Lab 1 – LTspice Simulations and Circuit Analysis

CE-3101/021 Digital Elex. and Comp. Interfacing

By:

Andrew Iliescu

3-27-2020

**Abstract:**

In today’s day and age, circuit simulation and analysis tools exist to help us build, run, and analyze circuits. One such tool is LTspice. These tools are incredibly beneficial, because they allow people to compare hand calculations carried out by standard circuit analysis techniques based on Ohm’s Law and Kirchhoff’s Laws to these digitally built circuits. In this lab, we used LTspice to model two separate circuits and subsequently solve for certain unknowns. These unknowns include, the voltage at a certain point in the circuit, the voltage over time, and the power generated/dissipated by each component.

**Methods:**

A close up of a clock

Description automatically generated

Figure 1: Circuit 1 diagram, taken from Ms.Varnell’s CE3101 Lab 1 document

A picture containing clock

Description automatically generated

Figure 2: Circuit 2 diagram, taken from Ms.Varnell’s CE3101 Lab 1 document

The circuits in figures 1 and 2 were implemented in LTspice (figures 7&8) and subsequently analyzed. For the circuit in figure 1, the *.op* command was used in order to calculate the value of the voltage at node 2. For the circuit in figure 2, the *.trans* command was used to sample the voltage every 0.01ms from 0ms-1ms. Then the voltage across each component was plotted in LTspice over this time interval. Also, the power was calculated in the same manner for each component and then plotted against time in Excel.

**Results:**

After conducting the lab, we attained all of the aforementioned data and completed the analysis on the two circuits (figures 1&2). Starting with the circuit in figure 1, we wrote a .cir file where we created the components of the circuit in LTspice and the proceeded to simulate the circuit in order to get the values for the voltage at node 2. The results of the simulation are shown in figure 3. We then went ahead and verified the results by using the old school, pen and paper method, and I used the node voltage analysis technique. The calculation and the answer are both show in figure 4. Both methods resulted in node 2 being equal to 2 volts.

A screenshot of a cell phone

Description automatically generated

Figure 3: Results from .op command (in LTspice) for circuit 1

A close up of text on a white background

Description automatically generated

Figure 4: Node voltage hand calculations for node 2 in circuit 1, where Vx is the voltage at node 2

After performing these calculations on circuit 1 (figure 1), we moved onto circuit 2 (figure 2). For this circuit, the analysis was slightly different than what needed to be done in circuit 1 (figure 1). We needed to create a model of the circuit in LTspice and instead of merely calculating the voltage at one node, we were tasked with plotting the voltage over time for each component in the circuit, and the power dissipated by each of the components in the circuit. I marked up figure 2 with labels and gave the resistors arbitrary pluses and minuses in order to help with the circuit analysis so that we may define a direction for the current. These directions don’t matter because even if they are wrong, it only means that we get a negative result instead of a positive one. Now by sampling the voltage every 0.01ms from 0ms-1ms we were able generate the plot shown in figure 5 that has the voltage across each resistor over the time. The 5Ω resistor is depicted as the black graph because in order to calculate the voltage across said resistor, you need to get the difference between the two nodes that encompass it, that being nodes V1 and V2. Since we defined the resistor to have a plus on the left (towards V1) and a minus on the right (towards V2) we need to subtract V2 from V1 to get the voltage at each instant that A close up of a map

Description automatically generatedthe resistor gets sampled. The same approach is done for the 1Ω resistor, which is the blue graph.

Figure 5: Graph of voltage through the resistors as a function of time for circuit 2

The next portion of the analysis for circuit 2 (figure 2) was to get the power that was dissipated through each resistor as a function of time. To do this, we simulated the circuit in LTspice and then exported the data into an excel file. From there we were able to graph the power as a function of time for each resistor and the resulting graph is shown in figure 6. Since power is positive through the resistors, it means that the resistors were dissipating the power, and that means that the voltage source had negative power since it was generating it.

A close up of a map

Description automatically generated

Figure 6: Graph of the power through the resistors as a function of time for circuit 2

Now the lab also posed some questions which need to be answered, and these are:

1. **Do the voltages across the components in part A) satisfy Kirchhoff’s voltage law? Explain.**

The voltages across the components in figure 1 do satisfy Kirchhoff’s Voltage Law, because as the law states, the sum of all voltages in a closed loop need to be zero. If we refer back to figure 3, we can see that the sum of V(1), V(2), and V(3) is equal to respectively, which is in fact equal to 0 thereby satisfying KVL. To link these voltages back to the circuit in figure 1 they are as follows, V(1) is the voltage source V1, V(2) is the voltage at node 2, and V(3) is the voltage source V2.

1. **Do the currents across the components in part A) satisfy Kirchhoff’s current law? Explain.**

The currents across the components in figure 1 do satisfy Kirchhoff’s Current Law, because the law states that, the total current entering a circuits node has to be equal to the total current leaving that same node. If we refer back to figure 3 and take a look at node 2, we can see that the current entering the node comes from I(R1) or the current coming from resistor 1, and the current leaving that same node splits in two directions towards R2 and R3, and this results in I(R2) and I(R3). By KCL we know that the current leaving the node needs to be equal to the current entering the node which means that I(R1) = I(R2) + I(R3). Therefore , which it is, thereby satisfying KCL.

1. **Explain the power plot from part B) using the law of conservation of energy.**

The Law of Conservation of Energy states that the total energy in an isolated system needs to remain constant. For our purposes this means that the net power in our circuit needs to be equal to 0 otherwise we would be either gaining or loosing energy in our isolated system. The graph in figure 6 shows the power across the resistors in our circuit, and we can see that the power is in fact positive because it is being dissipated by the resistors and turned into thermal energy. The power across the voltage source at this time is the negative equivalent of the sum of the power across the two resistors, because as stated before the net power needs to be 0 to uphold the conservation of energy, and we know that it is negative because the voltage source generates power. As the voltage changes over time in a sinusoidal manner, so does the power across all of the components and it changes with respect to the voltage in order to maintain the Law of Conservation of Energy.

**Summary:**

After conducting this lab and going through the directions step by step, the most important results are as follows: we can use LTspice to simulate circuits and have it do the calculations for us, we can see how voltage and power are sinusoidal and affect each other, and we can visually see how KCL, KVL, and the Law of Conservation of Energy are preserved.

**Appendix:**

This section contains additional information pertaining to the lab.

A screenshot of a cell phone

Description automatically generated

Figure 7: This is the SPICE code used to model circuit 1

A picture containing clock

Description automatically generated

Figure 8: This is the SPICE code used to model circuit 2