

Department of Electrical and Electronic Engineering  
Bangladesh University of Engineering and Technology

**EEE 102: Electrical Circuits I Laboratory**

**Experiment (Simulation) # 04: First Order DC Transients and Steady State AC Circuit Analysis**

**OBJECTIVES**

1. Demonstrate the DC transient analysis of RL circuits.
2. Demonstrate the DC transient analysis of RC circuits.
3. Demonstrate the Steady State AC RLC circuit analysis.

All parts are executed by computer simulation using SPICE. Simulation is done by both creating netlist and using schematics.

**THEORY**

Energy delivered by a source can be stored in an inductor or capacitor. Once initial conditions are known, a voltage in a capacitor ( $V_0$ ) and current in an inductor ( $I_0$ ), the application of Kirchoff's laws results in the circuit equations that will yield the voltage and current relationships with time. The definitions of the current/voltage relationships are applied for the storage elements and for resistors (Ohm's Law) and again, first order ordinary differential equations result.

The solution leads to equations that define an exponential build up with time of any energy that is stored.

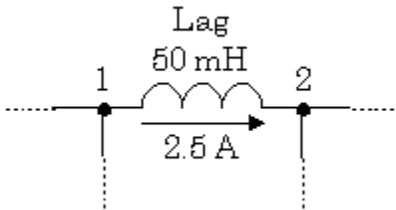
	Current	Voltage	Time Constant
Inductor	$i(t) = \frac{V_s}{R} + (I_0 - \frac{V_s}{R})e^{-(R/L)t}$ $t > 0$ $i(t) = \frac{V_s}{R}(1 - e^{-(R/L)t})$ $t > 0 \text{ Provided } I_0 = 0$	N/A	$\frac{L}{R}$
Capacitor	N/A	$v(t) = I_s R + (V_0 - I_s R)e^{-(1/RC)t}$ $t \geq 0$ $v(t) = I_s R(1 - e^{-(1/RC)t})$ $t \geq 0 \text{ Provided } V_0 = 0$	$RC$

In both instances, the growth in current and voltage is exponential with time to a final value determined by the circuit.

The time constant  $\tau$  is defined as shown to characterize the exponential growth in current/voltage in the inductor/capacitor. It does this through the factor  $e^{-t/\tau}$ . After a time interval of 5 time constants, the factor is within 1% of its final value.

## Inductors in PSpice

This part name begins with the letter, L, in column 1 of the source listing.



The figure shown above shows the circuit symbol for an inductor with node designations of "1" and "2," an initial current of 2.5 A, and a value of 50 mH. An appropriate code listing for entering this element into a PSpice circuit file is:

```
*name nodelist L_val
Lag 1 2 50m IC=2.5
```

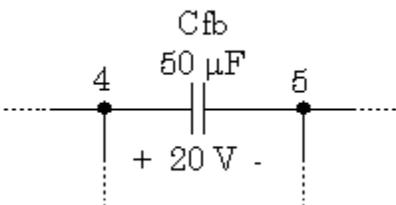
Note that the initial current is assumed to flow from the first node in the node list through the inductor towards the second node in the node list. If there is a need to change the direction of this initial current, either reverse the order of the nodes in the node list or place a minus sign in front of the value of the initial current. For better readability, the above line could be written as:

```
Lag 1 2 50mH IC=2.5A
```

The "H" for henrys and the "A" for amps will be ignored by PSpice.

## Capacitors in PSpice

The part name for a capacitor must start with the letter, C.



An appropriate code listing to represent this capacitor in a PSpice listing is:

```
*name nodelist C_val
Cfb 4 5 50u IC=20
```

The capacitance of the above element is 50 $\mu$ F. This can be represented as "50u" in PSpice. Note that the polarity of the initial voltage (as shown) is such that the positive side is the first node in the list with the negative side on the second node in the list. To reverse the polarity of the initial voltage for the simulation, either reverse the order of the nodes in the node list or place a minus sign in front of the value in the "IC=" phrase. For better clarity, the above capacitor could be coded as:

```
Cfb 4 5 50uF IC=20V
```

PSpice would ignore the "F" for farads and the "V" for volts.

### Use of the .TRAN command

This is the command that passes the user's parameters for performing the transient analysis on a circuit to the PSpice program. There are four time parameters and an instruction to use the initial condition rather than calculated bias point values for starting conditions. First, we show a sample .TRAN statement and then we will describe its parameters.

```
* prt_stp t_max prt_dly max_stp
.TRAN 20us 20ms 8ms 10us UIC
```

In the above statement, the "20us" value labeled "prt\_stp" (*print step*) is the frequency with which data is saved. In this case, the system variables are stored each 20 $\mu$ s of simulation time. The actual time steps used by PSpice may be different from this. The second parameter, "20ms," labeled as "t\_max" (*final time*) is the value of time at which the simulation will be ended. Since PSpice starts at  $t = 0$ , there will be a total of 20ms time span of simulation for the circuit. The third parameter, "8ms," labeled as "prt\_dly" (*print delay*) is the print delay time. In some cases, we do not want to store the data for the entire time span of the simulation. In our sample statement shown above, we ignore the data from the first 8ms of simulation and then store the data for the last 12ms. Most of the time, this parameter is set to zero or not used. The fourth parameter, "10us," labeled as "max\_stp" (*max step*) is the maximum time step size PSpice is allowed to take during the simulation. Since PSpice automatically adjusts its time step size during the simulation, it may increase the step size to a value greater than desirable for displaying the data. When the variables are changing rapidly, PSpice shortens the step size, and when the variables change more slowly, it increases the step size. Use of this parameter is optional. The last parameter in our list is "UIC." It is an acronym for "Use Initial Conditions." Unless you include this parameter, PSpice will ignore the initial conditions you set for your inductors and capacitors and will use its own calculated bias point information instead.

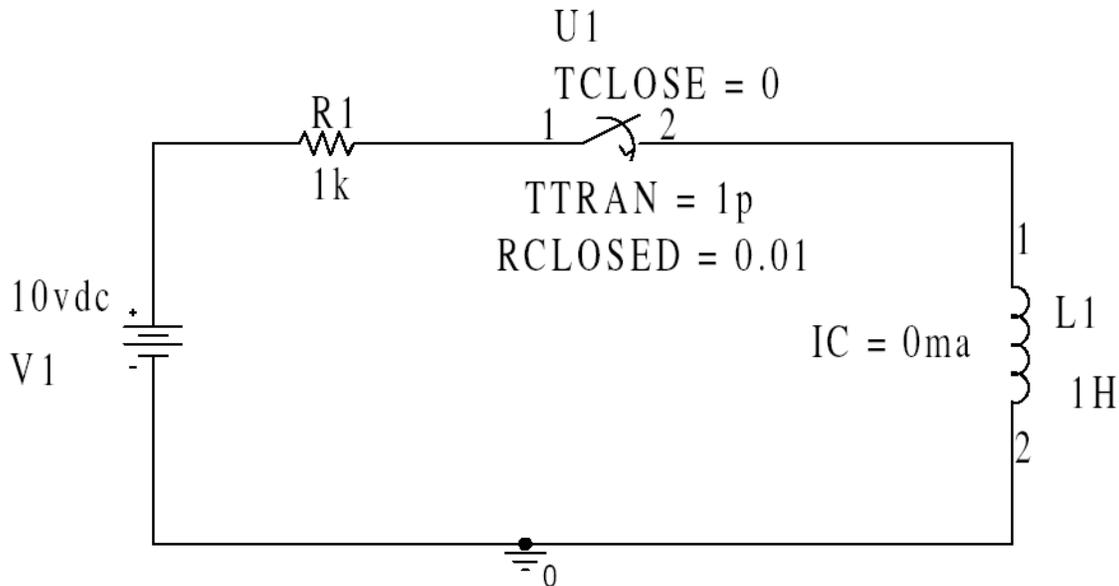
## PROCEDURE

### Part 1:

- 1.1. **Construct** the circuit shown in Figure 4.1 using both NETLIST and Schematic. Set the initial current in the inductor to zero.
- 1.2. **Setup** your PSPICE simulation to perform a transient analysis over a suitably chosen interval of time,  $5\tau$  the time constant.
- 1.3. **Calculate** the value of the time constant. Make sure you insert units!

$$\tau = \underline{\hspace{2cm}}$$

- 1.4. **Estimate**  $\tau$  from the plot of the inductor current and compare with the calculated value.
- 1.5. **Show** graphs of voltage, current, power and energy against time for the inductor L1. **Print** your results and attach them to your report.
- 1.6. **Change** L1 to 1mH and repeat above procedure.



**FIGURE 4.1**

- 1.7. **Set** the initial current in the inductor to 5mA. **Run** your PSPICE simulation. **Print** the trace for the current in the inductor. What is the final value of the inductor current?

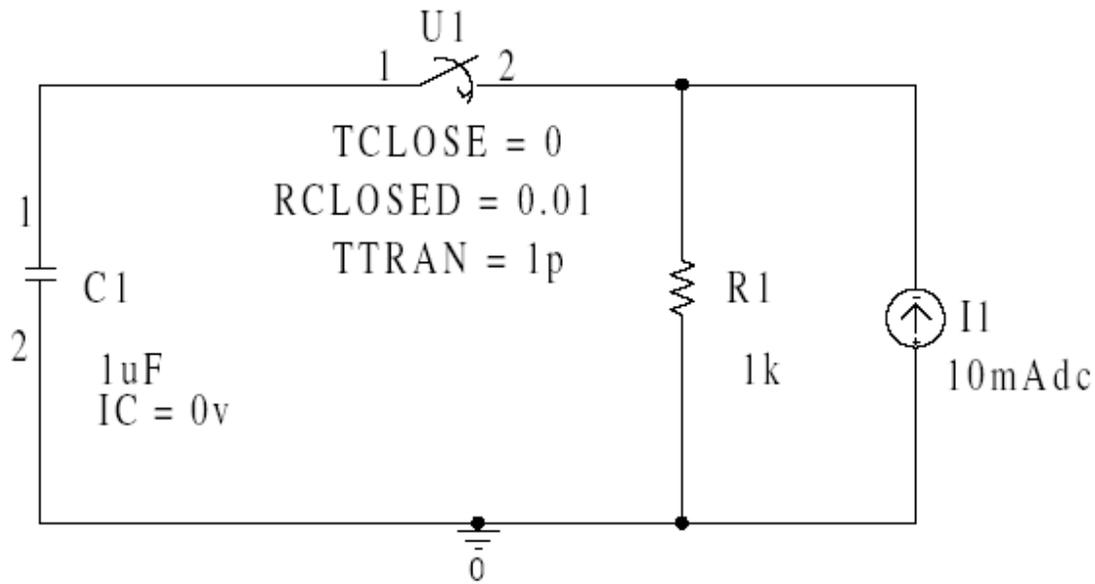
$$\text{Final Inductor Current} = \underline{\hspace{2cm}}$$

- 1.8. **Measure** how long it takes for the current to rise to 90% of its final value. **Calculate** this time with the equation provided. **Compare** the two results and calculate the % difference. **Show** your work.

## Part 2:

- 2.1. **Construct** the circuit shown in Figure 4.2 using both NETLIST and Schematic. Set the initial voltage across the capacitor to zero.
- 2.2. **Setup** your PSPICE simulation to perform a transient analysis over a suitably chosen interval of time,  $5\tau$  the time constant.
- 2.3. **Calculate** the value of the time constant. Make sure you insert units!

$$\tau = \underline{\hspace{2cm}}$$



**FIGURE 4.2**

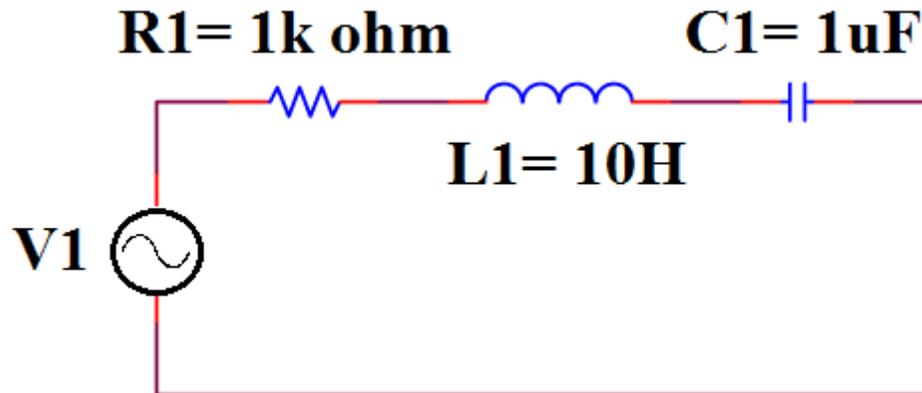
- 2.4. **Estimate**  $\tau$  from the plot of the capacitor voltage and compare with the calculated value.
- 2.5. **Show** graphs of voltage, current, power and energy against time for the capacitor, C1. **Print** your results and attach them to your report.
- 2.6. **Change** C1 to 10 $\mu$ F and repeat above procedure.
- 2.7. **Set** the initial voltage on the capacitor to 5Volts. **Run** your PSPICE simulation. **Print** the trace for the voltage in the capacitor. What is the final value of the capacitor voltage?

$$\text{Final capacitor voltage} = \underline{\hspace{2cm}}$$

- 2.8. **Measure** how long it takes for the voltage to rise to 90% of its final value. **Calculate** this time with the equation provided. **Compare** the two results and calculate the % difference. **Show** your work.

### Part 3:

Given the RLC series circuit shown in fig 4.3



**FIGURE 4.3**

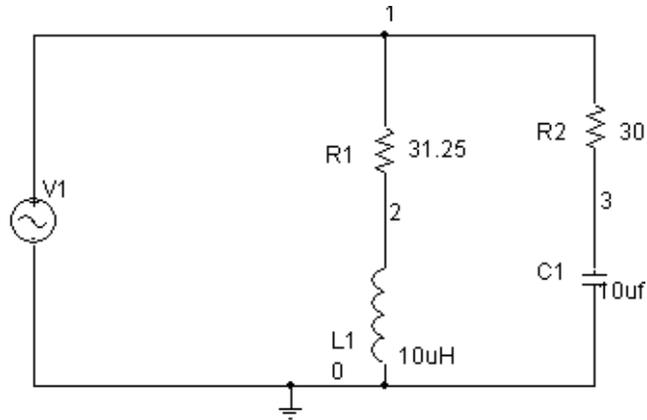
- 3.1. **Construct** the circuit in Figure 4.3 for both netlist and schematic.
- 3.2. **Setup** your PSPICE simulation to perform a transient analysis over a suitably chosen interval of time. Take V1 to be of Magnitude 100V. Choose a suitable frequency.
- 3.3. **Observe** the waveshapes of currents through and voltages across through the resistor, inductor and capacitor and mention their phase relationship.
- 3.4. **Observe** the mentioned waveshapes for the series resonance condition.

#### \*Schematics Netlist \*

```
V_V1      1 0 DC 0 AC 0 SIN 0 100v 1k 0 0 0
C_C1      3 0 1uf
R_R1      1 2 1k
L_L1      2 3 10h
.tran 20us 20ms 0 10us
.probe
.END
```

### REPORT

1. How final value of current and voltage and the time to reach 90% of the final value in RL and RC circuits respectively depends on the circuit parameters? Explain your answer.
2. In same plot, show the curves of voltage in RC circuits for various time constant. Explain it.



**FIGURE 4.4**

3.
  - a) Construct the circuit in Figure 4.4 for both NETLIST and schematic. Take V1 to be of Magnitude 100V. Choose a suitable frequency.
  - b) Setup your PSPICE simulation to perform a transient analysis over a suitably chosen interval of time.
  - c) Observe the waveshapes of currents through and voltages across the resistor R1 & inductor L1 and Resistor R2 & capacitor C1 and mention their phase relationship.