Introduction

LTspice is a free and powerful circuit simulation software application that can be downloaded here: http://www.linear.com/designtools/software/ The software is already loaded on the EET lab computers, but you may want to consider putting it on your home computer as well.

This software allows users to build virtual circuits in order to predict how a particular circuit might behave before, or instead of, physically building it. It can be an aid in circuit design, and it can also be used to test the circuit analysis techniques you will learn in this class. For example, if you are given a homework assignment that gives you a circuit drawing (schematic) and asks you to calculate the voltage across a particular resistor, you can build the circuit in LTspice, run the simulation, and make a virtual voltage measurement to see if your theoretical calculation was correct.

Many other circuit simulation programs exist, but LTspice is free, very capable, and is relatively simple to learn. It is also easy to save your work to a transportable file and the program can be run from a USB stick.

Part A: Schematic Capture

Schematic capture is the process of putting your circuit drawing (schematic) into a computer. After the virtual circuit is built, the user can then simulate the circuit to see how the circuit behaves. In this lab, we will want to draw (capture) the circuit below so that we can "measure" voltages and currents.

Figure 1
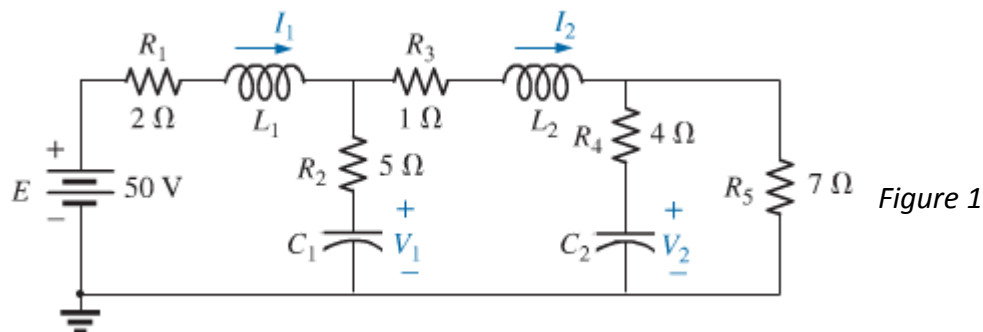
The diagram above is a circuit schematic. It utilizes standardized symbols for common circuit components. Pictured above are resistors ⎍⎍⎍, capacitors ⊤, inductors 000, and there is one voltage source ⊤ . Do not worry about what function these components serve or how they work. For the remainder of the lab, this drawing will be referred to as our original schematic. The goal for this lab will be to capture this circuit and then simulate it to obtain voltage values for V1 and V2 (volts) and current values for I1 and I2 (amps).

1) Open LTspice and select the "New Schematic" button in the upper-left corner of the screen [icon] (or press ctrl+N). Press ctrl+G to make a grid appear/disappear. The grid is simply for a visual reference. Your screen should look figure 2.
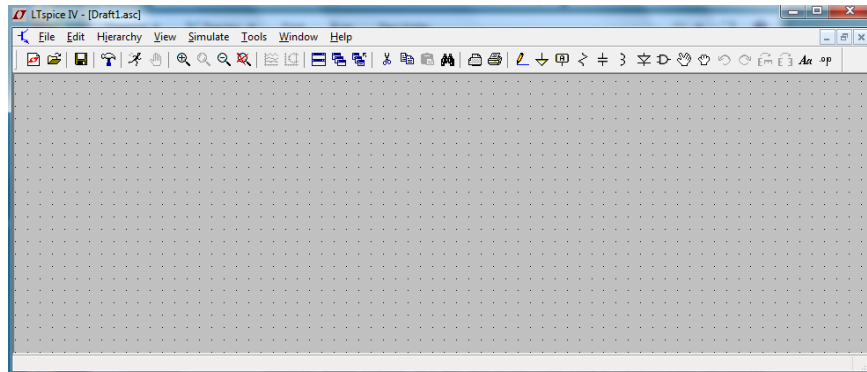

Figure 2

2) Now we will start at the left side with the voltage source. Click on the "Component" button [icon] and select "voltage" from the menu, as in figure 3.
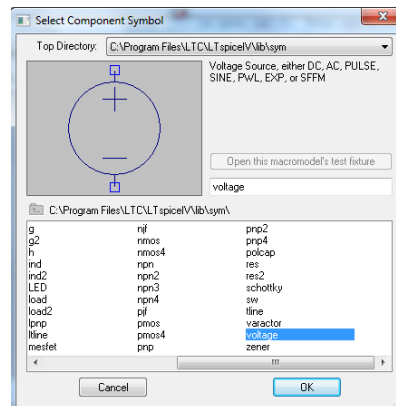

Figure 3

Click the "OK" button and place your voltage source on your schematic, preferably towards the left part of the screen to leave room for the rest of your components. You may notice that your voltage source symbol is different than that pictured in our original schematic. The symbol in the original schematic indicates, specifically, a DC voltage source, whereas the one pictured by LTspice is more generic and can be used to indicate an AC or DC voltage source. LTspice defaults to a DC source, so no further settings need to be changed.

You should notice that a "V1" and a "V" have appeared next to your voltage source. Right click on the "V1" box and you should get a dialog box titled "Enter a new reference designator for V1". A reference designator is simply a name given to a component to distinguish it from others like it. To match the original schematic, call the

voltage source "E" and click "OK". Use the "Move" button 🖐 to move the reference designator off to the left.

Right click the "V" box and enter a value of 50 in the "Enter a value for V" dialog box and click "OK". This lets LTspice know that this is a 50-volt voltage source. Be sure to *not* enter a "V" in this field. LTspice already knows the value represents a voltage, and uses letters in this field for representing powers of ten. Remember that r*eference designator* means *name,* and that *value* means *number*. You should now have a schematic like the one below.
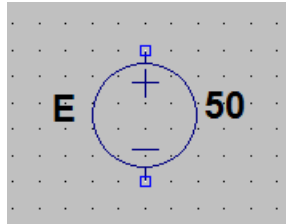

Figure 4

3) Now we will insert our electrical ground beneath the voltage source. This component tells LTspice what we are considering to be zero volts. It is the point on which all other voltage values will be based. Click on the ground symbol ⏚ (or simply type "g") and place your ground's dot (terminal) in line with and below the voltage source's bottom terminal, leaving about four grid dots in between the two terminals.
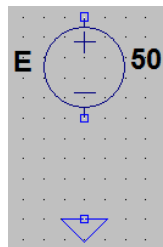

Figure 5

4) Connect the bottom terminal of the voltage source to your ground symbol using the "Wire" button ✏. Now the negative terminal of our voltage source is electrically connected to ground.

Figure 6

At this point, place another ground symbol at a random location on your schematic (not shown). Click the cut button ✂ or hit "delete". You should notice your cursor icon is now a pair of scissors. Click your randomly placed, unwanted ground symbol and you will see it disappear. Exit cut mode by either right-clicking or pressing the escape key.

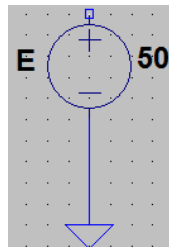**5)** Add a resistor representing R1 on our original schematic to your drawing by either typing "r" or clicking the resistor button ⌇ . Before placing, you should notice that the orientation of R1 in the original schematic is horizontal and that our new resistor is oriented vertically. Use the rotate function (ctrl+R) to reorient your new resistor so that it is horizontal. Change its reference designator to "R1" (if it isn't already called R1 by default) and its value to 2 ohms. Connect its left terminal to the positive terminal of the voltage source with a wire.
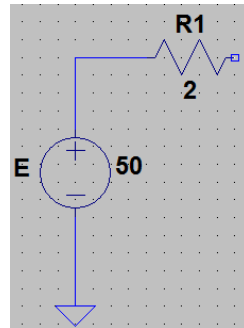
Figure 7

6) Use the capacitor button ⊥ (or type "C") and the inductor button ϟ (or type "L") along with the resistor button to place all of the remaining components with their correct orientations, reference designators, and values. Use the following values for the inductors and capacitors, as this information was not given in our original schematic.

L1: 2 mH (enter either "0.002" or "2m")

L2: 50 mH (enter either "0.05" or "50m")

C1: 20 µF (enter either "0.00002" or "20u")

C2: 5 nF (enter either "0.00000005" or "5n")

The letters "m", "u", and "n" are examples of how you can more easily enter powers of ten into LTspice. They are often more human readable than typing out the number in regular form. Note that LTspice automatically converts "u" to "µ".

You should have a circuit that looks like the one below.

Figure 8

7) Now that you have successfully captured your circuit, this would be a good time to save your work. Go to File→Save As, and save your file as "Lab_2_Circuit.asc". Choose a location that is convenient for you, such as the Desktop.

Part B: Circuit Simulation

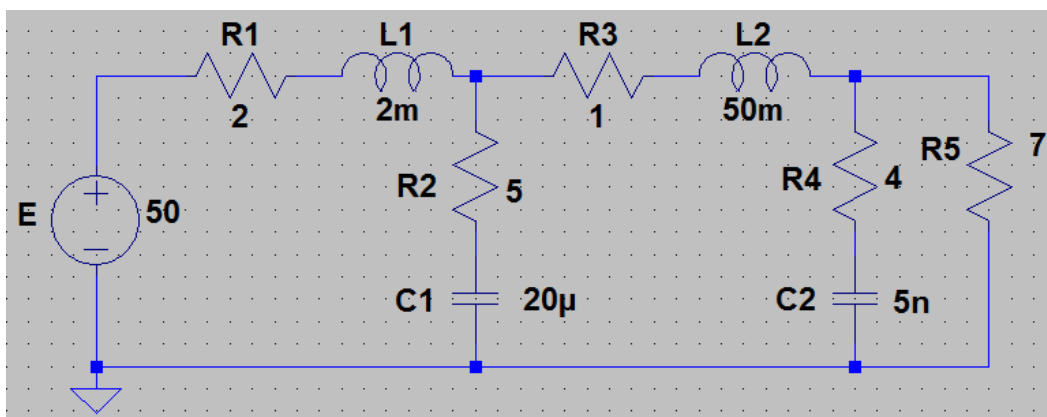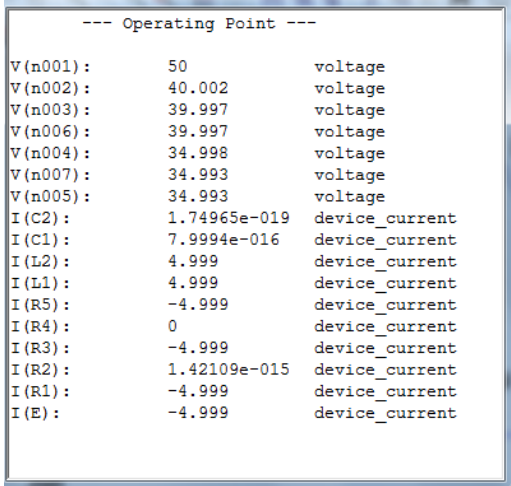With the circuit captured, it is now time to simulate the circuit and obtain values for V1, V2, I1, and I2.

1) Our first virtual measurement will be for the currents I1 and I2. Electrical current is measured in amperes, or amps, and flows *through* circuit components much like water flows through pipes.

We will be doing simple DC analysis for this lab, and leave more complicated simulations for future concern. Click the "run" button [icon] and go to the "DC op pnt" tab. Click the "OK" button and you should see a window of text pop up.

```
        --- Operating Point ---

V(n001):        50              voltage
V(n002):        40.002          voltage
V(n003):        39.997          voltage
V(n006):        39.997          voltage
V(n004):        34.998          voltage
V(n007):        34.993          voltage
V(n005):        34.993          voltage
I(C2):          1.74965e-019    device_current
I(C1):          7.9994e-016     device_current
I(L2):          4.999           device_current
I(L1):          4.999           device_current
I(R5):          -4.999          device_current
I(R4):          0               device_current
I(R3):          -4.999          device_current
I(R2):          1.42109e-015    device_current
I(R1):          -4.999          device_current
I(E):           -4.999          device_current
```

Figure 9

This window contains voltages for every node (or wire) in the circuit, as well as current through each component. Looking at the original schematic, I1 is the current flowing through the inductor L1. In our data table, the current through component L1 is indicated by the row starting with" I(L1)". This table states that there are 4.999 amps going through this inductor. In this case specifically, current is flowing from left to right (using conventional current flow).

Find and record the value for the current through L2.

2) For electrical current flow, direction matters. LTspice internally decides which direction is the positive direction flow through any circuit component. If you physically rotate inductor L1 by 180 degrees, you will see that LTspice will give you a value of -4.999 amps, as opposed to 4.999 amps before.

Use the move button [hand icon] and "lift" up inductor L1. Rotate it twice by pressing ctrl+R twice. Place it back into the circuit and perform the simulation again. This time you should see a value of -4.999 amps passing through L1. The current is still going from left to right, but this time it is reported as negative. This positive/negative current discrepancy can be a source of confusion when simulating a circuit.

3) Voltage is a quantity that is measured *across two points*. On the original schematic, you can see that the voltage V1 is defined to be across capacitor C1. This is denoted with the "+" and "-" symbols. This means V1 is the voltage difference between the node just above C1 (call this node "V1") and the node just below C1 (this node is already our ground, and thus it is automatically zero volts).

From the previous simulations, you could see that voltages were reported for seven different points, nodes n001 through n007. These numbers will not mean much to us unless we know the physical location of these nodes. Since we did not label any of our nodes (or wires), LTspice picked default names for us, being n001, n002, and so forth.

To make these voltages more meaningful, we will label nodes with our own names. Click on the "label net" button [button icon], enter "V1" next to the "ABC" box, and click "OK". Drop the *dot* of the resulting symbol on the *wire* directly above C1. You should not see the node label's dot.
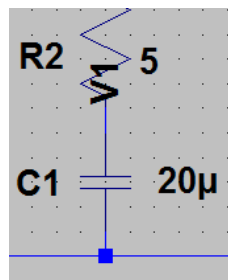
Correct:

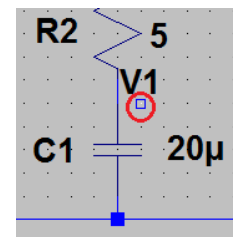*Figure 10*

Incorrect:

Figure 11

Use this process to place the label "V2" just above capacitor C2 and run the circuit simulation again. You should obtain data matching the data below.

```
            --- Operating Point ---

V(n001):        50              voltage
V(n002):        40.002          voltage
V(n003):        39.997          voltage
V(v1):          39.997          voltage
V(n004):        34.998          voltage
V(v2):          34.993          voltage
V(n005):        34.993          voltage
I(C2):          1.74965e-019    device_current
I(C1):          7.9994e-016     device_current
I(L2):          4.999           device_current
I(L1):          -4.999          device_current
I(R5):          -4.999          device_current
I(R4):          0               device_current
I(R3):          -4.999          device_current
I(R2):          1.42109e-015    device_current
I(R1):          -4.999          device_current
I(E):           -4.999          device_current
```

Figure 12

Now "V(v1)" and "V(v2)" appear in the data table, giving us the voltages at our labeled nodes "V1" and "V2" as 39.997 volts and 34.993 volts, respectively. Voltage is a quantity defined across *two* points. If the voltage at node "V1" is 39.997 volts, where is the second point? By default, the second point is ground. This means the voltage between V1 and ground is 39.997 volts, and the voltage between V2 and ground is 34.993 volts.

4) In this step, we will make a design change and see what effect it has on the current I1. Change the value of resistor R1 from two ohms (2 Ω) to 2 megaohms (that is, 2 million ohms, or 2 MΩ). Using LTspice's abbreviations for powers of ten, try changing R1's value to "2M". Notices that after you click "OK", LTspice changes "2M" to "2m". We want two *million* ohms and LTspice changed it to two *milli*ohms. This is different by a factor of one billion.

This is a common mistake when entering in values into LTspice. The proper abbreviation for two megaohms is "2meg". Enter this into R1's value and simulate the circuit.

Recall that the current I1 from our original schematic is the current through the inductor L1. LTspice should give you a value of "-2.49999e-005". In LTspice's notation, this means $-2.49999*10^{-5}$ amps. How many microamps does this equal? How many milliamps does this equal?

Notice that making a design change, such as changing a resistor's value, is very quick and easy to do. We could see that changing R1 from 2 Ω to 2 MΩ, we saw that I1 dropped to a small fraction of its original value, all without reconstructing an actual circuit.