. Choose the final time in the Transient Analysis to be equal to five time constants (five times the value you found in P1).

1. Open the Pspice tool – start > All Programs > Cadence > OrCAD 16.5 Lite> OrCAD Capture CIS Lite (You may have to launch AppsAnywhere first, and select OrCAD to be installed on your machine before Capture CIS Lite is available).
2. Create a New Project – File> Project > New
3. Input a project name; select Analog or Mixed A/D
4. Save the project on your flash-drive.
5. Create a Blank project. If you drag the scroll down, you may see a ground already present with some explanation written underneath, you can use this ground or delete and add it later from parts list.

6. Use the “place part” tab the main program toolbar, or directly click  on the right side of the window to begin placing parts with the Part Browser pop-up window.
7. Use the Part Browser pop-up window to assist in finding and placing parts.
8. Before you add any parts, make sure necessary libraries are added by   clicking the add/delete library icon. Libraries needed for this lab include “analog”, “source” and “eval”.
9. The text input allows searching for parts.





10. Typing the part names in the text input will help you find the part. Make sure the necessary libraries are highlighted for locating parts. You can highlight all the selected libraries (left click + “ctrl” one by one, or left click + “ctrlA” to highlight all).
11. Select the part needed, click  to add the part.
12. Implement the circuit as shown in Figure 1. The parts list is shown in Table 1.
13. Remember to use the 0V GND reference from SOURCE library.
14. After all the parts are added, double click the name of the part to change the name, and double click the value of the part to change the value. You can also double-click the part symbol itself and a menu will come up to change all part attributes.
15. Add a voltage marker by clicking , which is the voltage marker, and put it at the node as shown in Figure 2. Note that a voltage marker indicates the node voltage of the marked position with respect the GND node. We want to observe the voltage across the capacitor with respect to time.

Table 1. Parts List

PSpice Part Name	Description
VDC	DC Voltage Source
GND	Ground (0 V reference)
R	Resistor
C	Capacitor
L	Inductor
Sw_tOpen	Switch that opens at time t (Library: EVAL)

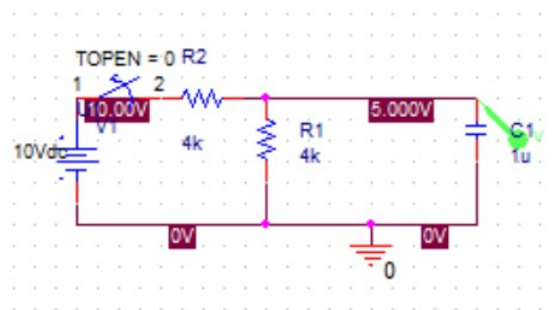


Figure 2: RC Circuit in PSPICE

16. Create a “Simulation Profile”, by going to the simulation profile settings. Click the PSpice tab and get the drop-down menu.
 - a. In the “Analysis type” set “Time Domain (Transient)”.
 - b. Run to time: set to five time constants (five times the value you found in P1). Set maximum step size to one hundredth of the stop time value.
 - c. Apply and close the window.
17. Run the simulation by clicking the “Run Pspice” tab. The simulation result will pop up, with the voltage as a function of time plot shown (Figure 2).
18. On the graph, draw a line at $0.3678 \cdot v_C(0)$. Then use the cursor command to locate the time constant. Record this value. To obtain the cursor, click on Trace->Cursor->point. Look in the trace box at the lower right hand corner for the corresponding value in time domain (X).
19. Print the graph and circuit schematic to include in your report.

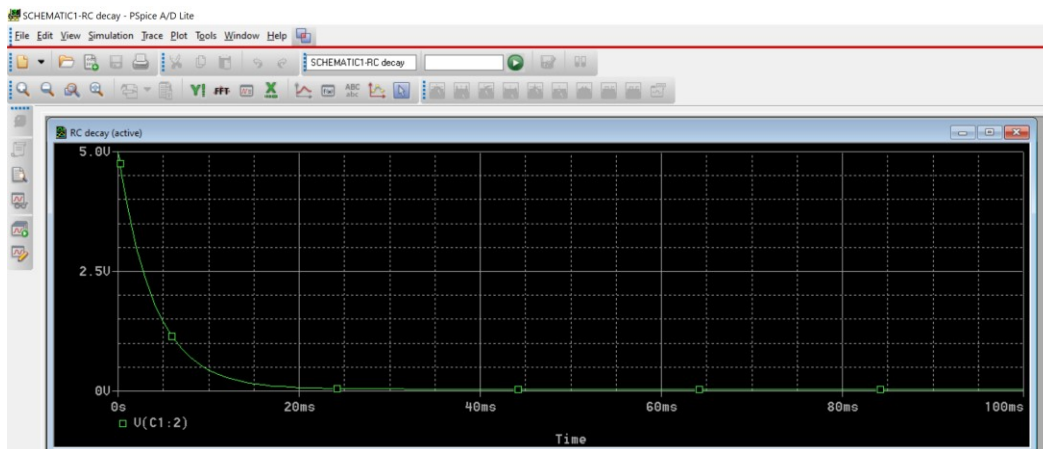


Figure 3: Voltage across capacitor

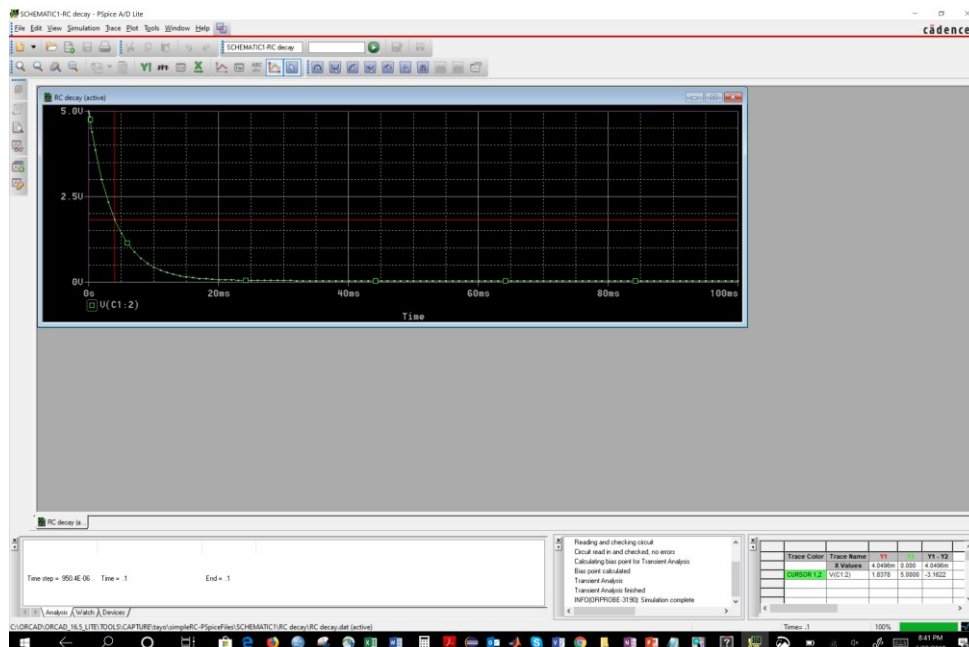


Figure 4: Cursor and trace to locate time constant

Part II: Dampening in a Series RLC Circuit

The switch in the circuit below has been closed for a long time. The switch is opened at $t=0$.

Use PSPICE to obtain the transient response for the voltage across the capacitor for $R=8.5\text{ k}\Omega$, $R=4\text{ k}\Omega$, and $R=1\text{ k}\Omega$ in the circuit in Figure 2. In the Transient Analysis use a value of 50 ms.

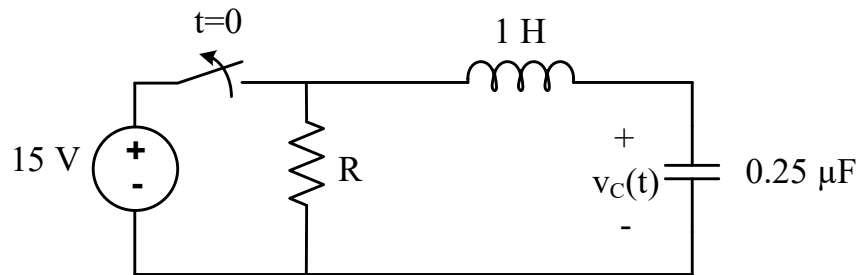


Figure 5: RLC Circuit

1. Calculate the initial voltage $v_C(0)$ across the capacitor.
2. Follow similar steps from part 1 from #1 - #15 to build the RLC circuit described in Figure 5. An additional library called “special” will be needed to add “PARAMS”.
3. For the resistor R, set the value as $\{R\}$. The curly braces signify that it is a parameter.
4. Get a part called PARAM from “special” library and place it in your circuit.
5. Double click on PARAM to edit the values for R.
 - a. Click on ‘New Column’;
 - b. Enter name: R
 - c. Enter initial value: 8.5k
 - d. Click on “Display”; select Display Format: Name and Value

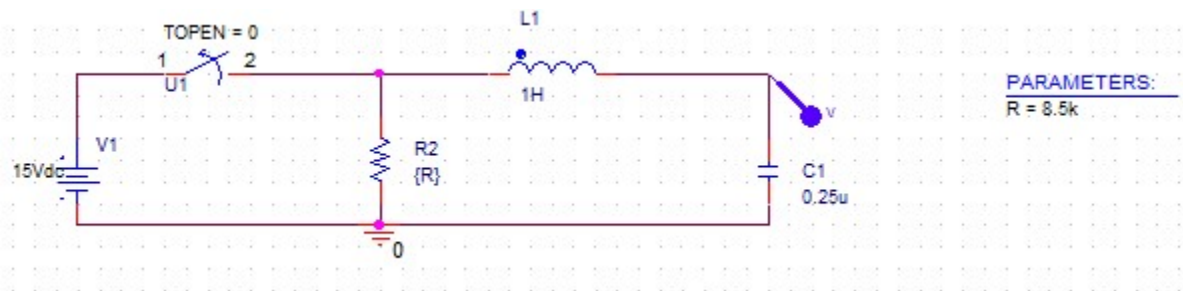
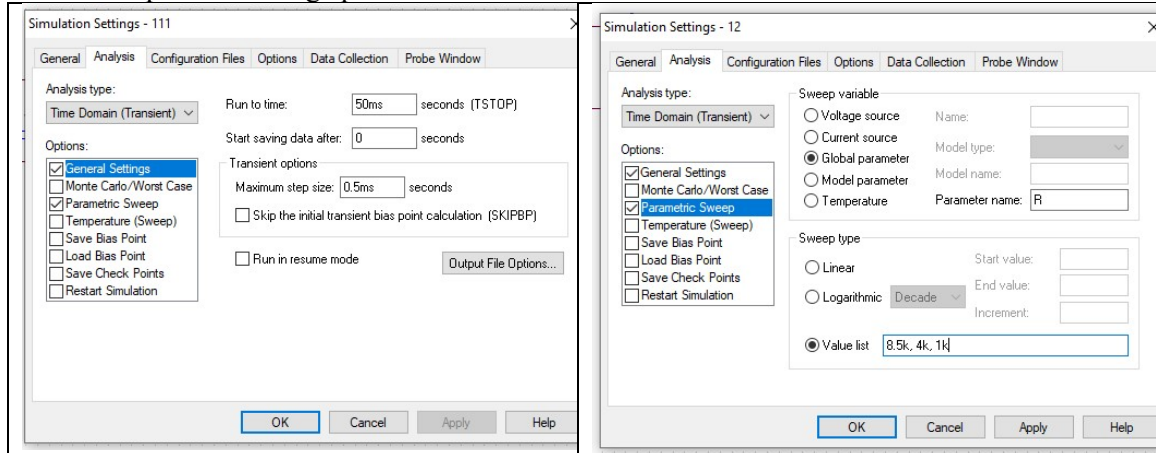


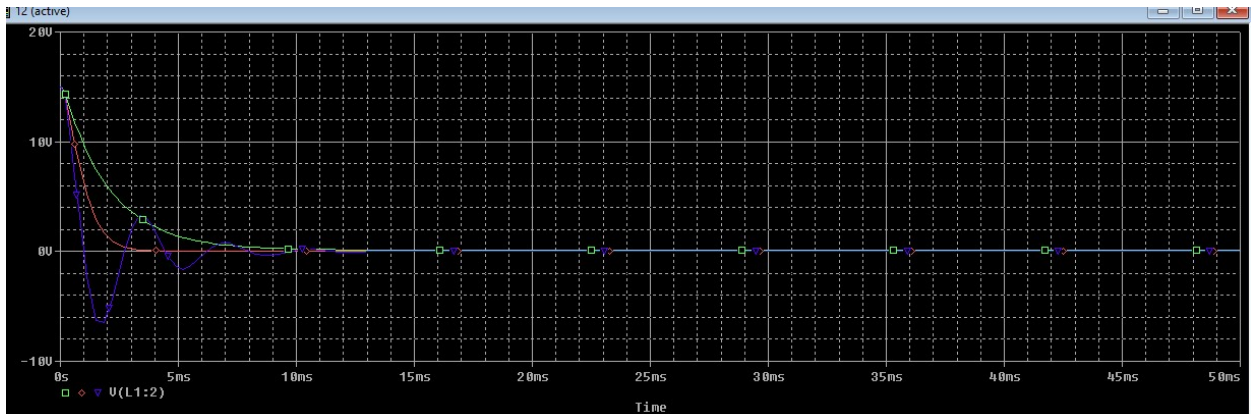
Figure 6: RLC Circuit in PSPICE

6. Create a “Simulation Profile”, by going to the simulation profile settings. Click the PSpice tab and get the drop-down menu.
 - a. In the “Analysis type” set “Time Domain (Transient)”.
 - b. Run to time: 50ms. Set maximum step size to one hundredth of the stop time value.
 - c. Check “Parametric Sweep”

- d. Under Sweep variable, select Global Parameter
 - e. Enter Parameter name: R
 - f. Under Sweep type, select Value list and enter the values for the resistor: 8.5k, 4k, 1k.
7. Run the simulation by clicking the “Run Pspice” tab. The three sections for the parameter sweep will come up on the voltage plot.



8. Print the graph and circuit schematic to include in your report.



9. Which resistor value corresponds to under dampening? Verify this by calculating the natural solution of the differential equation modeling this circuit for that resistor value.