ECE250 Lab 1. Review Problems and Orcad PSPICE Familiarization

Lab Team # _____ Date: _______ Members: _____________________

(Your Lab Team # is your lab station number) _____________________

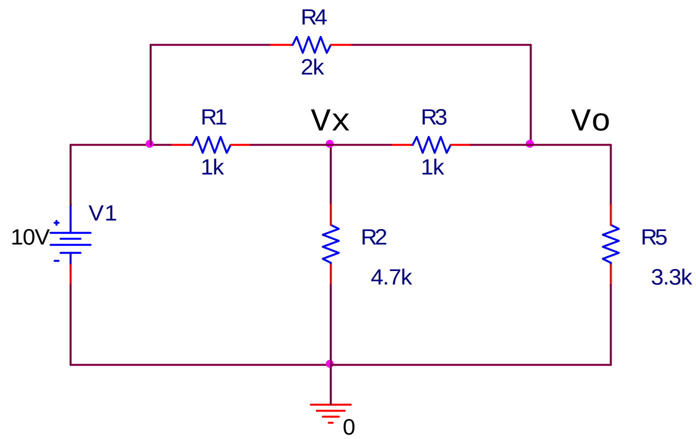
NOTE: *Individual writeups are required for this lab (please circle the submittor's name above)*

In this lab exercise, you will become familiar with (1) dc bias point, (2) time-domain transient, and (3) ac sweep Orcad Lite 9.2 PSPICE analyses. You will also have a chance to review some basics from your previous circuits courses, and also to review your use of MAPLE. Please start by reading and studying the various SPICE examples in the "ECE250 Getting Started with Cadence Orcad Lite V. 9.2" document. ***Before*** coming to lab, please install the ORCAD Lite 9.2 (PSPICE) program on your laptop PC, following the installation directions appearing in Part 5 of the "Getting Started" document. Also, please work through the tutorial found in Part 6 of this document ***before*** coming to lab.

Part 1. DC Circuit Analysis and SPICE DC "Bias Point" simulation

Consider the "bridged tee" resistive circuit of Fig. 1.

Figure 1. Bridged Tee Resistive Circuit to be analyzed



A. Write node (KCL) equations at node Vx and node Vo in the circuit of Fig. 1. Solve using MAPLE. Include your Maple Worksheet as ***Attachment A***.
B. Define *clockwise* circulating mesh currents in the circuit of Fig. 1 (I1 in the lower left mesh, I2 in the lower right mesh, and I3 in the top mesh). Write mesh (KVL) equations around each of the three meshes. Then use MAPLE to solve for I1, I2, and I3 using MAPLE. From these I1, I2, and I3 values, determine Vx and Vo from these mesh currents. Include your Maple Worksheet as ***Attachment B***.
C. Simulate the circuit of Fig. 1 using Orcad PSPICE. Use a "***VDC"*** voltage source for the battery. In the "***Simulation Profile***" window, select "***Bias Point***" analysis, then close this window and click on the "***Run PSPICE***" arrow icon. After the simulation runs, close the blank window that appears so you can once again view your schematic. If necessary, click the "***V***" and the "***I***" buttons to display the dc ("bias point") node voltages and the element currents in the circuit. Print this schematic diagram ***with the currents and voltages displayed***, and include as ***Attachment C.***

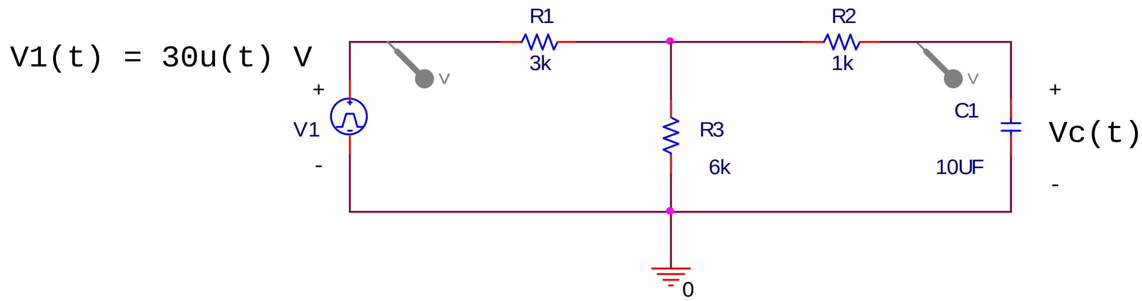
NOTE: ***The results of Parts A, B, and C MUST agree if you are to get any credit for this lab project. If they do not agree, be sure to get help before you leave the lab. Fill in the blanks below with the answers obtained in each of the three parts above:***

Vx = ________________ ***Vo =*** _____________

Part 2. RC circuit transient analysis, Thevenin reduction, and SPICE "Time Domain (transient)" simulation

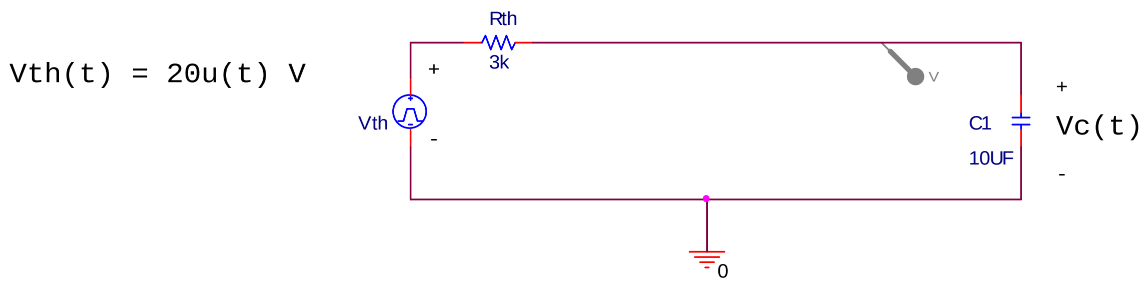
A. Consider the RC circuit shown in Fig. 2. It is driven by step voltage source v1(t) = 30u(t) Volts. First find the Thevenin equivalent of the circuit that would be seen "looking out" from the capacitor's terminals.

Figure 2. First-Order RC Circuit with step input voltage to be analyzed



Show, in ***Attachment D*** that this Thevenin equivalent may be drawn as in Fig. 3.

Figure 3. "Theveninized version of Fig. 2"



Also show in ***Attachment D*** that the voltage across the capacitor, Vc(t) is given by
*Vc(t) = 20(1-exp(-t/(Rth*C1)))* for $t > 0$, and also use MAPLE to plot Vc(t) for $0 < t < 0.1$ second.

B. Now perform a SPICE transient simulation on the (original) circuit of Fig. 2. Use a "***VPULSE***" pulse train voltage source (since SPICE has no single step source). Set it for a low voltage ***V1*** = 0V, a high voltage ***V2*** = 30V, the rising edge of the pulse should be delayed from zero by ***TD*** = 0s, the pulse should rise in ***TR*** = 0s, and fall in ***TF*** = 0s, the time the pulse is high, or the pulse width, should be set to ***PW*** = 0.2s (or anything greater than 0.1 s), and set the pulse repetition period ***PER*** = 0.4s (or anything greater than 0.2s), which is the time before the pulse rises back high again, starting a new cycle. In the Simulation Profile window, select ***"Time Domain (Transient)"*** simulation. The "***Run to***" time should be set to 0.1s, and the "***Start Saving Data after***" time should be set to 0, and the "***Maximum Step Size***" should be set to 0.001s. Place a voltage probe (by clicking on the voltage probe icon above and then dragging the probe to the desired node in the schematic) on the V1(t) node, and also place a second probe on the Vc(t) node, as shown in the original schematic diagram above.

To learn how to work with the "cursor" tool and the "Mark Cursor" button, click on the "Toggle Cursor" button (the 13th button from the left along the top of the plot window). Then select the output voltage trace, which is probably labeled "V(R2:2)" by clicking the symbol just to the left of its label so a highlighted.box appears around it. Then X-Y (horizontal and vertical) cursors should appear, and will follow the output waveform as the mouse cursor is dragged horizontally back and forth by pressing down the LEFT mouse button. Note that a "Probe Cursor" box also appears, and the first line of this box should read "A1 = xxxx, yyyy", where "xxxx" represents the horizontal coordinate (time) and "yyyy" represents the vertical coordinate (probe voltage). Note for future reference that if you now press down the RIGHT mouse button and drag the cursor horizontally back and forth a second pair of cursors will show up, and the position of this second set of cursors appears on the second line of the "Probe Cursor" box, which should read "A2 = uuuu, vvvvv"; where "uuuu" corresponds to the horizontal coordinate (time) and "vvvv" corresponds to the vertical coordinate (probe voltage). The third line of the Probe Cursor box should read "dif = "mmmm, nnnn", where mmmm = xxxx – uuuu and nnnn = yyyy – vvvv, and is therefore useful for calculating slopes between the two coordinate points when plots are approximately linear.

In this lab project, we will use only one set of cursors. Left click the mouse and drag the cursor pair out to where the voltage has risen to (1 – exp(-1)) * 20 = 12.64 V (you may not be able to get this voltage precisely, since PSPICE only computes discrete output voltage samples). The corresponding observed time should be quite close to time t = Rth*C = one time constant, which in this case should equal 3E3 * 10E-6 = 30 ms. Once you have positioned the cursor as closely as possible to the desired vertical coordinate, check to make sure that your horizontal coordinate is indeed close to the expected value of 30 ms. Then click on the "Mark Label" button, which is the rightmost button above the plot window. This should cause the precise coordinates of the marked point to be labeled on the plot! After you toggle OFF the cursors (by hitting the "Toggle Cursor" button once again) you should be able to drag the marked coordinates to a more convenient position. You can also right click on the "callout arrow" to change its position as well.

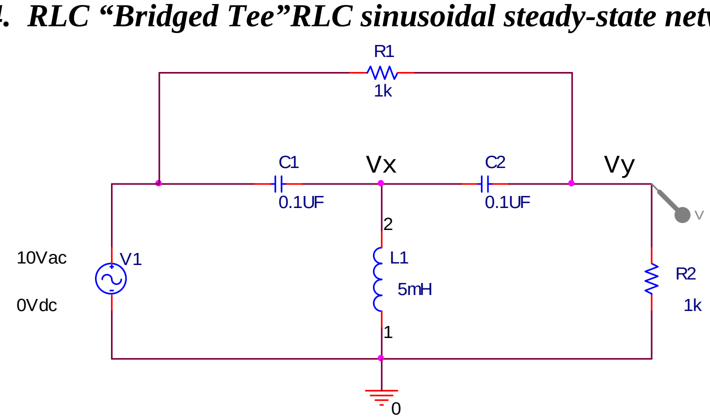
Include in ***Attachment E*** both your PSPICE schematic diagram and also your simulation plot that shows both V1(t) and also Vc(t). Be sure that the coordinates corresponding to one time constant are also marked on this plot, as explained in the previous paragraph!

Note: to receive <u>any</u> credit for this lab, the plots of Vc(t) in both Attachment D and Attachment E <u>must match precisely</u>, and the coordinates of Vc(t) corresponding to 1 time constant must be clearly labeled on your plot!

Part 3. AC phasor sinusoidal steady-state analysis and PSPICE "AC Sweep" simulation

Consider the third-order RLC "bridged Tee" circuit that is shown in Fig. 4.

Figure 4. RLC "Bridged Tee"RLC sinusoidal steady-state network to be analyzed



A. Write phasor node (KCL) equations for the network of Figure 4, and use MAPLE to solve for the complex-valued phasor voltage as a function of angular frequency, **Vy**(jω). Recalling that angular frequency ω, in rad/s, is related to linear frequency f, in Hz, by the formula ω = 2πf; use MAPLE to plot the polar magnitude of Vy(jf), or *abs(Vy(I*2*Pi*f))*. Also use MAPLE to plot the phase angle (in degrees) of Vy(jf), or in MAPLE, evalf(*180/Pi*(argument(Vy(I*2*Pi*f)))* over a range of frequencies extending from f = 1 Hz up to f = 20,000 Hz. Include your MAPLE worksheet (with MAPLE plots) in ***Attachment F***.

B. Perform a PSPICE "***AC Sweep***" simulation of this circuit of Fig. 4. Choose the VAC voltage source, setting its (peak) amplitude to 10V and its dc offset to 0V. Place a voltage probe on the Vy node, as indicated in the figure above. In the simulation settings window, choose the "AC Sweep/Noise" simulation, and set the starting frequency to 1Hz, ending frequency to 20000Hz, and number of points to 200. Do not enable the noise analysis section. When you click the "Run PSPICE" button, the magnitude of Vy(jf) will be plotted. To get the phase angle of Vy(jf), you must click on ***Trace – Delete All Traces***, and then click on ***Trace – Add Trace*** to bring up the "***Add Traces***" window. Then, in the "***Trace Expression***" box at the bottom of this window, add a trace that corresponds to the phase angle of the desired voltage Vy(jf) = V(R2:2), using the "PSPICE phase function, P( )" by typing "***P(V(R2:2))***". Note that the "***R2:2***" indicates "Terminal #2 of resistor R2". If you had typed "***R2:1***" (Terminal #1 of resistor R2) you would have gotten no phase variation, since Terminal #2 of R2 is at ground (0 V) potential! Include your PSPICE schematic and also your PSPICE magnitude and phase plots in ***Attachment G***.

Once again, in order to get any credit for this lab, you must demonstrate close agreement between your MAPLE plots and your PSPICE plots.

The only difference between your PSPICE and MAPLE results is that the MAPLE phase plot is plotted with a "branch cut", where a 360 degree phase shift is inserted by MAPLE to keep the phase variation to within the -180 to +180 degree range.

Your Lab 1 Report

Each person must submit an individual lab report for this lab. The report is due by the beginning of the next lab period. Include this lab handout as the beginning of your lab report. Then staple (in proper order) each of the required attachments (Attachments A – G). Each attachment should be properly labeled (using hand lettering is fine) with sequential attachment letter *and also an appropriate caption*. For example, your first attachment might be labeled "***Attachment A. DC Analysis of the Resistive Circuit of Figure 1 using MAPLE***."