EECE 2150 - Circuits and Signals: Biomedical Applications

Lab 3

Basic Instruments, Components and Circuits.

Introduction to Spice and AC circuits

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 Spice/PSpice, the powerful and 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.

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. As we have already discussed, 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.

As another reminder: You should record the date at the beginning of each session and take a lot of notes on your experiments in your notebook. 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. 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.

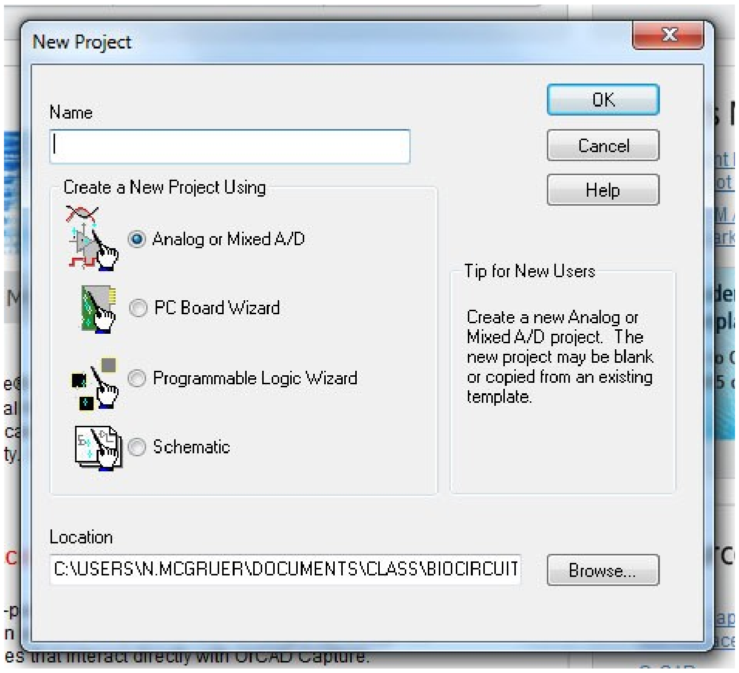
PART 1. Spice Modeling

Spice was originally written as open-source software in 1973 (before that term even existed). Today, it exists in many commercial and non-commercial flavors, and some are not even called Spice. This semester, we will be using OrCAD PSpice. PSpice 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 Spice, 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 PSpice.

Use the OrCAD capture program (***OrCAD Capture CIS Lite***) 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 Windows computer from the Cadence OrCAD Download site. Choose the OrCAD PCB Designer Lite Download and follow the instructions. After receiving an e-mail, this will get you to an OrCAD download site where you can download the demo software. You only need the Capture and PSpice, although you can download more if you want to. You should also be able to use PSpice on a Mac computer via the COE Virtual Lab. We have tested this briefly but have not used it extensively so if you try this and have trouble let us know and we can work with you to try to get it working---it may work well but it may also be a bit of an adventure!

Either on your computer or the lab computer, start the OrCAD capture program by clicking on the Start button>All Programs>Cadence>OrCAD 16.5 Lite> OrCAD Capture CIS Lite. Then click on File>New>Project in the OrCAD Capture window (or on New Project in the Getting Started sub-window). This should look like (Fgure 1):



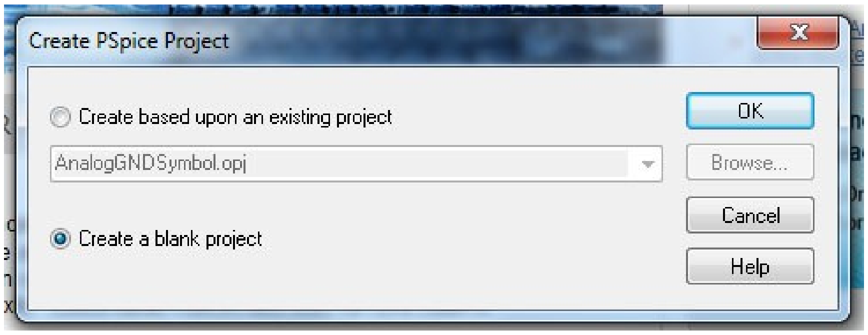
Create a new Analog or Mixed A/D project. The new project may be blank or copied from an existing template.

(●) Analog or Mixed A/D

Location: C:\USERS\N.MCGRUER\DOCUMENTS\CLASS\BIOCIRCUIT

Figure 1: Screen capture of PSpice New Project window.

Make sure you have Analog or Mixed A/D selected (***<u>not</u>*** Schematic), then type a name in the box at the top for your project and a location where you can store the simulation in the box at the bottom (you must have access to this on the computer – your desktop will work, for example, or a network drive that you have access to). Finally, click the OK button. This will bring up another window (Figure 2), where you want to create a **BLANK** project (<u>important</u>)!!

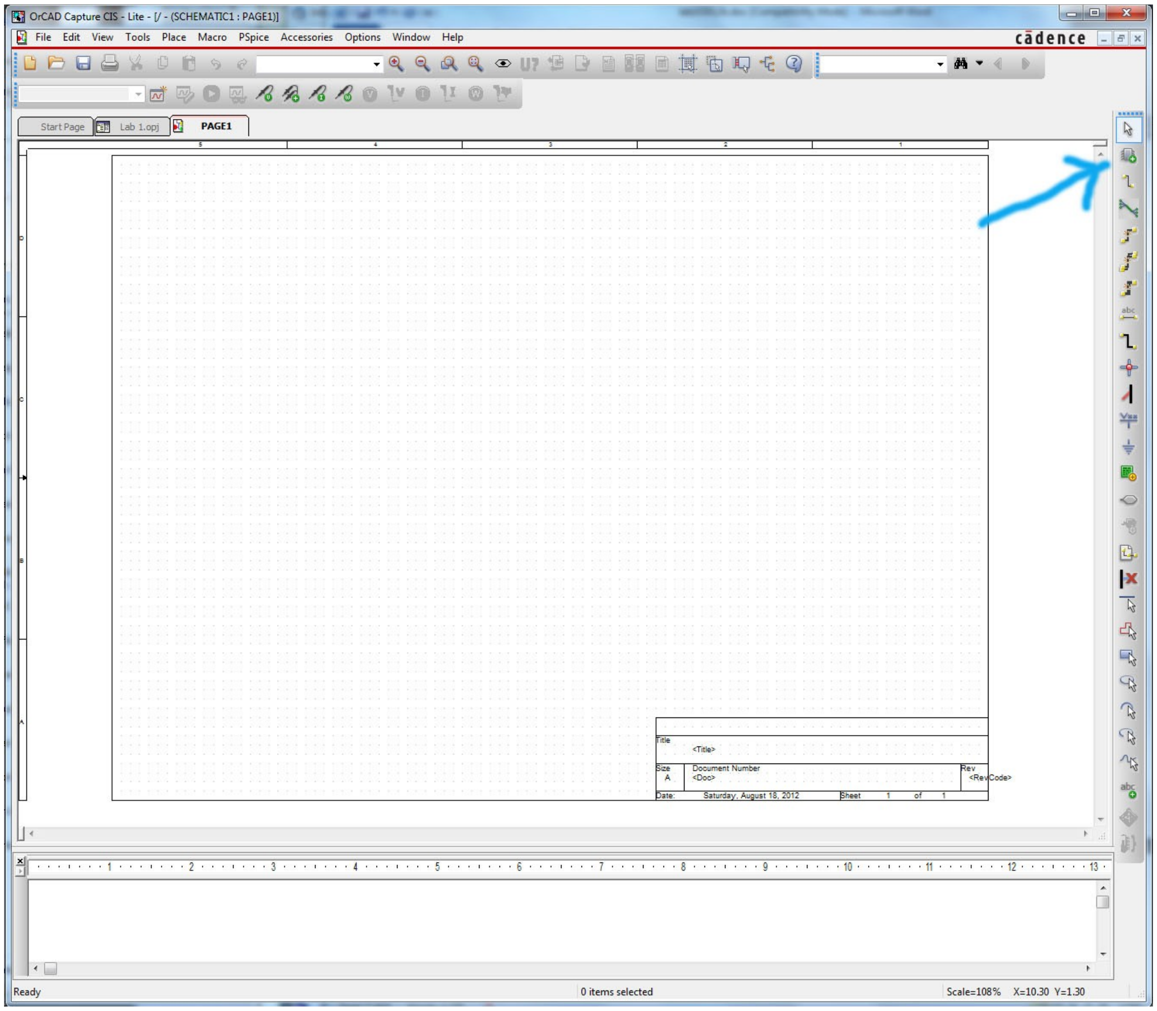


AnalogGNDSymbol.opj

(●) Create a blank project

Figure 2: Screen capture of PSpice New Project Step 2 window.

You should then have your Capture/PSpice work area displayed (Figure 3), with just a grid in the middle of the screen and various menus on the top and side. Use the button second from the top on the right to add components.


OrCAD Capture CIS - Lite - [/ - (SCHEMATIC1 : PAGE1)]
File Edit View Tools Place Macro PSpice Accessories Options Window Help
cādence
Start Page
Lab 1.opj
PAGE1

Title <Title>
Size A
Document Number <Doc>
Rev <RevCode>
Date: Saturday, August 18, 2012 Sheet 1 of 1

Ready
0 items selected
Scale=108% X=10.30 Y=1.30

Figure 3: Illustration of how to choose components in PSpice Capture window.

This should bring up a sub-menu on the right that will allow you to add parts. Depending on who has used the software before you, you may need to add parts libraries. If necessary, do this by clicking on the add library button. You will want ***analog***, ***source***, and ***eval***. Now click on ANALOG in the library box. A list of parts will appear. Click on R in the parts list (the resistor), then on the second ***place parts*** icon. Place the resistor where you want it in the schematic window (Figure 4). You can use Ctrl-R to rotate the resistor. Continue placing resistors and then hit the ***Esc*** key to exit resistor placement.

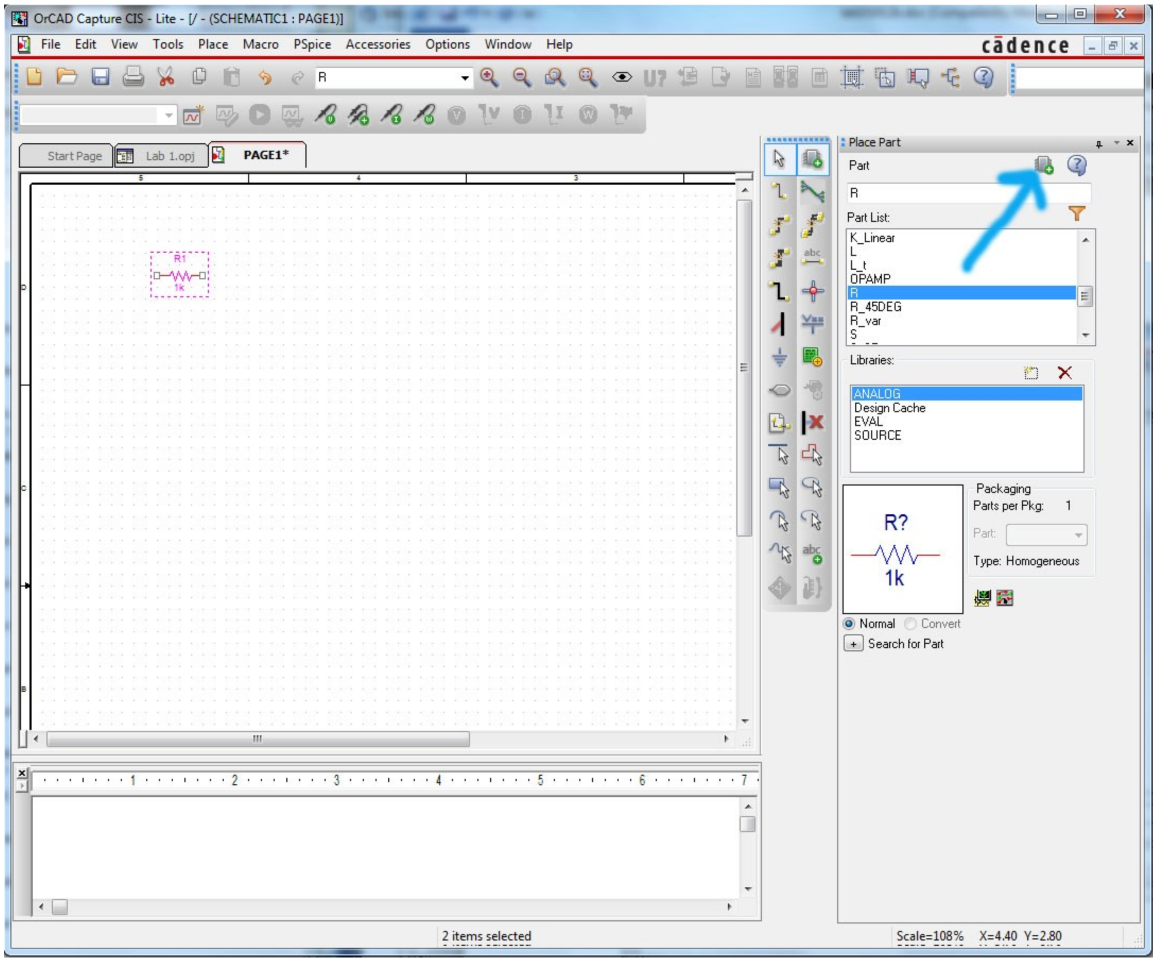
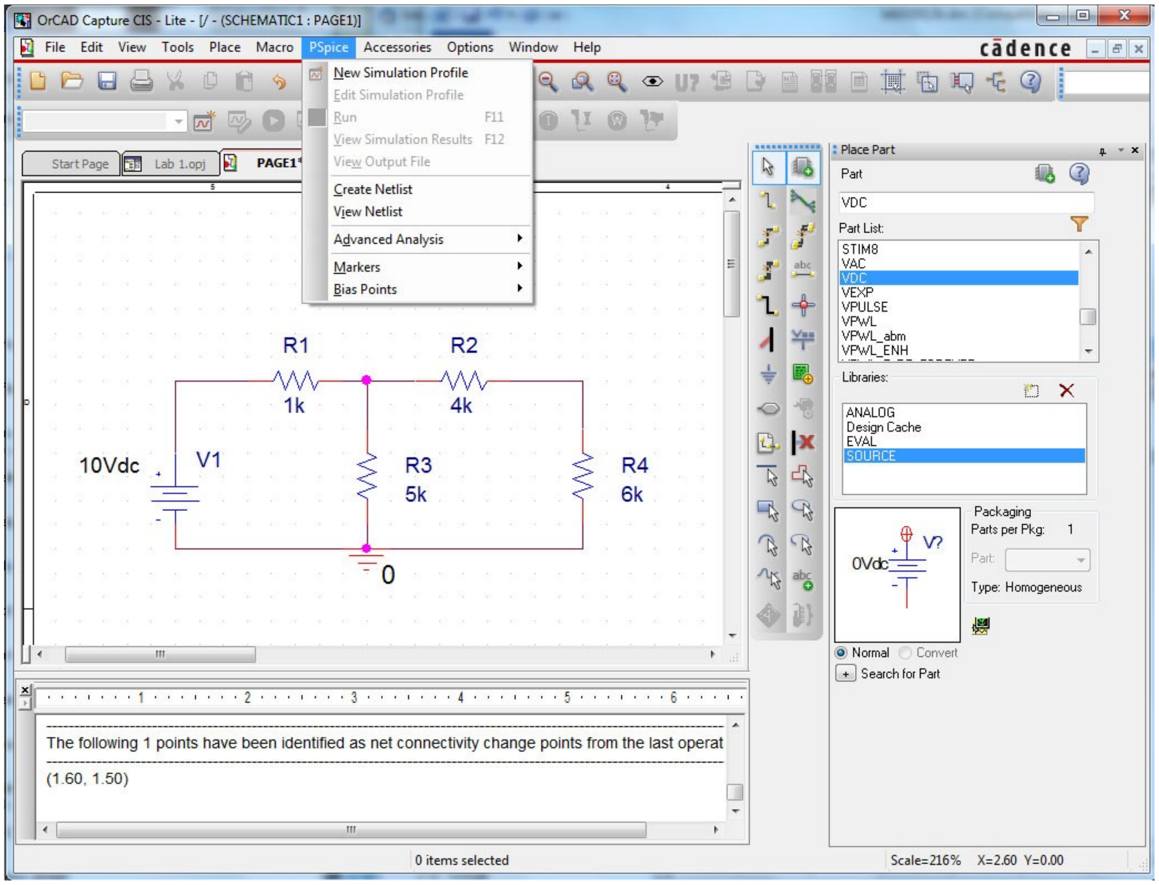


Figure 4: Illustration of how to choose place parts function.

You can double click on the value of the resistor to change the resistance, or double click on the name (R1) to change the name of the resistor. Continue until you have built the circuit shown in Figure 5. You will need to edit the values of the resistors and the voltage source. Be sure to place a ground (ground defines the node in the circuit that is at zero Volts). The "place ground" button is located 5 spaces below the "place wire" button. There are several choices for "ground", use the one labeled "0" in the Source library.

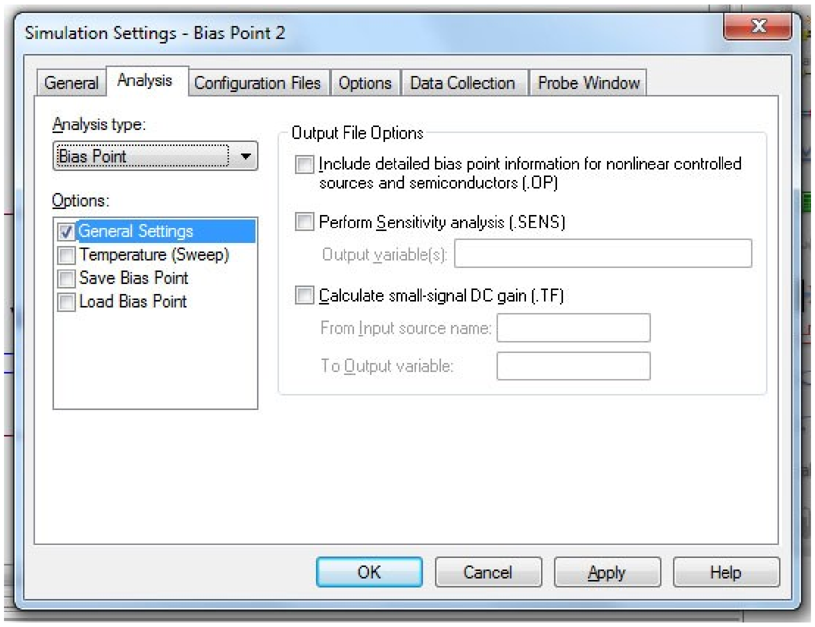
Next, you need to specify what analysis you want PSpice to do. The first step is to name the simulation. Choose ***PSpice>New Simulation Profile***, and then type in a name.



```
The following 1 points have been identified as net connectivity change points from the last operat
(1.60, 1.50)
```

Figure 5: Circuit to be modeled in PSpice in Part III.

Once you hit create after typing in the name you will get a Simulation Settings window (it may be behind another window, so look for it!). Choose Bias point, and the default settings are fine for our purposes. Click OK.


Simulation Settings - Bias Point 2

General | Analysis | Configuration Files | Options | Data Collection | Probe Window

Analysis type: Bias Point

Options:

[✓] General Settings
[ ] Temperature (Sweep)
[ ] Save Bias Point
[ ] Load Bias Point

Output File Options

[ ] Include detailed bias point information for nonlinear controlled sources and semiconductors (.OP)

[ ] Perform Sensitivity analysis (.SENS)

Output variable(s):

[ ] Calculate small-signal DC gain (.TF)

From Input source name:

To Output variable:

OK | Cancel | Apply | Help

Figure 6: Illustration of Simulations Settings window.

Now you are ready to run the simulation. Click ***PSpice>Run***. A new display window appears that in the future will be useful for plotting results. For this simple simulation, error messages may appear. At the same time node voltages will appear on the circuit diagram. These are the voltage of each node relative to the ground node (which is assumed to be at zero volts (hence the 0 notation). You can turn these voltages on and off using the V button above the simulation. When the simulation runs PSpice creates an output file (.OUT), which contains simulation information including error messages. PSpice also creates a data file (.DAT). However, we will generally use the graphical interface to look at the results.

Verifying Kirchoff's laws:

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

a. First display the currents by clicking on V to remove the voltage display and clicking on I to turn on the current display. (Or you can display both.)

b. 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. **Q1:** How does PSpice indicate the direction of the current? Hint: Change the voltage source to -10 V and see what the difference is.

c. **Q2:** 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.

d. Display the voltages by removing the current display and enabling the voltage display. The voltage across each component is the difference between the node voltages on the two sides of the component.

e. Add up the voltages around each of the 3 loops in the circuit – **Q3:** 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.) **Q4:** How do you keep track of the polarities (the signs) of the voltages?

PART 2. 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 7 below. Verify the signal on the oscilloscope using a BNC to BNC connector.

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

As above, review the document "Lab_Components_Basics" on Blackboard before proceeding. Full equipment manuals are also posted online.

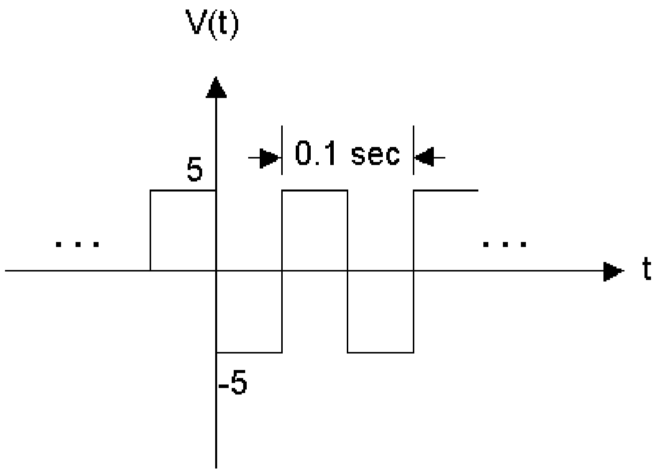


Figure 7. Diagram of desired output from the signal generator for Part 3.

3.2 Build the circuit shown in Figure 8 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.

3.3 Change the square-wave frequency gradually from 100Hz to 0.5Hz. **Q5:** 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 **Q6 (all 3 parts):** 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, or for different light intensities, or if you change the waveform to square or triangle? What is your guess as to what causes this limit? **Q7:** Is the frequency at which you can see that they are blinking "out of phase" (one off while the other is on and vice-versa) different from the frequency at which you can tell that they are blinking at all?

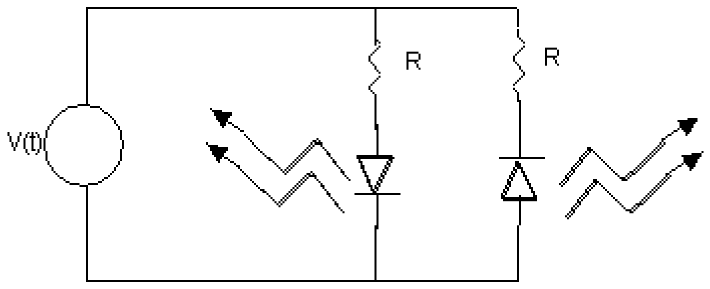


Figure 8. Blinking LED circuit for Part 3.

Part 4 – Instructions For the Lab Reflection (DiMarzio section only; others may have different expectations)…

Answer the numbered questions, Qn in your notebook.

Prepare a short "reflection" in which you discuss the following topics:

1. There are standard frame rates associated with the three main television standards, NTSC, PAL, SECAM. What do you think of these in light of what you have learned?

2. When you can no longer see the LED blinking, are you sure it is the response of your eye and not that of the LED? What experiment could you do to answer the question for sure?

3. What would be the differences if we substituted Tungsten-filament lamps for the LEDs?

Make sure the instructor or TA signs the book before you leave.

IMPORTANT: BEFORE YOU LEAVE THE LAB:

(a) **Place all of the components that your removed from the red tool box back in that box and return it to the cabinet that houses them**

(b) **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.**

(c) **Turn off all of the equipment you have used on your workbench.**

(d) **Make sure you return your protoboard, the equipment wires and your reusable container to the front window.**

(e) **Make sure to have your notebook signed by an instructor or TA before you leave the lab.**

Department of Electrical Engineering, Northeastern University.
Last updated: Minor edits by D Brooks 9/17 , 9/13/16 from version from 9/12/16 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