EE 220/220L Circuits I (Fall 2019)
Laboratory Project 11 PSpice for Transient Circuits

Introduction

In this laboratory project, you will use PSpice to model the first-order circuits investigated in Laboratory Project 10 RL and RC Circuits (Lab 10). This is meant to aid in satisfying the ABET course objective to "Use PSpice to model/simulate simple DC, transient, and AC circuits."

PSpice Transient Analysis (put in logbook and keep electronic copies for technical report)

Follow prior instructions to open 'Capture CIS Lite'.

1) Use PSpice to simulate the *RL* circuit shown in Figure 1a of Lab 10 using component values given in the **Preliminary**.
   - To simulate $v_s(t)$, use the Part Browser to select and place a voltage source of type **VPULSE**. Then, use the mouse to double click V1=' to open a 'Display Properties' window, enter **0** in the 'Value:' box, and click 'OK'. Repeat to set V2 to **7V**, TD to **0**, TR to **1ns**, TF to **1ns**, PW to **1.2ms**, and PER to **2.4ms**
   - Place and define the remaining parts of the circuit, including a ground, and wire the circuit interconnections.
   - Add text to title your circuit including- EE220L-xx, lab number and title, *your name*, and *date*.
   - After the circuit is created, go to the top menus and click PSpice ⇒ New Simulation Profile. In the New Simulation window enter a *Name* and click 'Create'.
   - In the 'Simulation settings- *Name*' window, click on the 'Run To Time :' box and enter **2.4ms**. In the 'Transient Options box', click on the box for 'Maximum Step Size' and enter **1u**. Then, click on 'Configuration Files' (on left) ⇒ 'Library' ⇒ 'OK'.
   - Use mouse to select the Voltage Level/Marker icon (looks like a voltage probe with a small green V). The cursor will now have a voltage marker attached to it. Click on the wire to the right of R_s to place a marker for $v_{in1}(t)$ and click on the wire at the top of the inductor to place a marker for $v_L(t)$. Right click the mouse or press the <Esc> key to stop placing markers.
   - Save your project and click the 'Run' icon.
   - Run simulation. Assuming circuit is correctly set up, a new program window entitled "Schematic1 – *filename* - PSpice A/D Lite" will now come up with a plot showing both $v_{in1}(t)$ and $v_L(t)$.
   - Click "Toggle cursor" icon (or use mouse to select the "Trace" dropdown menu, select "Cursor" to bring its dropdown menu, and select "Display") to activate cursor and the cursor window (lower right-hand corner). Next, click mouse cursor on the $v_L(t)$ trace which will cause a red cross-hair to appear with the corresponding ***X*** (i.e., t) and **CURSOR 1 & 2** (i.e., $v_{in1}(t)$ and $v_L(t)$) values displayed in the cursor window. Collect PSpice v_L values at **200 μs** intervals as was done in Lab 10. [Note: Arrow keys can move probe cursor as well.] Prepare a **table** to compare **analytic** and **PSpice** inductor voltage v_L values. Put time values in first column, analytic v_L values generated from Lab 10 equations in second column, PSpice v_L values in third column, and percent difference in fourth column.

- Next, add text to the plot including- EE220L-xx, lab number and title, *your name,* and *date* by successively selecting the dropdown menu "Plot", "Label", and "Text…". Note, fonts, axes, line widths, labels, etc. can be added or edited as desired. The plot can be copied and pasted into MS Word (or other MS-Windows program) by selecting "Windows" drop down menu and selecting "Copy to Clipboard". Insert copies of the **PSpice circuit** and **voltage output plot** in logbook.
- Compare PSpice results with results of preliminary from Lab 10 using table and by inserting another copy of your Lab 10 Matlab **plot** of $v_L(t)$. How well do they compare?

2) Repeat 1) using **measured/calculated** circuit component values for the *RL* circuit. Include/add wire resistance of the actual inductor, i.e., real inductor model consists of an ideal inductor in series with the resistance of the inductor. In a **table**, compare these PSpice results with results measured using oscilloscope in experimental part of Lab 10. Put time values in first column, measured v_L values in second column, PSpice v_L values in third column, and percent difference in fourth column. Also, insert copy of the *RL* circuit voltages **bitmap** from the oscilloscope. How well do they compare?

3) Use PSpice to simulate the *RC* circuit shown in Figure 1b of Lab 10 using component and source values given in the **Preliminary** to obtain a plot of $v_{in2}(t)$ and $v_C(t)$. Follow steps given in part 1). Compare PSpice results with results of preliminary from Lab 10 using table and by inserting another copy of your Lab 10 Matlab **plot** of $v_C(t)$. How well do they compare?

4) Repeat 3) using **measured/calculated** circuit component values for the actual *RC* circuit. In a **table**, compare PSpice results with results measured using oscilloscope in experimental portion of Lab 10. Also, insert copy of the *RC* circuit voltages **bitmap** from the oscilloscope. How well do they compare?

Logbook(s) due by end of labs on Thursday, November 14, 2019 for grading of Labs 10 & 11.