Lab 7
PSpice: Time Domain Analysis

OBJECTIVES
1. Use PSpice Circuit Simulator to simulate circuits containing capacitors and inductors in the time domain.
2. Practice using a switch, and a Pulse & Sinusoidal voltage source in RL and RC circuits.

EQUIPMENT
PSpice Program

THEORY
The inputs to an electric circuit have generally been independent voltage and current sources. PSpice provides a set of voltage and current sources that represent time varying inputs. "Time Domain (Transient)" analysis using PSpice simulates the response of a circuit to a time varying input. Time domain analysis is most interesting for circuits that contain capacitors or inductors. For the capacitor, the part properties of interest are the capacitance and the initial voltage of the capacitor (the "Initial Condition"). For the inductor on the other hand, the part properties of interest are the inductance and the initial current of the inductor (the "Initial Condition").

In this lab we consider three examples. The first example illustrates transient analysis of a RL circuit with a single switch; the second and third circuits analyzed are the response of an RC circuit to a Pulse voltage source and a sinusoidal voltage source.

We will again use a five-step procedure to organize circuit analysis using PSpice.

Part 1: Transient Analysis of an RL Circuit with a Switch
Time varying voltages and currents can be caused by opening or closing a switch. PSpice provides parts to represent single-pole, single-throw (SPST) switches in the parts library. These parts are summarized below:

Description: Open switch will close at $t = T_{CLOSE}$; Open switch will close at $t = T_{CLOSE}$; TCLOSE = 0

PSpice Name: Sw_tCLOSE
Sw_tOPEN

Library: Eval

PSpice Problem
The circuit shown is at steady state before the switch closes at time $t = 0$.

The current in the inductor before the switch is closed is $i_L(t = 0) = 40$ mA. Find the current of the inductor as a function of time after the switch closes. That is, use PSpice to simulate the circuit after the switch closes.
Analytical Solution: set $i_L(0^-) = 40$ mA and $I_{SC} = 60$ mA.

$$i_L(t) = I_{SC} + (I_L(0^-) - I_{SC}) \cdot e^{-t/\tau} = (60 - 20e^{-40000t}) \text{ mA}$$

where $\tau = L/R_{Th} = 25 \mu\text{s}$.

**Step 1: Formulate** a circuit analysis problem

After the switch closes the circuit’s time constant is 25 $\mu\text{s}$. Plot the inductor current, $i_L(t)$, for the first 150 $\mu\text{s}$ (6 time constants) after the switch closes.

**Step 2: Describe** the circuit using Capture

Place the parts (an inductor has terminal points 1 & 2 that indicate a positive and negative polarity, respectively) and set the initial condition $i_L(0^-) = 40$ mA. That is, double click on the inductor and set the property named "IC" $\equiv$ initial condition $= 40$ mA.

**Step 3: Simulate** the circuit using PSpice.

Edit the Simulation to be “Time Domain (Transient)”. The simulation starts at time zero and ends at the "Run to time". Specify the "Run to time" as $6\tau_{RL} = 150 \mu\text{s}$ and run it.

**Step 4: Display** the results of the simulation using Probe

After a successful simulation, a Probe window will automatically open. Add trace I(L1:1) (the current through inductor L1 through node 1) and the resulting plot is

Hints

i. To remove grid lines from the plot, right-click on the plot and select settings. Under the X & Y Grid tabs, select none two times. One can set this as the default by clicking “Save As Default.”

ii. To change the color or thickness of a line, edit Trace Properties.

iii. To copy this plot go to the “Window” tab on the tool bar and select “copy to clipboard” and paste into your document.

iv. To search for specific values, click the “Toggle Cursor” followed by the “Search Cursor”. Type in the commands SFXV (xvalue) for x-values or SFLE (yvalue) for y-values.

**Step 5: Verify** that the simulation results are correct.

The initial response of the inductor current is $i_L(0^-) = 40$ mA and its final state current is $I_{SC} = i_L(\infty) = 60$ mA. Verify that the plot agrees with the mathematical prediction.

To ask a more detail question such as what is the value of $i_L(t)$ at a time of 30.405 $\mu\text{s}$, there are several tools that are at our disposal: the Probe cursor button and Mark Data points & adjust value buttons allow one to select a particular data value on the curve.

To find the current at $t = 30.405 \mu\text{s}$, one can find that the plot indicates that $i_L(t) = 53.592$ mA when $t = 30.405 \mu\text{s}$. Substituting $t = 30.405 \mu\text{s}$ into the complete natural response gives $i_L(t) = 54.073$ mA, a difference of 0.5%. The simulation results are correct.

**Part 2: The Response of an RC Circuit to a Pulse Input**

**PSpice Problem**
The source voltage and RC circuit are

Use PSpice to simulate the voltage source and the capacitor’s voltage \( v_{C,PS}(t) \) and compare it to theoretical prediction

\[
v_{C,th}(t) = \begin{cases} 
4 \cdot (1 - e^{-1000t}) & \text{if } 0 < t < 2 \text{ ms} \\
-1 + 4.46 \cdot e^{-1000(1-0.002)} & \text{if } 2 \text{ ms} < t < 10 \text{ ms}
\end{cases}
\]

**Step 1: Formulate** a circuit analysis problem
Plot three voltages simultaneously: \( V_S(t) \), \( v_R(t) \) and \( v_C(t) \) vs \( t \).

**Step 2: Describe** the circuit using Capture.
Place the parts, adjust parameter values and wire the parts together. The voltage and current sources that represent time varying inputs are provided in the “SOURCE” parts library. In our circuit, the voltage source is a VPULSE part.

The plot of \( v_{in}(t) \) shows making the transition from -1 V to 4 V instantaneously. Zero is not an acceptable value for the parameters \( tr \) or \( tf \). Choosing very small value for \( tr \) and \( tf \) will make the transitions appear to be instantaneous when using a time scale that shows a period of the source waveform. In this example, the period of the source waveform is 10 ms so 1 ns is a reasonable choice for the values of \( tr \) and \( tf \).

Set \( td \), the delay before the periodic part of the waveform, to zero. Then the values of \( v1 \) and \( v2 \) are -1 and 4, respectively. The value of \( pw \) is the length of time that \( v2 = 4V \), so \( pw = 2 \) ms in this example. The value of \( per \) is the period of the pulse function, 10 ms.

Nodes can be given more convenient names using an “off-page connector:” OFFPAGELEFT-R or OFFPAGELEFT-L. Edit the properties naming each connector with “SOURCE” and “CAP.”

**Step 3: Simulate** the circuit using PSpice.
Specify the run time for two periods (20 ms).

**Step 4: Display** the results of the simulation, for example, using Probe. Add traces \( V(SOURCE) \), \( V(CAP) \), and \( V(SOURCE)−V(CAP) \).
Go back and change the pulse width to 5ms. How do the resulting plots look like?

**Step 5: Verify** that the simulation results are correct.

To verify the results, label four points (two from each RC curve) and check with the predictions of the complete natural response. I’ve done two of them, now you check the other two listed in the table below.

<table>
<thead>
<tr>
<th>t (ms)</th>
<th>$v_{C,PS}(V)$</th>
<th>$v_{C,thy}(V)$</th>
<th>%</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.9912</td>
<td>3.4638</td>
<td>3.4539</td>
<td>0.3</td>
</tr>
<tr>
<td>2.7876</td>
<td>1.0551</td>
<td>1.0551</td>
<td>2.4</td>
</tr>
<tr>
<td>12.000</td>
<td>3.3385</td>
<td></td>
<td></td>
</tr>
<tr>
<td>12.757</td>
<td>1.0506</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Part 3: The Response of an RC Circuit to a Sinusoidal Input PSpice Problem**

Use PSpice to plot the capacitor’s voltage $v_{C,PS}(t)$ and power $P_C(t)$ delivered to the capacitor

$$v_{C}(t) = 1.55\sin(2\pi t - 76.6^\circ) \text{ V}$$

$$P_C(t) = v_C(t) \cdot i_C(t) = 3.77\sin(40\pi t - 153^\circ) \text{ W}$$

**Step 1: Formulate** a circuit analysis problem.
Simulate the circuit to determine the capacitor voltage, $v_C(t)$. Plot the $v_C(t)$ versus $t$. Also, plot the power delivered to the capacitor as a function of time.

**Step 2: Describe** the circuit using Capture.
Use capacitor C/ANALOG_P such that node-1 is the positive polarity

**Step 3: Simulate** the circuit using PSpice.
Specify the run to time as 0.8s to run the simulation for eight full periods.

**Step 4: Display** the results of the simulation.
In the Probe window, add the trace V(CAP). One observes that the plot is disappointing for a couple of reasons. First, it’s a very rough representation of a sine function. Second, it takes a while for the capacitor voltage to settle down. In other
words, the capacitor voltage includes a transient part as well as the steady state response. In this example, we only want the steady state response and so would like to eliminate the transient part of the response.

- To obtain a smoother plot, edit the Simulation Settings and set the "Maximum step size" to smaller step size: 0.001s.
- The transient part of the capacitor’s voltage can be eliminated by adjusting the initial capacitor’s voltage to

\[ v_c(t = 0) = 1.55 \sin(20\pi t - 76.6^\circ) \text{ V} = -1.55 \sin(76.6^\circ) = -1.506 \text{ V} \]

Set the initial condition (IC) of the capacitor to be this value.

With these two adjustments, the resulting plots are

![Plot of capacitor voltage](image1)

![Plot of power dissipation](image2)

**What to turn in.**

1. The circuit is at steady state before the switch is thrown at \( t = 0 \).

![Circuit diagram](image3)

   a. Determine and plot the voltage across the capacitor \( v_c(t) \) and the current \( i_{2k\Omega}(t) \) through the 2k\( \Omega \) resistor for \( t > 0 \). PSpice does not let you plot current and voltage simultaneous since they have different units.
   
   b. Simulate the circuit, print out the plots and explain (using short concise sentences) the behavior of the capacitor and 2k\( \Omega \) resistor using your plots.

2. Turn in parts 1, 2 and 3 of the PSpice lab. Make sure that you show your work for verifying the simulation results!