

EECE 2150 - Circuits and Signals: Biomedical Applications

Lab 3

Basic Instruments, Components and Circuits.

Introduction to KVL, KCL, and LTSpice

Introduction and Preamble:

In this lab you will experiment with some of the devices, instruments, and tools that we will use throughout the course. You will also begin using LTSpice, the widely used circuit analysis software. (You can even use this to check some of your homework problems!) Please ask about things you don't understand or are curious about.

As you do this and future labs, keep a detailed record in your lab notebook of what you are doing. You do not need to put down a formal list of procedures. However, you should diagram any circuit that you build and clearly indicate component values. Also record the results of measurements and sketch the waveforms you see on the oscilloscope. Make notes on what works and does not work and on your troubleshooting experiences you should write down what you did each day in lab in enough detail that a reader can understand your measurements, results, mistakes, discoveries, etc.

You should make sure that you answer all questions in this write-up and comment on the success, failure, and/or interpretation, of major tasks. The instructor or TAs will check your notebook at the end of each lab, sign the notebook to record that they have done so, and may offer you feedback and suggestions on content and format.

IMPORTANT: PRE-LAB PREPARATION

1. **Buy a bound laboratory notebook if you have not already done so!! This is important.** You will use this for all labs and part of your grade will be based on the work documented in your lab book. Each student should have their own notebook. Record the date at the beginning of each session and take a lot of notes on your experiments in your notebook. You will rely on these notes through the term. The pages in the lab notebook are where you record information in the lab, and where you work on circuit designs. It is not meant to be a finished piece of work like a lab report or a paper, or even homework. The notebook in this course is meant to function the same way a working engineer's notebook functions, as a complete record of the design and invention process, complete with dead ends, mistakes, corrections and insights. In the working world it is this richness that provides a record of invention that can even be used to defend a patent in court! You should not record information in the lab on other paper and then transfer it later to your notebook, except for any printouts of code or figures from MATLAB exercises or pictures you take, for example, of a finished ProtoBoard circuit or significant oscilloscope trace. The notebook should contain partial circuit designs, calculations, speculations, redesigns, revisions and corrections. This does not reflect poorly on your work; it is what is expected. It should be a complete, if rough, record of your work in the laboratory. It should contain corrections to designs based on experiments as well as results and comments on what was necessary to get the circuits to work.

2. Read Lab 3 (THIS DOCUMENT).

PART 1. LTSPICE MODELING

This semester, we will be using LTSpice. LTSpice uses a file called a netlist that is a list of components and connections that describes the circuit to be simulated. In the “old days” engineers would translate a circuit to the netlist for the simulation, and this can still be done. However, these days we generally draw or otherwise generate a circuit *diagram* and this circuit diagram is translated by the schematic capture program into the netlist used by LTSpice.

Use the LTSpice Program to draw and analyze a DC circuit. (After this is done, it will internally produce the netlist mentioned above.)

Note: You can download the software for your computer from <https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>. Download and follow the instructions to Install.

Either on your computer or the lab computer, start the LTSpice program. Then click on File>New>Schematic in the LTSpice window. Choose Save As from the File menu to specify the name and location of your

You should then have your LTSpice work area displayed (figure 1), with just a grid in the middle of the screen and various menus on the top and side. Click on the resistor symbol to place a resistor, click (Ctrl-R) to rotate and (Ctrl-E) to mirror if needed. Continue placing resistors and then hit the **Esc** key to exit resistor placement mode.

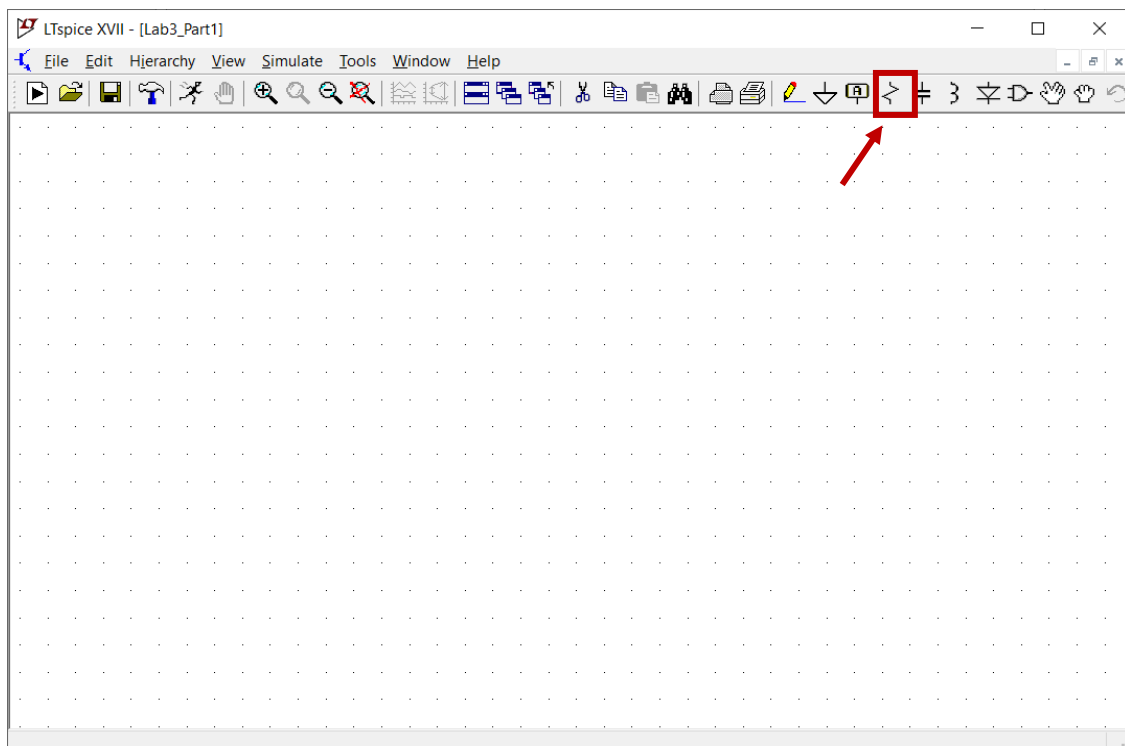


Figure 1: Illustration of how to choose components in LTSpice Schematic capture window.

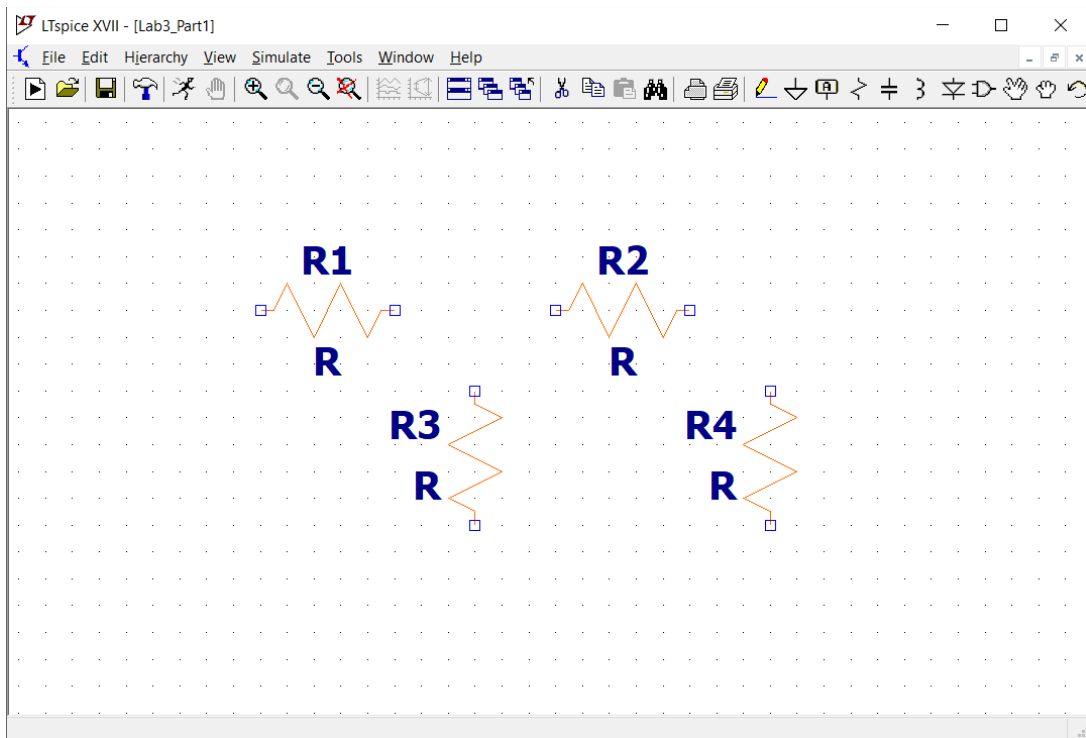


Figure 2: Illustration of schematic with four resistors placed.

You can right click on the resistor to change the resistance to enter or change the value. Click on the place component tool (AND symbol) to place the voltage source as shown in Figure 3. Right click to set the value of the DC source You will need to edit the values of the resistors and the voltage source as shown in Figure 4.

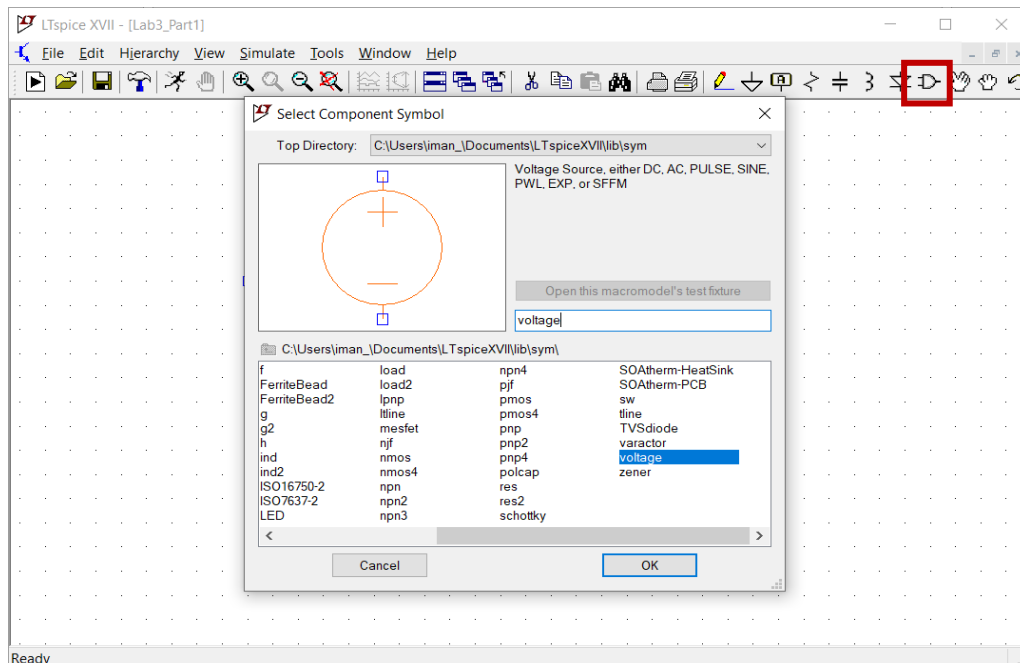


Figure 3: Adding the DC voltage source

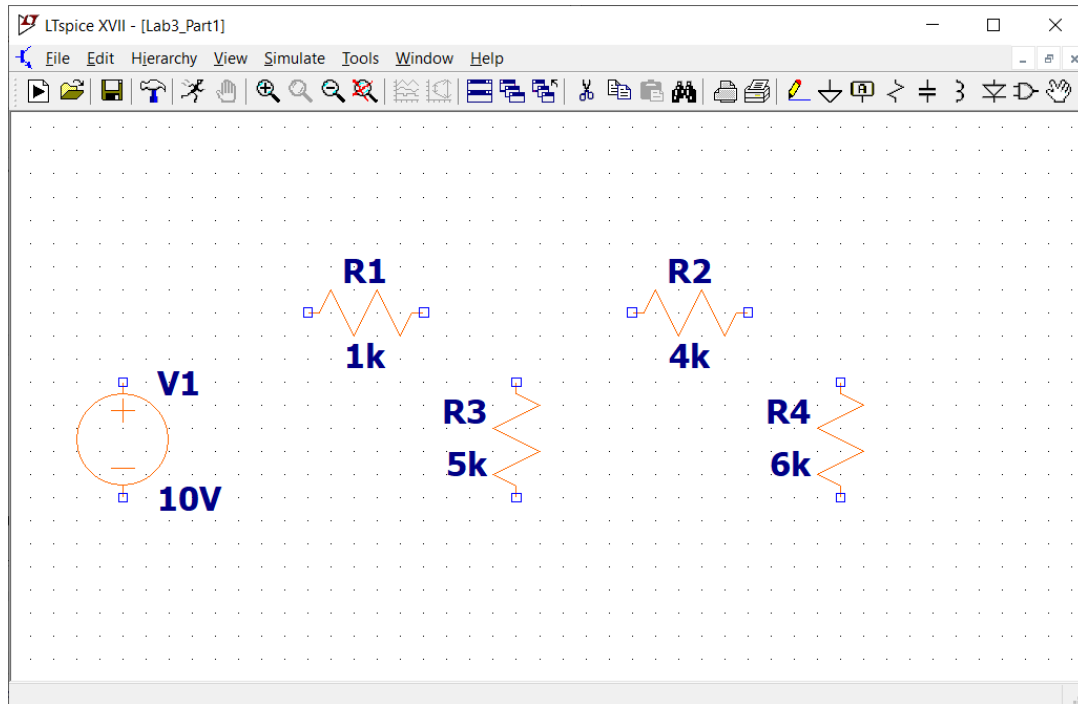


Figure 4: Right click on any component to change its value

Be sure to place a ground (ground defines the node in the circuit that is at zero Volts). The “place ground” button shown in Figure 5. Use the wire tool to wire the circuit.

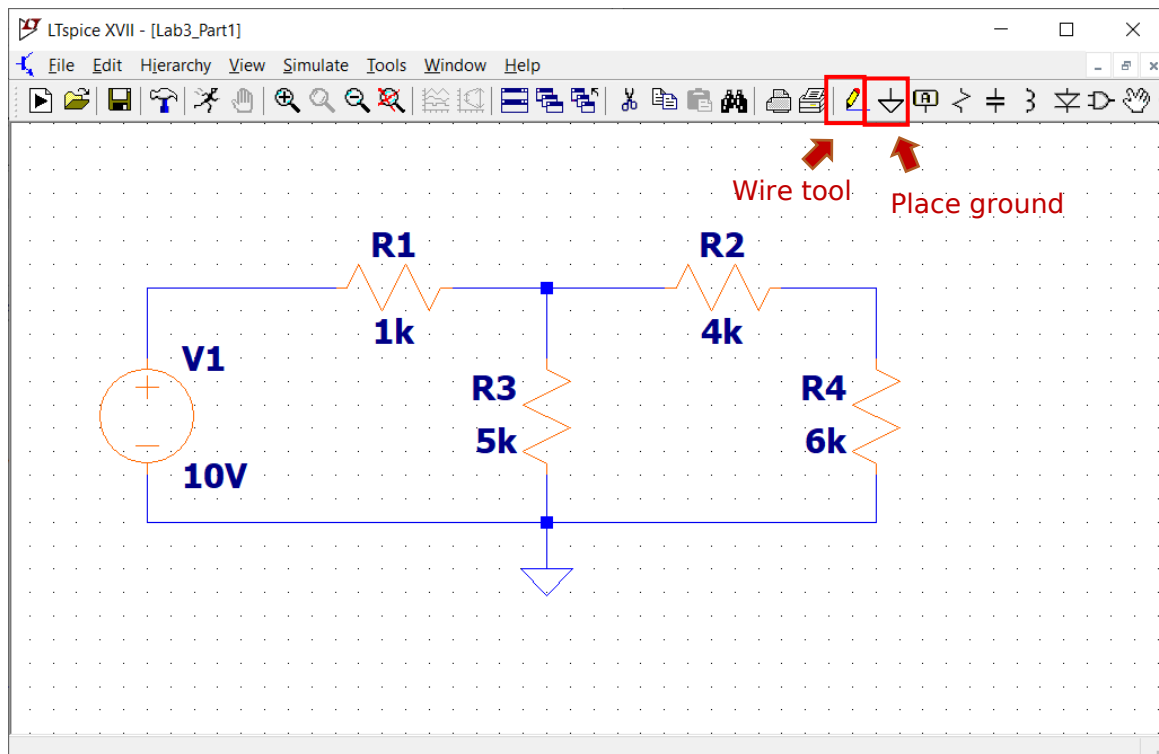


Figure 5: Placing Ground and wiring the circuit

Next, you need to specify what analysis you want LTSpice to do. Choose **Simulate>Edit Simulation Cmd** and choose DC Operating point as shown in Figure 6. Click OK, and then click on the schematic to place the simulation command, in this case '.op' for DC operating point

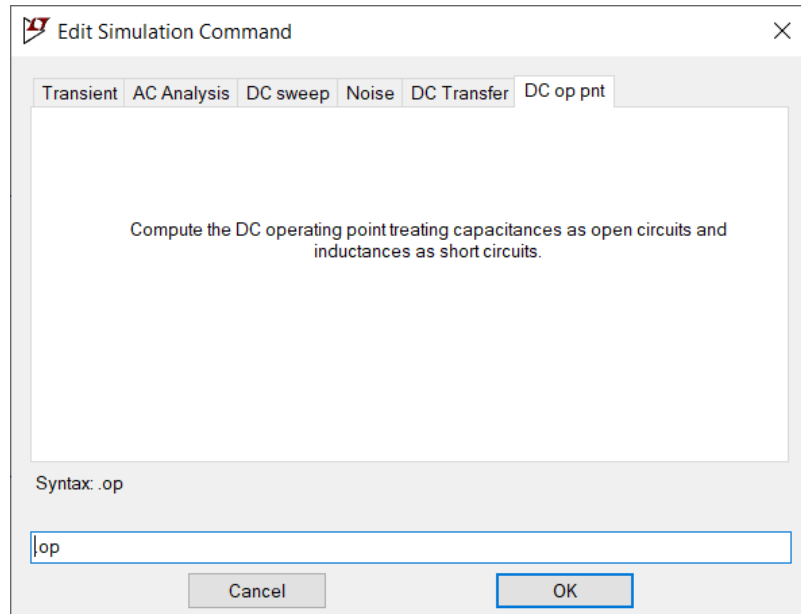


Figure 6: Choosing the simulation type for the circuit in Part I.

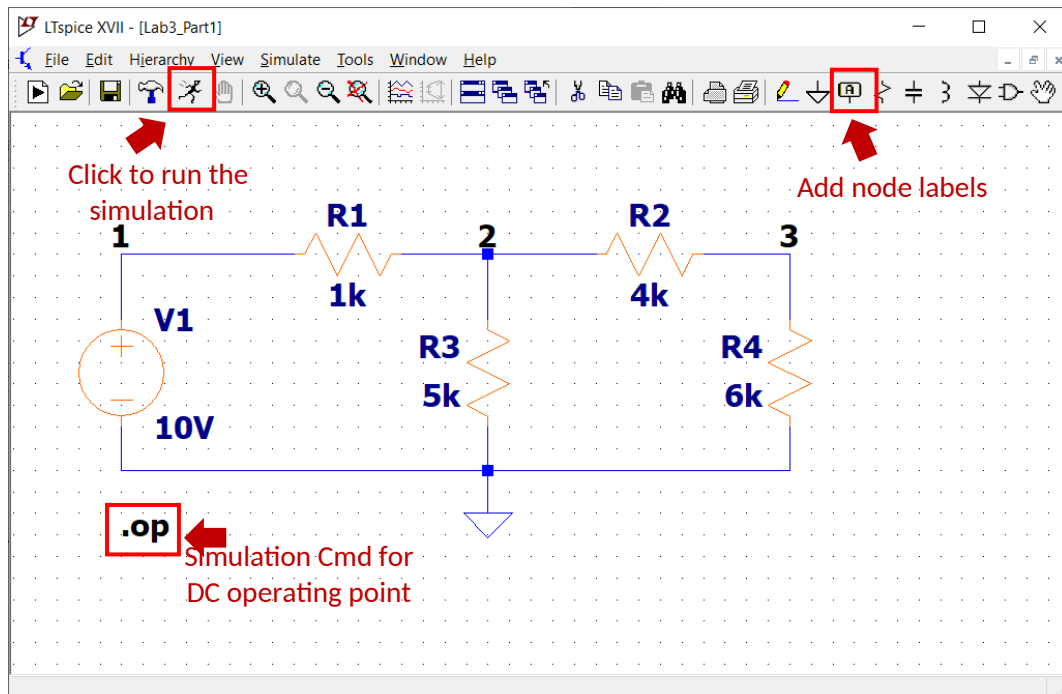
Figure 7: Complete circuit with simulation Cmd, '.op' added to schematic

You can add node labels to easily identify the nodes in the simulation results. Add labels by clicking the node label tool shown in Figure 7. You are now ready to run the simulation. Click **Simulate>Run**. A new window appears displaying all node voltages and branch currents. For this simple simulation, error messages may appear. The voltage of each node is relative to the ground node (which is assumed to be at zero volts (hence the 0 notation)).

PART 2 VERIFYING KIRCHOFF'S LAWS:

2.1 KCL: The sum of all currents entering a node is zero.

- a. Check KCL for all 4 nodes by summing the currents entering each node. Write down the results of the simulation and the KCL checks for all nodes in your lab notebook. Note



that a positive current leaving a node is equivalent to a negative current entering the node. How does LTSpice indicate the direction of the current? Hint: Change the voltage source to -10 V and see what the difference is.

- b. Do the currents at each node add up to zero as expected?

2.2 KVL: The oriented sum of all voltages around a closed loop is zero.

- c. The voltage across each component is the difference between the node voltages on the two sides of the component.
- d. Add up the voltages around each of the 3 loops in the circuit - do they all add up to zero? (This is kind of a trick question! If each node has a unique voltage, and the

voltage of a component is the difference of node voltages, this has to work – but that's Kirchoff's voltage law, and maybe this procedure gives you some insight into why it works.) How do you keep track of the polarities (the signs) of the voltages?

PART 3. AC Circuits

3.1 Configure the **Function Generator** to produce a 10 Hz **square wave** with 5 volt (10 V peak-to-peak) amplitude, as in Figure 8 below. Verify the signal on the oscilloscope using a BNC to BNC connector.

Note that the function generator will produce twice the peak-to-peak amplitude of the set value, when driving a high resistance load such as the oscilloscope. (It is configured by default to produce the set peak-to-peak value when driving a 50 Ohm load. You can change this configuration to 'high Z' to produce the set peak-to-peak value when driving the oscilloscope, but it is not necessary!)

As above, review the document "[Lab Components Basics](#)" on Canvas before proceeding. Full equipment manuals are also posted online.

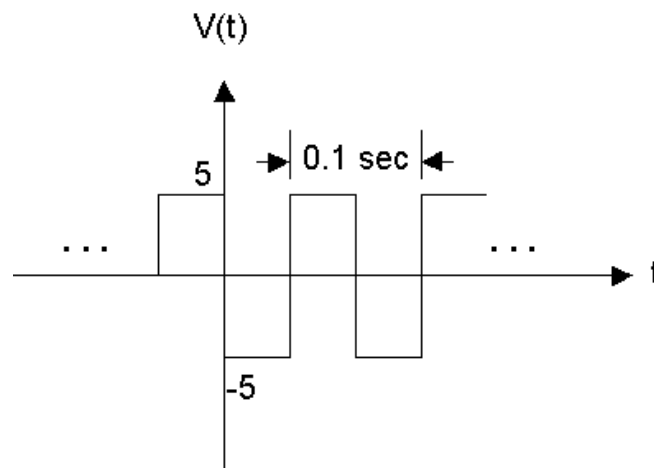


Figure 8. Diagram of desired output from the signal generator for Part 3.

3.2 Build the circuit shown in Figure 9 on your protoboard (use the protoboard worksheet if you find it helpful). **You will be using the function generator signal as the voltage source ($V(t)$).** Assuming that the diode voltage is 2V when the diode is on, calculate the resistance you need to limit the diode current to 20mA (to prevent overheating and damage). Both LEDs should blink. If not check the orientation (and review Lab1)

3.3 Change the square-wave frequency gradually from 10 Hz to 0.5 Hz. Should the two LEDs blink at the same time or not and why? Explain what is happening based on your experience of using the LEDs from earlier in the experiment.

3.4 Change the waveform from a square wave to a triangle wave to a sine wave. Comment on your results.

3.5 What is the highest sine wave frequency at which you can still see the LEDs blink on and off for the three waveforms? Is this frequency different for different people, for different light intensities, or if you change the waveform to square or triangle? What is your guess as to what causes this limit?

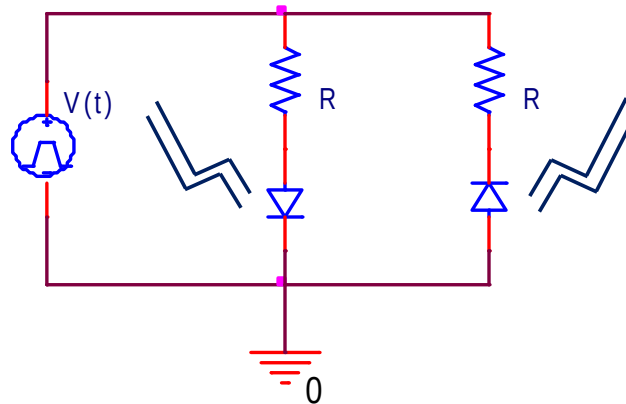


Figure 9. Blinking LED circuit for Part 3.

PART 4 – INSTRUCTIONS FOR THE WRITE-UP...

To be submitted on Canvas, one week after lab is finished:

- Answer all questions above.
- Follow instructions for writing lab reports available on Canvas
- Submit to the Canvas

Reports will be required for each lab. They will need to be submitted on Canvas one week after the session in which the lab work is finished. If you are not finished with the lab work, it is your responsibility to finish the work during any of the laboratory “office hours” during the week. Late Lab reports will be accepted with a 10% penalty up to one day late. When you have finished the lab work on a Lab, you should have your Lab Notebook (see below) signed off by a TA or the instructor who will ask a few questions to see if you have understood what you have measured.

IMPORTANT: BEFORE YOU LEAVE THE LAB:

- Place all of the components that you removed from the toolbox back in that box and return it to the cabinet that houses them
- Collect all used components and wires from your bench and place them in your group’s reusable plastic container. If you are not going to use these components or wires again, please discard them in the trash bin located in your lab room.
- Turn off all the equipment you have used on your workbench.
- Make sure you return your protoboard, the equipment wires and your reusable container to the cabinet.
- Make sure to have your notebook signed by an instructor or TA before you leave the lab.

Last updated 8/21/2021 I. Salama

Updated: 1/19/16 N. McGruer 8/11/15, M.Niedre, based on a version from 8/27/14, D. Brooks and from 8/18/12 by N. McGruer