Name \_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_Avery Peiffer\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_\_

ECE 0257 Quiz #1 Reflection

You should have received an email containing a link to your graded quiz #1. This exercise is designed to encourage to reflect on your results and identify gaps in your knowledge of circuit theory.

Instructions

* Briefly review your quiz and identify problems that you answered incorrectly (and problems that you didn’t know how to solve, but guessed correctly or were unsure about!)
* Use PSPICE to model and simulate the problem
* Based on the simulation results, re-examine / re-solve the problem and try to identify where you made a mistake or gaps in your knowledge
* Fill out the worksheet on the following pages.
* This quiz will be re-graded in its entirety. Even if you previously answered a question correctly, re-enter your final answer.

Final Answers

Problem #1

A

Problem #2

A

B

Problem #3

B

Problem #4

A

Problem #5

C

Problem #6

A

\*One way to simulate this problem is to use a VPULSE source to model the input voltage. Time t = 0ms is relative and can be taken to be the time at which the input switches from a low to a high voltage. Make sure that the rise/fall times of the VPULSE source are small in comparison to the period you decide to use.

Problem #7

B

\*This problem can be modeled using a Piece-Wise Linear (PWL) source, a voltage source where time/value points are specified. You can find this component under Place🡪PSPICE Component🡪Modeling Application🡪Sources🡪PWL Sources. In the simulation environment, you can use the measurement tools to automatically determine the period and/or frequency of a signal if you go to Trace🡪Measurements, select the measurement and signal of interest and then click EVAL

Problem #8

B

Similar to problem #7, the simulation measurement tools can be used to analyze a waveform. A sinusoidal voltage source can be used (VSIN) to model the input. The RMS value of a signal can be plotted by going to Trace🡪Add Trace, selecting RMS as the measurement and the signal that you want to plot the RMS value of.

Problem #9

C

To model this problem, you can create a simple circuit with a voltage source and a capacitor. By looking at the current through the capacitor for a given voltage, you can determine the capacitor’s impedance.

Problem #10

A

Problem #11

B

\*This problem can be modeled in a manner similar to what was shown in the tutorial.

Problem #12

C

\*Steady state are DC conditions. You can model this problem by looking at a DC bias point or by using a transient simulation with a simulation duration that is large in comparison to any time constants in the circuit

Problem #13

A

\*Op-amps can be found under PSpice Component🡪Discrete🡪OpAmp. Amplifiers are easily verified in the time domain. You can model this problem by driving the amplifier with a sinusoidal source and then directly measuring Vout/Vin at any instance in time. You can also use the simulator to plot Vout/Vin

* Being able to analyze your quiz and reflect on your errors is an essential part of the learning process. For the quiz analysis/reflection, please answer the questions on the following page.
* After you complete this worksheet, save it as a PDF document and upload it to courseweb.

Explain the source of any differences between your quiz responses or calculations and the simulated results.

N/A

How can you use this comparison experience to improve in the future?

Even for the questions I answered correctly, it is useful to reflect on my answers and revisit my methods for solving those questions. For some of the questions, I did not have an immediate intuition as to how to solve the problem – I instead had to think about the parameters of the problem and come up with a line of reasoning that I believed was accurate and led to a correct answer. It is helpful to revisit these questions and know that my logic was correct, meaning I can continue to apply it to future problems.