ElecEng 2CF3

ASSIGNMENT 4

Laplace Transform in Circuit Analysis

Ronald Luo, Mihail Georgiev, and Natalia K. Nikolova

Instructions and Objective

- Perform all exercises.
- For each exercise, there are instructions on what to include in a short report about your observations. The report also must contain the answers to the posed questions.
- For each exercise, you are required to submit: (i) your <u>final</u> LTspice (*.asc) file and (ii) your short report (*.pdf) file. Submit to the respective Dropbox on A2L by the respective deadline. Do not include intermediate *.asc files. Failure to submit either in the correct file format will be penalized.
- Each exercise report (the PDF) must include (at the top of the first page) a suitable short title, your full name <u>as it appears on Avenue to Learn</u>, your student ID number, and your MacID login name.
- If you need help in using LTspice, please refer to LTspice *QuickStart Guide* PDF file on A2L.
- There is a requirement to include SPICE netlists in the report. Just copy-paste the netlist text from LTspice. Do not export or submit any netlist files.

The objectives of Assignment 4 are to learn how to: (1) use LTspice to carry out analysis of transient circuits, and (2) use the Laplace transform to analyze transient circuits in the *s*-domain.

EXERCISE #1: TRANSIENT 1ST DEGREE CIRCUIT

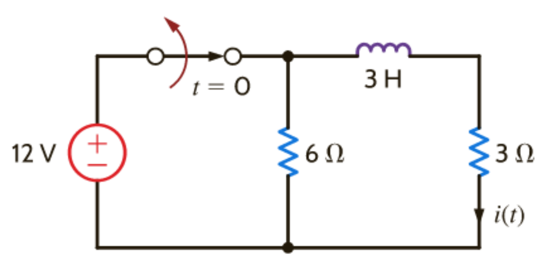

Fig. 1. Transient circuit from Problem 13.59, Chapter 13, Erwin & Nelms.

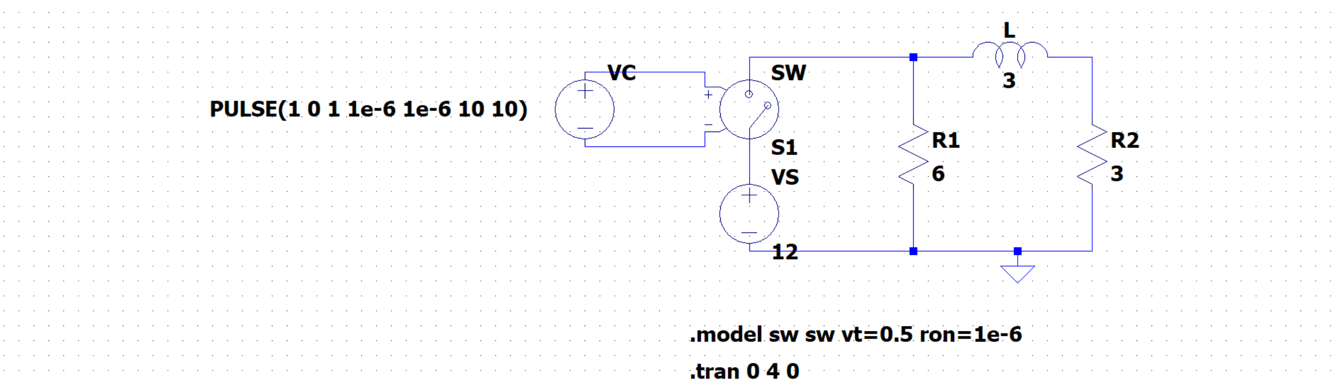

Fig. 2. Implementation of the switched circuit with a voltage-controlled switch in LTSpice.

The circuit shown in Fig. 1 has a DC voltage source $V_S = 12$ V. The resistances are $R_1 = 6\ \Omega$ and $R_2 = 3\ \Omega$, respectively. The inductance is $L = 3$ H. A switch is initially closed and opens at $t = 0$ s. The task is to determine the time-dependent current $i(t)$ through the inductor branch.

First, you must derive the solution for $i(t)$ using *s*-domain (Laplace transform) analysis. To this end, you must determine the initial value of the current $i(t = 0_-)$ from the circuit state before the switch is opened. You can then proceed to the *s*-domain analysis to obtain $i(t)$. Your solution will need to be included in the report.

Second, you must generate a plot of $i(t)$ versus time t using MATLAB. The snippet below will help you code the plotting routine in MATLAB. All you need to do is to modify the actual function describing $i(t)$. When finished, run the code. A plot will appear. Your code and plot will need to be included as part of the report.

<u>Sample MATLAB Code</u>

```
clear all; close all; % clean up memory and close all open plot windows
t = linspace(0, 1, 1001); % vector of time samples where function is calculated
i = exp(-t); % change this to the function i(t) you found from Laplace analysis
figure;
plot(t, i);
grid on;
title('Current through Inductor in Exercise 1');
xlabel('{\it t} (s)');
ylabel('{\it i} (A)');
```

Third, you will simulate the switched circuit in Fig. 1 in LTspice. To this end, you first need to build the circuit using an inductor $L = 3$ H, two resistors, $R_1 = 6\ \Omega$ and $R_2 = 3\ \Omega$, and a DC voltage source $V_S = 12$ V (see Fig. 2). You also need to emulate the switch using a *voltage-controlled switch* `sw` along with a *controlling voltage source* V_C. Start by including the controlling voltage source (pulse source) V_C as per the specification:

`PULSE(1 0 1 1e-6 1e-6 10 10)`

This is a pulse source with low and high values of 0 and 1 V, respectively. It starts high (1 V) at $t = 0$, drops to 0 at $t = 1$ s, and stays low for 10 s, which is longer than the length of the simulation time (4 s) recommended below. This ensures that the simulation emulates a steady state after switching that extends to infinite time. The source controls the switch so that it is closed from 0 to 1 s, then opens at $t = 1$ s and stays open until the end of the simulation. Note that this setting is somewhat different from the original problem where the switch opens at $t = 0$. This will simply shift the time response $i(t)$ to the right on the time axis by 1 second. On the other hand, it will allow you to observe well the steady state of $i(t)$ before the switching occurs in the interval from 0 to 1 s. Further, you need to connect the voltage control source to the switch. The switch component `sw` can be found in the main component menu. When placing the switch in the circuit, you may need to rotate (or flip) it to match the polarity of V_C as shown in Fig. 2. Once placed, right click on it and set its attribute `Value` to `SW`.

Before running the simulation, you will also need to add the following SPICE directive to your schematic. Select  from the toolbar, select `SPICE Directive`, and add the following text:

`.model sw sw vt=0.5 ron=1e-6`

Click OK, then click anywhere in the schematic to include the directive. This directive specifies how the voltage-controlled switch functions: (1) `vt` sets the threshold voltage (in this case 0.5 V) – this is the control voltage above which the switch is closed; (2) `ron` is the internal switch resistance (should be very small).

Make sure your source voltage, control voltage, and other components are properly labeled. The following simulation parameters are suitable for the setup in Fig. 2:

```
.tran 0 4 0
```

Run the simulation and observe $V_C(t)$ and $i(t)$ (the current through the inductor branch) on a common plot. Save the plot, the circuit diagram, and the netlist for your report.

For the Report

1. Include the complete schematic (screenshot or image export).
2. Include the complete netlist (`View→SPICE Netlist`).
3. Include the plot of $V_C(t)$ and $i(t)$ from the LTspice simulation.
4. Include your analytical solution for $i(t)$ based on the Laplace transform.
5. Include the MATLAB source code and plot in your PDF report.
6. Does the LTspice simulation result agree with the MATLAB plot of the theoretical result? Justify your answer.

Note: Do not forget to submit the LTspice (*.asc) file, named properly, e.g., `Exercise1.asc`.

EXERCISE #2: TRANSIENT 2ND DEGREE CIRCUIT

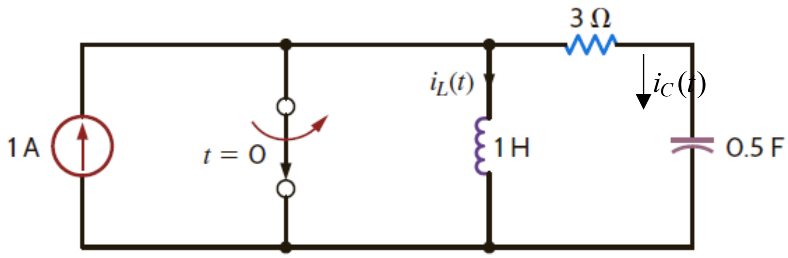

Fig. 3. Transient circuit from Problem 13.60, Chapter 13, Erwin & Nelms.

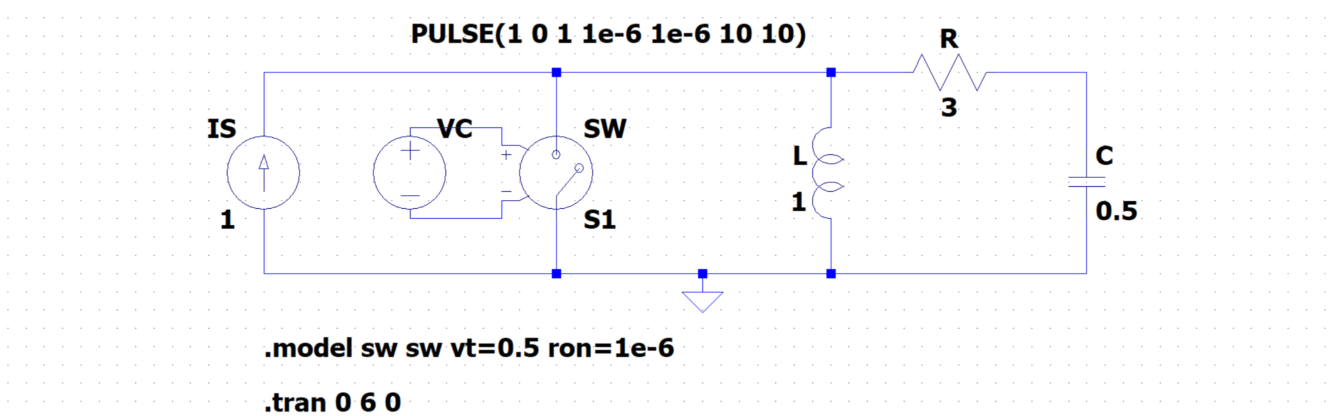

Fig. 4. LTspice implementation of the problem shown in Fig. 3.

In this exercise, you will analyze the switched *RLC* circuit shown in Fig. 3. Its implementation in LTspice is shown in Fig. 4. A DC current source is given on the left as $I_S = 1$ A. A voltage-controlled switch is placed in parallel. When the switch is closed (as it is initially), the current source is shorted, leaving the *RLC* circuit on the right without power. When the switch opens, the current source powers the circuit, which goes through a transient state before reaching a steady (DC) state. It is this transient state that you need to analyze.

First, using *s*-domain (Laplace transform) analysis, determine analytically the transient current through the inductor $i_L(t)$ and that through the capacitor $i_C(t)$. Your solution will need to be included in the report.

Second, using MATLAB, plot the so obtained waveforms of $i_L(t)$ and $i_C(t)$. The snippet below will help you code the plotting routine in MATLAB. All you need to do is to modify the actual functions describing $i_L(t)$ and $i_C(t)$. When finished, run the code. A plot will appear. Your code and plot will need to be included in the report.

Sample MATLAB Code

```
clear all; close all;
t = linspace(0, 1, 1001);
iL = exp(-t); % change this to the function iL(t) you found from Laplace analysis
iC = exp(-t); % change this to the function iC(t) you found from Laplace analysis
figure;
plot(t, iL,'-k') % plot curve in solid black line
hold on;
plot(t, iC,'--b') % plot curve in dash blue line
hold off;
grid on;
legend('{\it i}_L','{\it i}_C')
title('Currents in Exercise 2')
xlabel('{\it t} (s)');
ylabel('{\it i} (A)');
```

In this simulation, a voltage-controlled switch is used again. You can follow the same instructions as in Exercise #1 to add the voltage controller, the switch, and the SPICE directives as shown in Fig. 4. The following parameters can be used to run the simulation:

`.tran 0 6 0`

Plot the simulated waveforms of $i_L(t)$ and $i_C(t)$.

For the Report

1. Include the complete schematic (screenshot or image export).
2. Include the complete netlist (`View→SPICE Netlist`).
3. Include the plot of $i_L(t)$ and $i_C(t)$ from the LTspice simulation.
4. Include your analytical solution for $i_L(t)$ and $i_C(t)$ based on the Laplace transform.
5. Include the MATLAB source code and plot in your PDF report.
6. Does the LTspice simulation result agree with the MATLAB plot of the theoretical result? Justify your answer.

Note: Do not forget to submit the LTspice (*.asc) file, named properly, e.g., `Exercise2.asc`.