Lab 8: LTSpice – RC, RL, and RLC Circuits

GOALS:

- Simulate and analyze transient response in RC, RL, RLC circuits
- Understand voltage/current behavior for L and C in response to change in circuit state

Lab report:

1. Introduce justification for experiment.
   a. Analyze transient reponse of RC, RL, and RLC circuits via simulation
2. Properly label and document simulation schematics and simulation results
   a. Label components, interesting nodes
   b. Label simulation output plots
   c. Label specified values (steady state, rise/fall)

You will need to RECORD all of your data independently.

The simulation data required for your lab report are listed in black boxes like this throughout the following parts.

FOLLOW ALL STEPS AND INCLUDE ALL REQUIRED DATA IN YOUR REPORT!

<u>Introduction</u>: RC, RL, RLC Transient Response

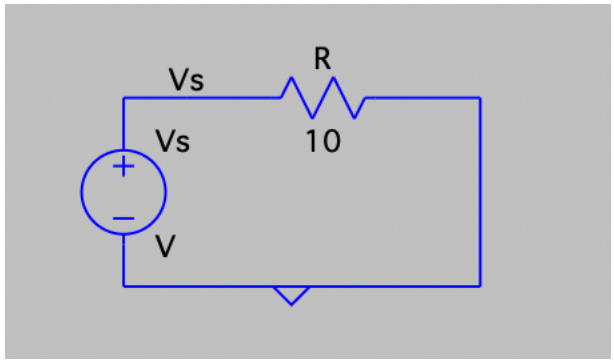
Most of our circuit analysis so far has focused on resistive circuits (only resistors and DC voltage/current sources). In these circuits, we could "easily" find any voltage/current at any time t using basic techniques like Ohm's Law (V=IR), KVL/KCL, etc. In other words, the voltages/currents would not vary with respect to time.

When we added capacitors and inductors, some voltages/currents would now vary with respect to time, and our circuit analysis became more complicated (i.e., required differential equations to model time dependent voltages/currents). We are usually interested in knowing how these voltages/currents behave after some change occurs in the circuit. For example, if a switch in a circuit closes, a capacitor may begin to discharge (output current) – what will the voltage across the capacitor be 5 seconds after the switch opens? 10 seconds? (in reality the time scale may be in micro- or nanoseconds).

In the homeworks, you have been finding the differential equations and solutions (e.g., v(t) = B/A + ke^(-A*t*)) that model these voltages and currents after some change in a circuit. To help visualize what these equations mean, you will simulate the voltage/current values in RC, RL, and RLC (from HW9, problem 2) circuits in LTspice. In each circuit, a voltage source Vs will change values at a specific time (t = 5s). Your simulation will show you what happens to the voltages/currents before and after the change (t < 5s and t > 5s). You will then perform some analysis to determine initial condition and steady state values.

<u>Reference Lab 3 as you work to remind yourself of how to use LTSpice</u>. You will be doing simulations for 4 circuits – it may save you time in the end if you build a separate schematic for each circuit, rather than doing one, then building onto it to make the next circuit, and so on.

<u>Part 1</u>: Simple R Circuit

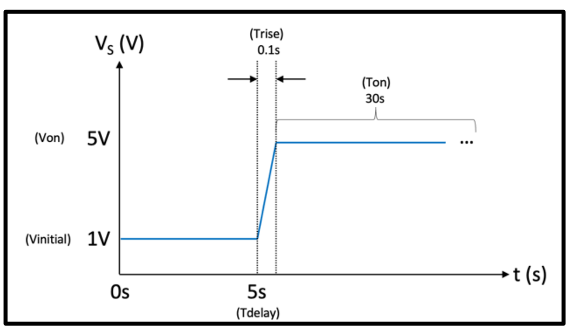


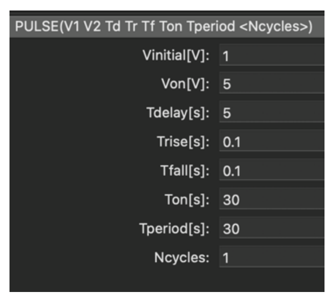
Pictured aboved is a simple resistive circuit with voltage source Vs and resistor R. The bottom of the circuit is grounded. The node between Vs and R is labeled as **Vs** (note this is equal to the voltage across R). As a "control," we will simulate the transient response of this circuit.

- Before t = 5s, Vs = 1V (for a "long" time = steady state)
- At t = 5s, Vs changes to 5V (t=5 here will be like t=0 in the homeworks)
- R = 10 ohms

→ **Simulate the node voltage Vs and current IR (left to right through R)**

1. <u>Build the circuit schematic shown above</u>
   1.1. For the source value of Vs, replace **V** with **PULSE(1 5 5 0.1 0.1 30 30 1)**
      1.1.1. This is saying Vs will act like a square wave pulse with the parameters below.

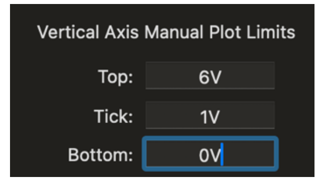




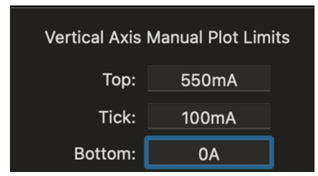
   1.2. When placing the resistor, LTSpice assigns a polarity (+ on top if you place the resistor vertically as is). In other words, LTSpice will tell you the current flowing down the resistor when you simulate it. <u>To get LTSpice to simulate the current flowing left to right through the resistor (clockwise) in the schematic above, rotate it 270deg before placing it into the circuit. The polarity will be + on the left.</u>
      1.2.1. After running the simulation, you can hover your cursor over any component in your schematic – the cursor will change to a symbol indicating the direction of the current that LTspice is plotting.
   1.3. Make sure the node between Vs and R is labeled as **Vs**.

2. <u>Run the transient simulation</u>
   2.1. Add a SPICE directive to measure the transient response for 30s: **.tran 30s**
   2.2. Run the simulation. Right click inside the empty simulation pop-up window and select "**Add plot pane.**" This will split the simulation window into two plots.

   2.3. In the top plot, add the voltage trace for node Vs: **V(vs)**.
      2.3.1. Right click the y axis and set the following parameters:

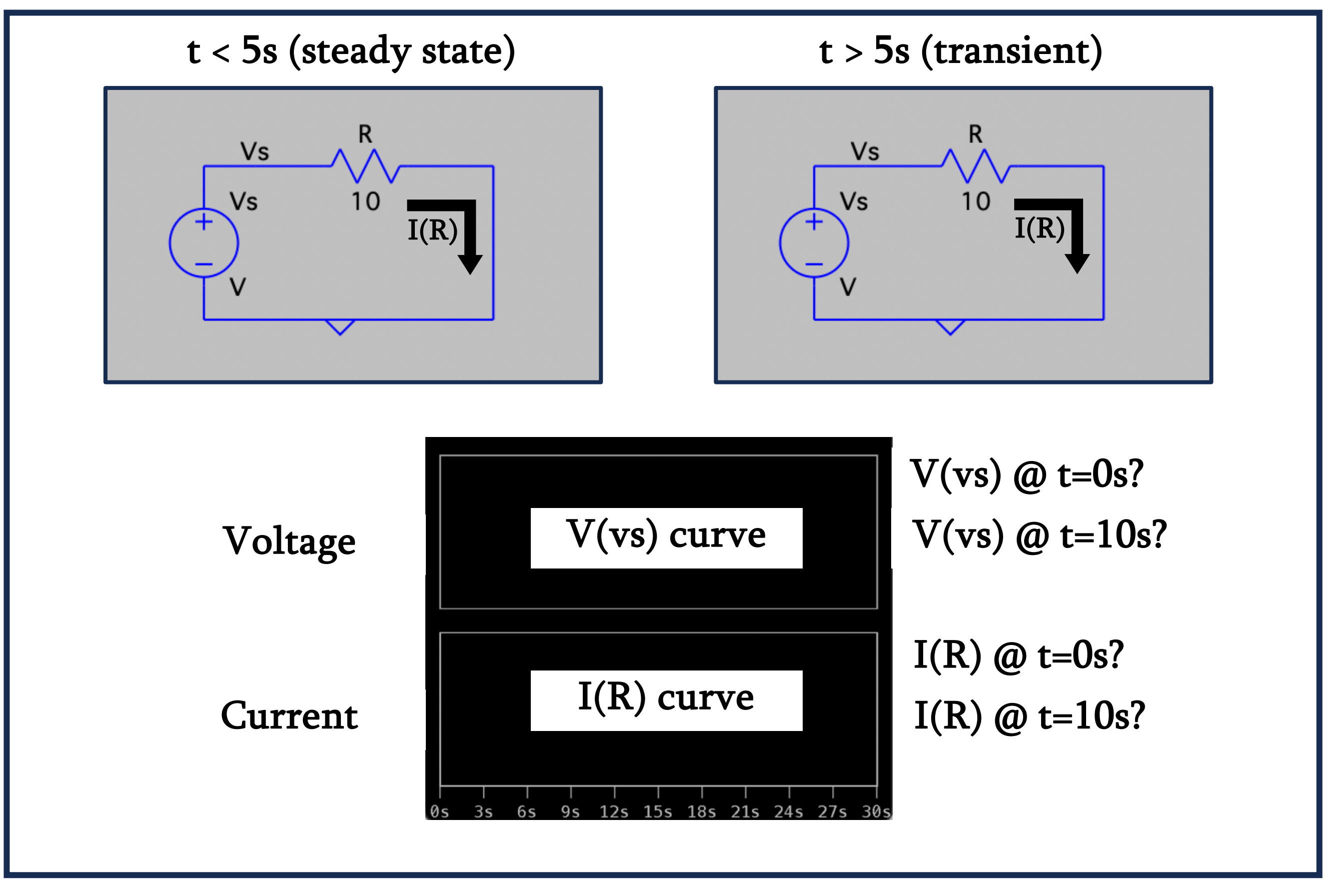


   2.4. In the bottom plot, add the current trace for the resistor current: **I(R)**. You should see positive current if you placed the resistor as directed above.
      2.4.1. Right click the y axis and set the following parameters:

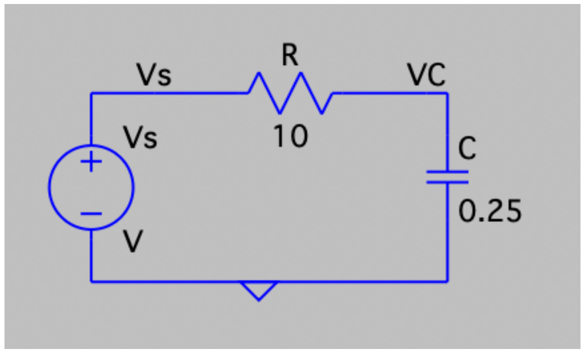


In your lab report, include the following (matching the format below)…

- ☐ Your circuit schematic in the t > 5s box
- ☐ Draw the steady state circuit for t < 5s (you can just include your schematic again for this part, since the resistor will not change).
- ☐ The direction of the simulated current, shown on the schematics.
- ☐ Your voltage plot (with **V(vs)**) and current plot (with **I(R)**).
- ☐ The voltage and current values at the specified times, estimated visually from your plots.



<u>Part 2</u>: RC Circuit



Pictured aboved is an RC circuit with voltage source Vs, resistor R, and capacitor C. The node between Vs and R is labeled as **Vs**. The node between R and C is labeled as **VC** (equal to the voltage across the capacitor).

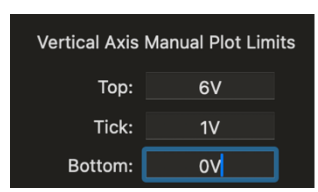
- Before t = 5s, Vs = 1V (for a "long" time = steady state)
- At t = 5s, Vs changes to 5V
- R = 10 ohms
- C = 0.25 F

→ Simulate the voltages Vs and VC, and the current IC (down C)

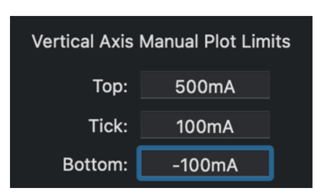
1. <u>Build the circuit schematic shown above</u>
   1.1. For the source value of Vs, replace **V** with **PULSE(1 5 5 0.1 0.1 30 30 1)** (same as before)
   1.2. Make sure the nodes are labeled accordingly.

2. <u>Run the transient simulation</u>
   2.1. Add a SPICE directive to measure the transient response for 30s: **.tran 30s**
   2.2. Run the simulation. Right click inside the empty simulation pop-up window and select "**Add plot pane.**" This will split the simulation window into two plots.

   2.3. In the top plot, add the voltage traces for nodes **Vs** and **VC**: **V(vs)** and **V(vc)**.
      2.3.1. Right click the y axis and set the following parameters:



   2.4. In the bottom plot, add the current trace for the capacitor current: **I(C)**.
      2.4.1. Right click the y axis and set the following parameters:



3. Find the rise time of VC
   3.1. From your voltage plot, you should see that VC starts a steady state value, VC($t=5^-$) (like VC($t=0^-$) normally), then rises and settles at a final value after some time.

   3.2. Enter the following SPICE directive:
      3.2.1. .meas TRAN VC_riseTime FIND time WHEN V(vc)=xV
      3.2.2. This statement finds the time when VC = x and returns that in a variable named VC_riseTime, which you can find in your log file (Hotkey: **CMD+L or CTRL+L**).
      3.2.3. REPLACE x WITH A VALUE SLIGHTLY LESS THAN THE FINAL VALUE YOU SEE IN YOUR PLOT.
      For example, if your final value is 5, enter 4.99 for x.

   3.3. Re-run the simulation and open the log file. With no errors, you should see:

```
vc_risetime: time=20.0256 at 20.0256
```

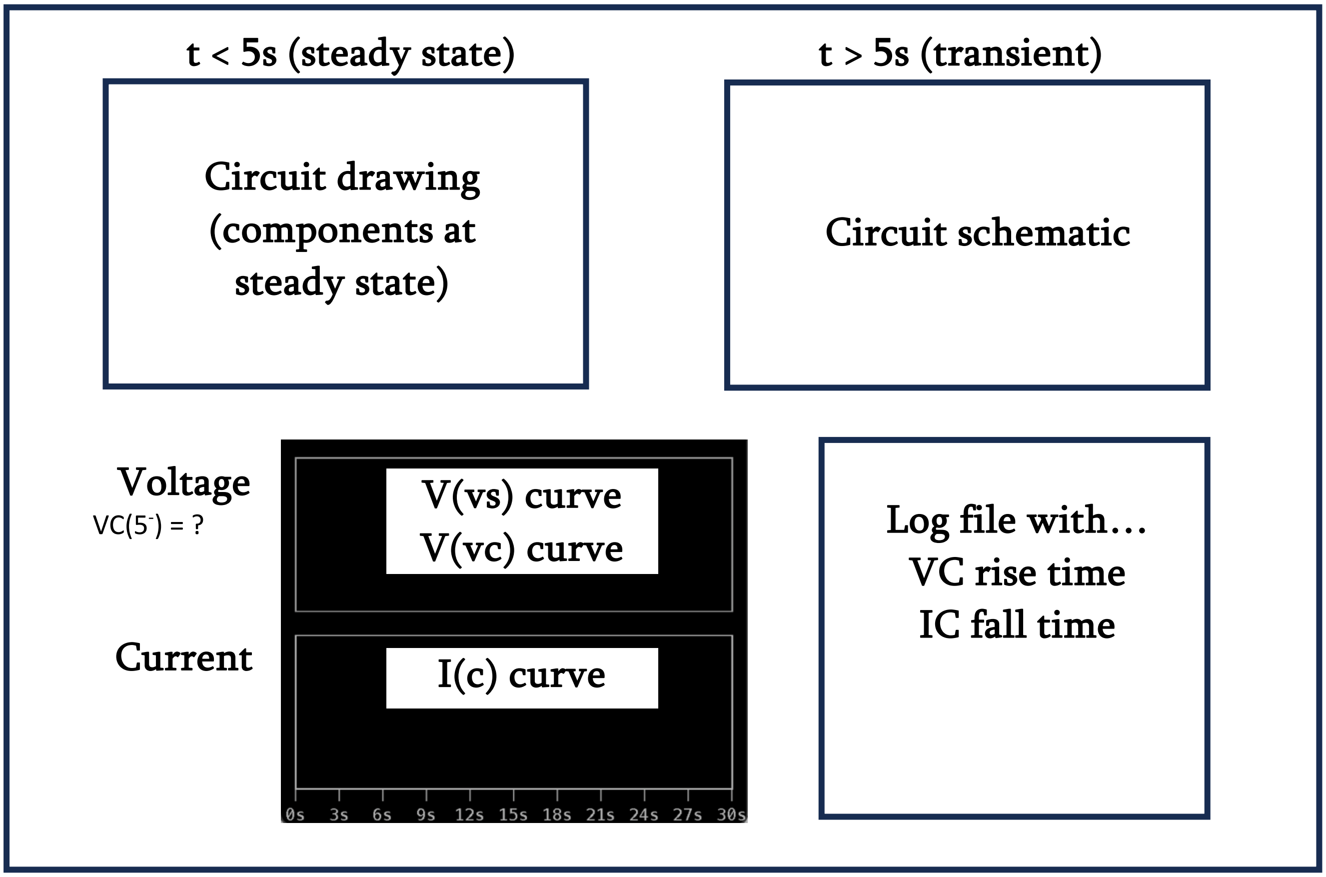
4. Find the fall time of IC
   4.1. From your current plot, you should see that IC starts at some steady state value, spikes to some maximum at t=5s, and then falls until it settles at a final value after some time.

   4.2. Enter the following SPICE directive:
      4.2.1. .meas TRAN IC_fallTime FIND time WHEN I(c)=xmA cross=last
      4.2.2. This statement finds the time when IC = x and returns that in a variable named IC_fallTime, which you can find in your log file (Hotkey: **CMD+L or CTRL+L**). The “cross=last” parameter is needed because IC = x earlier in the plot (before t=5s), but we want the time when IC = x afterwards (after t=5s).
      4.2.3. REPLACE x WITH A VALUE SLIGHTLY GREATER THAN THE FINAL VALUE YOU SEE IN YOUR PLOT.
      For example, if your final value is 0, enter 0.1 for x.

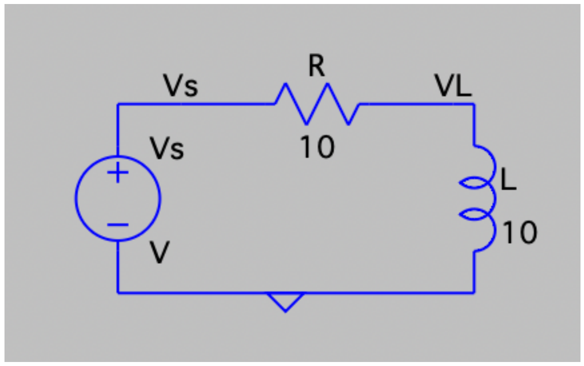
   4.3. Re-run the simulation and open the log file to check your results.

In your lab report, include the following (matching the format below)…

- ☐ Your circuit schematic in the t > 5s box
- ☐ Draw the steady state circuit for t < 5s (remember what capacitors act like at steady state).
- ☐ The direction of the simulated current, shown on the schematics.
- ☐ Your voltage plot (with **V(vs)**, **V(vc)**) and current plot (with **I(C)**).
- ☐ The log file including the indicated rise and fall times.
- ☐ The capacitor voltage at $t = 5^-$ (think $Vc(0^-)$ from the homeworks; list next to or indicate on your plot)



<u>Part 3</u>: RL Circuit



Pictured aboved is an RL circuit with voltage source Vs, resistor R, and inductor L. The node between Vs and R is labeled as **Vs**. The node between R and L is labeled as **VL** (equal to the voltage across the inductor).

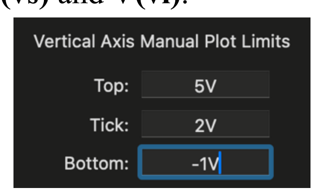
- ☐ Before t = 5s, Vs = 1V (for a "long" time = steady state)
- ☐ At t = 5s, Vs changes to 5V
- ☐ R = 10 ohms
- ☐ L = 10 H

→ Simulate the voltages Vs and VL, and the current IL (down through L)

5. <u>Build the circuit schematic shown above</u>
   5.1. For the source value of Vs, replace **V** with **PULSE(1 5 5 0.1 0.1 30 30 1)** (same as before)
   5.2. Make sure the nodes are labeled accordingly.

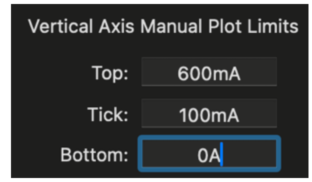
6. <u>Run the transient simulation</u>
   6.1. Add a SPICE directive to measure the transient response for 30s: **.tran 30s**
   6.2. Run the simulation. Right click inside the empty simulation pop-up window and select "**Add plot pane.**" This will split the simulation window into two plots.

   6.3. In the top plot, add the voltage traces for nodes **Vs** and **VL**: **V(vs)** and **V(vl)**.
      6.3.1. Right click the y axis and set the following parameters:



6.4. In the bottom plot, add the current trace for the inductor current: **I(l)**.

6.4.1. Right click the y axis and set the following parameters:



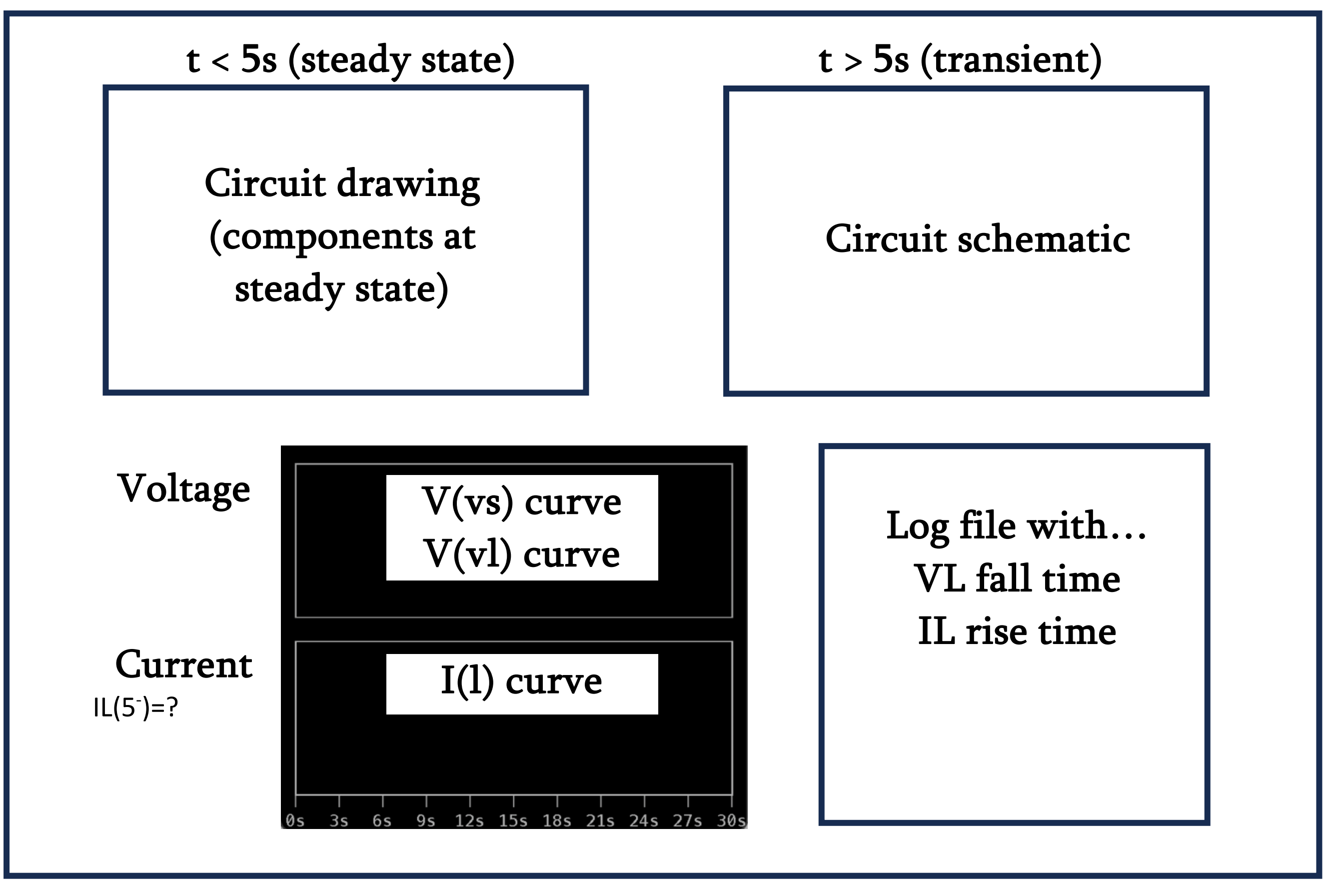
7. <u>Find the fall time of VL and the rise time of IL</u>

7.1. Using similar **.meas** SPICE directives as in the RC circuit, find the fall time of VL and rise time of IL. You can eyeball the final values easily from the plots (if you set the y-axes as above). For VL, make sure to use "**cross=last**" in your statement.

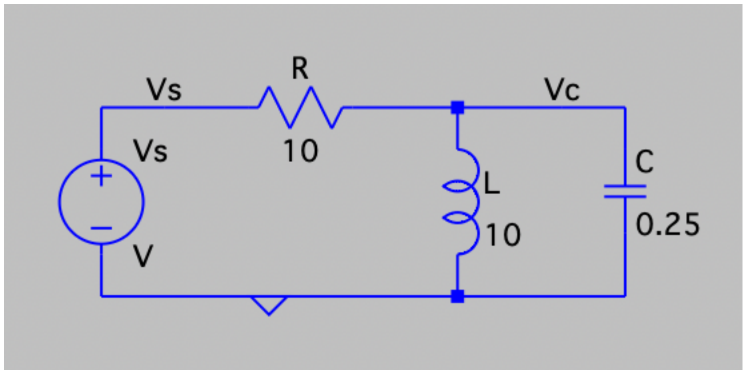
7.2. Re-run the simulation and open the log file to check your results.

In your lab report, include the following (matching the format below)…

- ☐ Your circuit schematic in the t > 5s box
- ☐ Draw the steady state circuit for t < 5s (remember what inductors act like at steady state).
- ☐ The direction of the simulated current, shown on the schematics.
- ☐ Your voltage plot (with **V(vs)**, **V(vl)**) and current plot (with **I(l)**).
- ☐ The log file including the indicated rise and fall times.
- ☐ The inductor current at $t = 5^-$ (think $IL(0^-)$ from the homeworks; list next to or indicate on your plot)



<u>Part 4</u>: RLC Circuit



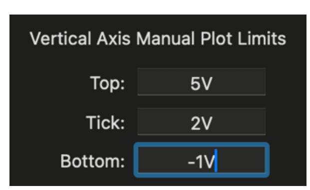
Pictured aboved is an RLC circuit (problem 2 in HW9) with voltage source Vs, resistor R, capacitor C, and inductor L. The node between Vs and R is labeled as **Vs**. The node between R, L, and C is labeled as **VC** (voltage across both the inductor and capacitor, which are in parallel).

- ☐ Before t = 5s, V_S = 1V (for a "long" time = steady state)
- ☐ At t = 5s, V_S changes to 5V
- ☐ R = 10 ohms
- ☐ L = 10 H
- ☐ C = 0.25 F

→ Simulate the node voltages V_S and VC, and the currents IR (left to right), IL (down), and IC (down)

8. <u>Build the circuit schematic shown above</u>
   - 8.1. For the source value of V_S, replace **V** with **PULSE(1 5 5 0.1 0.1 30 30 1)** (same as before)
   - 8.2. Make sure the nodes are labeled accordingly.

9. <u>Run the transient simulation</u>
   - 9.1. Add a SPICE directive to measure the transient response for 30s: **.tran 30s**
   - 9.2. Run the simulation. Right click inside the empty simulation pop-up window and select "**Add plot pane.**" This will split the simulation window into two plots.
   - 9.3. In the top plot, add the voltage traces for nodes **V_S** and **VC**: **V(vs)**, **V(vc)**.
     - 9.3.1. Right click the y axis and set the following parameters:



9.4. In the bottom plot, add the current traces for the resistor, inductor, and capacitor currents: **I(R), I(L), and I(C)**.

9.4.1. The default y axis values should be okay, but you can change them if you want.

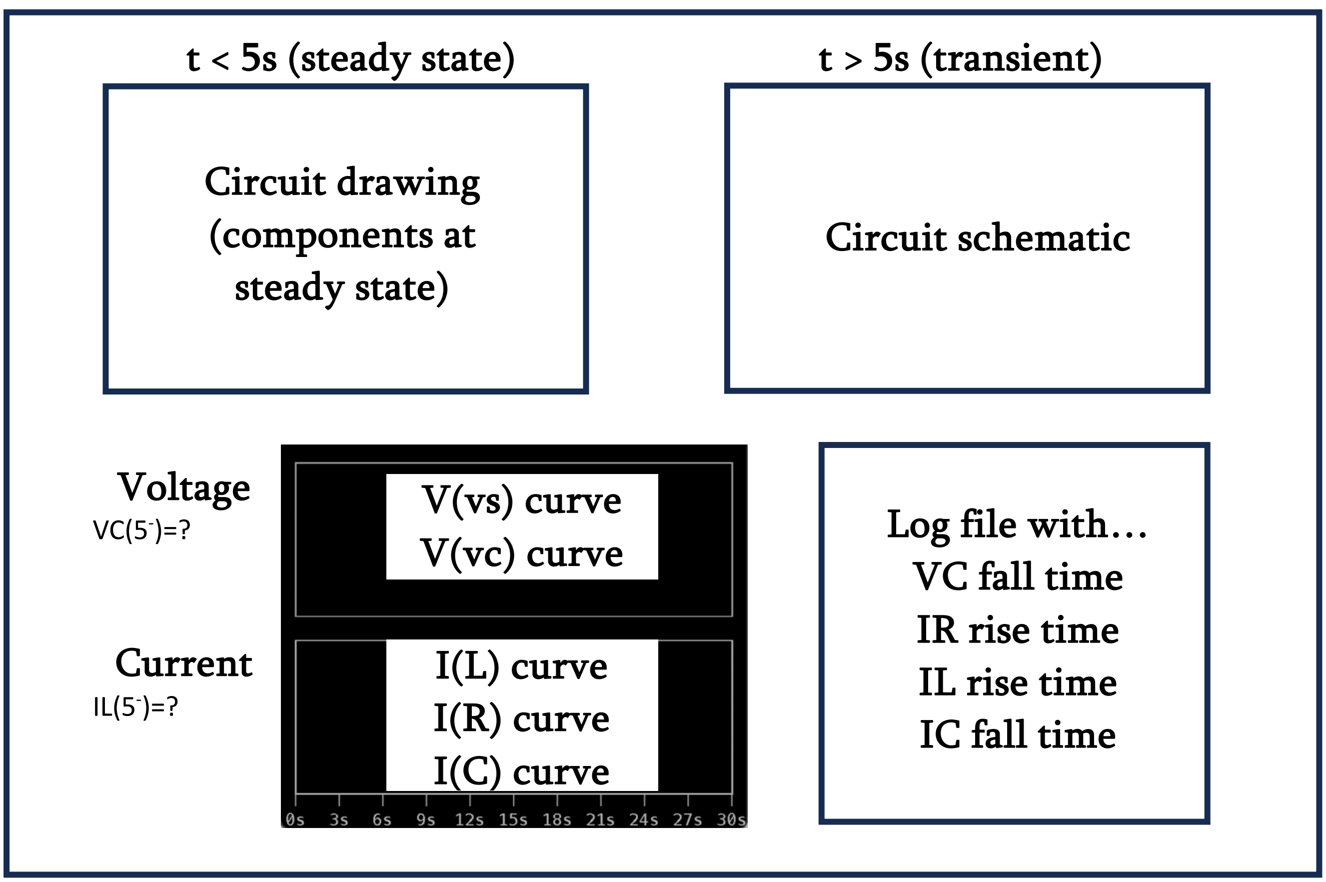
10. <u>Find the rise times of IR and IL, and fall times of VC and IC</u>

10.1. Using similar **.meas** SPICE directives as in the RC and RL circuits, find the specified rise and fall times. You can hover your cursor over the curves to estimate each final value, and then enter just above/below that for your **.meas** statement. In each **.meas** statement, use “**cross=last**.”

10.2. Re-run the simulation and open the log file to check your results.

In your lab report, include the following (matching the format below)…

- ☐ Your circuit schematic in the t > 5s box
- ☐ Draw the steady state circuit for t < 5s (remember what inductors/capacitors act like at steady state).
- ☐ The direction of the simulated currents, shown on the schematics.
- ☐ Your voltage plot (with **V(vs)**, **V(vc)**) and current plot (with **I(R)**, **I(L), and I(C)**).
- ☐ The log file including the indicated rise and fall times.
- ☐ The inductor current at $t = 5^-$ (think $IL(0^-)$ in the homeworks; list next to or indicate on your plot)
- ☐ The capacitor voltage at $t = 5^-$ (think $VC(0^-)$ in the homeworks; list next to or indicate on your plot)



LAB REPORT DUE NEXT WEEK

LAB REPORT 8 – RUBRIC

2,000-word limit 1 report per person

Submission contents listed briefly below, but double check the black box for each section!

Part	Submission Material	Points
	Introduce and define concept (transient analysis)	2.5
	Motivation for experiment	2.5
1	R Circuit	15
	Two schematics	5
	Voltage and current plots	5
	Specified voltage/current values	5
2	RC Circuit	20
	Two schematics	5
	Voltage and current plots	5
	Log file	5
	Specified voltage/current values	5
3	RL Circuit	20
	Two schematics	5
	Voltage and current plots	5
	Log file	5
	Specified voltage/current values	5
4	RLC Circuit	20
	Two schematics	5
	Voltage and current plots	5
	Log file	5
	Specified voltage/current values	5

Appendix

Helpful list of LTspice syntax and shortcuts

COMMANDS

SPICE Analysis

.OP	find the DC operating point
.TRAN	perform nonlinear transient analysis
.AC	perform small signal AC analysis
.DC	perform DC source sweep analysis
.TF	find the DC small-signal transfer function
.NOISE	perform noise analysis

SPICE Directives

.BACKANNO	annotate subcircuit pin names on port currents
.END	end of netlist
.ENDS	end of subcircuit definition
.FOUR	compute fourier component
.FUNC	user defined functions
.FERRET	download a file from URL
.GLOBAL	declare global nodes
.IC	set initial conditions
.INCLUDE	include file
.LIB	include library
.LOADBIAS	load a previously solved DC solution
.MACHINE	arbitrary state machine
.MEASURE	evaluate user-defined electrical quantities
.MODEL	define a SPICE model
.NET	compute network parameters in .AC analysis
.NODESET	supply hints for initial DC solution
.OPTIONS	set simulator options
.PARAM	user-defined parameters
.SAVE	limit the quantity of saved data
.SAVEBIAS	save operating point to disk
.STEP	parameter sweeps
.SUBCKT	define a subcircuit
.TEMP	temperature sweeps
.TEXT	user-defined string
.WAVE	write selected nodes to a .WAV file

ANALOG DEVICES

LTspice®

Fast • Free • Unlimited

SHORTCUTS

Schematic and Symbol Editing Modes

	Choose Mode then select component Exit mode: Press Esc or right-click	
F5 or Delete or Ctrl X	cut/delete	F5
F6 or Ctrl C	copy/duplicate*	F6
F7	move* *unselected wires remain*	F7
F8	drag* *connected wires adjust*	F8
Esc	exit current mode *or right-click*	Esc

Zoom and Grid

	Zoom in and out with scroll wheel or track pad pinch	
Ctrl Z	**Schematic** zoom area (drag over area) zoom in (click on scheme) **Waveform** zoom area is default mode F9 for previous zoom **Symbol** zoom in	
Ctrl B	zoom out	
Space	zoom to fit (schematic viewer)	Space
Ctrl E	zoom extents (waveform viewer)	
Ctrl G	toggle grid	

Place Component Modes*

	Press Esc or right-click to exit place component mode	
R	resistor	R
C	capacitor	C
L	inductor	L
D	diode	D
G	ground	G
V	voltage	V
S	spice directive *right-click text field to open "Help me Edit" dialog*	S
T	text/comment	T
F2	component	F2
F3	draw wire	F3
F4	label net	F4
	bus tap	B

*Rotate and Mirror

	*enabled in place modes	
Ctrl R	rotate	⌘ R
Ctrl E	mirror	⌘ E

Undo/Redo

	### Levels of Undo	
F9	undo	F9 or ⌘ Z
⇧ F9 or Ctrl ⇧ Z	redo	⇧ F9 or ⌘ ⇧ Z

TRICKS

Waveforms

	when clicking waveform label	
click	add cursor and see measure	click
Alt click	highlight corresponding net in schematic	⌘ click
Ctrl click	integrate waveform	Ctrl click

Schematics

Alt click	**component:** plot instantaneous power **wire:** plot current	⌘ click
hold Ctrl	draw wires at an angle	hold Shift
Ctrl Alt Shift H	show hidden component values/text, e.g. parallel or series resistance and capacitance	

any text preceded by an underscore, e.g. "_FAULT" is displayed with an overbar, active low, signal

NUMBERS

Prefixes (Case Insensitive)

LTspice	Means	Value
T or t	tera	10^{12}
G or g	giga	10^{9}
meg	mega	10^{6}
K or k	kilo	10^{3}
M or m	milli	10^{-3}
U or u	micro	10^{-6}
N or n	nano	10^{-9}
P or p	pico	10^{-12}
F or f	femto	10^{-15}

Constants

LTspice	Means
e	Euler's number
pi	π
k	Boltzmann constant
q	charge constant
true	1
false	0
mil	25.4×10^{-6}m