ECE 220 Laboratory 4
Volt Meter, Comparators, and Timer

Michael W. Marcellin

Please follow all rules, procedures and report requirements as described at the beginning of the document entitled ECE 220 Laboratory 1.
Always wear your safety glasses when performing your lab experiments.

Prelab Part 1

In Lab 2, we built a simple volt meter using a d'Arsonval panel meter. We found that the main problem with this simple meter is that its Thevenin resistance is rather small. Because of this, it tends to load the circuit being measured, changing the value of voltage to be measured. Here, we consider an improved volt meter that uses a $1\ mA$ d'Arsonval panel meter and an op amp as shown in Figure 1. In this figure, assume power rails of $+7$ V, and -2 V.

a) Calculate the value of R required so that $i_d = 1\ mA$ when $v_i = 5$ V.
b) For the value of R calculated above, write an expression for i_d as a function of v_i.

Observe that your equation in b) results in a linearly increasing needle deflection as a function of v_i, with full (100%) deflection at 5 V. Note also, that neither the value that you calculated for R, nor your equation for i_d depend on the internal resistance of the panel meter.
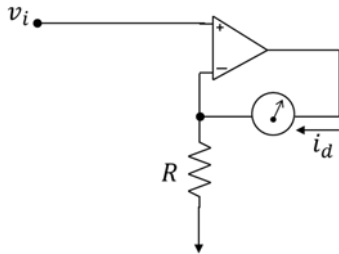

Figure 1. Improved volt meter.

In Lab 2, we used your simple meter to measure the voltage at terminals a and b in Figure 2 below. We discovered that your meter performed poorly when R_{Th} was not small. Specifically, we saw a large %error for the case of $R_{Th} = 4.7\ k\Omega$. This is because the input (Thevenin) resistance of your meter was not very large compared to $4.7\ k\Omega$.

Recall that an ideal volt meter has infinite input resistance. The input resistance looking into the input of the improved volt meter of Figure 1 is very large. In class, we would model it as infinity. In reality, it is "only" many Meg-Ohms. This results in the %error of the improved meter being negligible for values of R_{Th} up to at least $1\ M\Omega$. We will now verify this via a PSpice simulation.
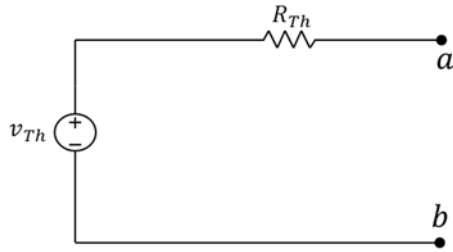
Figure 2. Thevenin equivalent circuit.

Start a new project in PSpice and draw the circuit shown below in Figure 3. This figure represents the use of the improved meter of Figure 1 attempting to measure v_{ab} in Figure 2. The 155 Ω resistor is used to model the panel meter although this resistance value is unimportant as noted above.

We have not used the V_{cc} labeling trick this time, and have just allowed wires to cross. Note carefully that PSpice includes a red dot (node) where wires connect. There is no such connection where the negative power supply wire crosses the feedback path. Note also, that this time we have included two parameters (variables). Each of them is created using the new property button in the property editor spreadsheet for the part PARAM, as before. You do not need two copies of PARAM. You will just create two different properties (columns) within the same instance of PARAM. Finally, note that the (default) values of the two parameters should be chosen as $v_{Th} = 5$ V and $R_{Th} = 1000 \ k\Omega = 1 \ M\Omega$. The property editor spreadsheet does not seem to allow $1 \ M\Omega$ to be entered directly, so we trick it by entering $1000 \ k\Omega$.

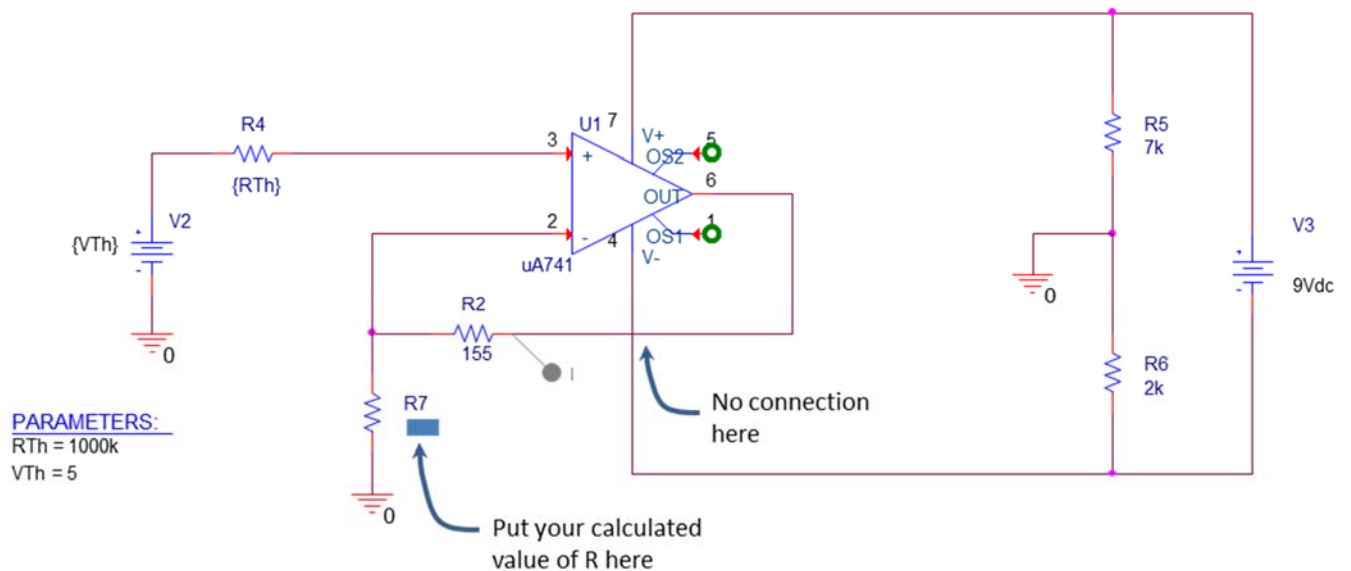Figure 3. PSpice schematic.

Perform a linear sweep of R_{Th} from $1 \ k\Omega$ to $1 \ M\Omega$ in increments of $1 \ k\Omega$. Note that v_{Th} will (automatically) stay at its default value of 5 V throughout this sweep. Use a current marker, as

shown in Figure 3, to measure the current through the panel meter. The resulting graph will show this current, i_d, as a function of R_{Th}. Ideally, it should be constant at $1\ mA$, corresponding to 100% needle deflection, since the voltage the meter is trying to measure is 5 V. In reality, i_d will decrease very slightly as R_{Th} increases due to the loading effect.

c) What value of R_{Th} (among those tested) yields the maximum error between i_d and its ideal value of $1\ mA$?
d) For this value of R_{Th}, calculate the %error in i_d.
e) print your graph.

Now edit the simulation profile and redo the sweep. This time we will sweep v_{Th} from 0 to 5 V in increments of 0.1 V. The value of R_{Th} will (automatically) stay at its default value of $1\ M\Omega$ throughout the sweep. You may have to add the current marker again after you edit the simulation profile. Ideally, the graph of the current through the panel meter, i_d, should rise linearly from 0 to $1\ mA$, corresponding to a linear needle deflection from 0 to 100%. Your graph should (essentially) show this desired behavior.

f) print your graph.

Prelab Part 2

In class, we discussed an op amp comparator circuit as shown below in Figure 4. For this circuit, whenever $v_i > v_{ref}$, v_o goes to the positive power rail. Else, v_o goes to the negative power rail. Throughout this part, assume that the power rails are ± 5 V.
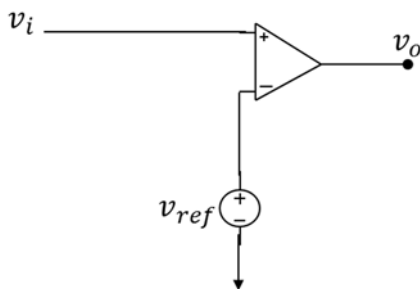

Figure 4. Comparator circuit.

Now consider the circuit of Figure 5.

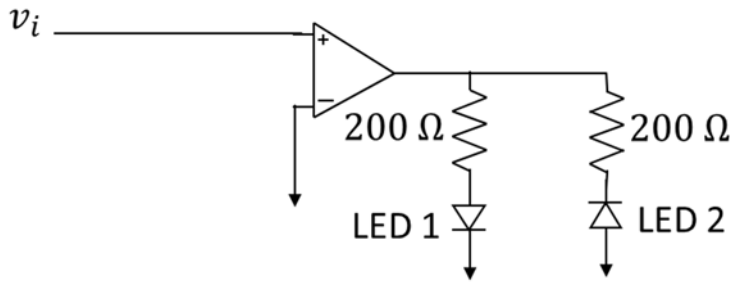a) Which LED will be turned on when $v_i = 1$ V? Which LED will be turned on when $v_i = -1$ V?

Figure 5. Comparator with LEDs.

Now consider the circuit of Figure 6.

b) Write an equation for v_1 as a function of the pot setting F. Assume $F = 0$ corresponds to the slider being all the way up.
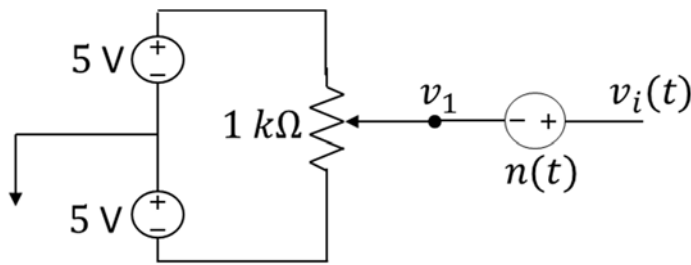

Figure 6. Input signal.

To make things more interesting, we add some "noise" to v_1. The resulting "noisy" signal is $v_i(t) = v_1 + n(t)$. Assume that the noise $n(t)$ is a square wave of amplitude $v_p = 0.1$ V ($v_{pp} = 0.2$ V) and frequency $f = 2$ Hz. Figure 7 shows a graph of $v_i(t)$ for a pot setting $F = 0.4$. Make sure that you understand this figure. Changing the pot setting F will raise or lower the level of the signal in Figure 7. As another example, when $F = 0.5$, the square wave will be oscillating above and below 0 V, rather than above and below 1 V as shown in Figure 7.
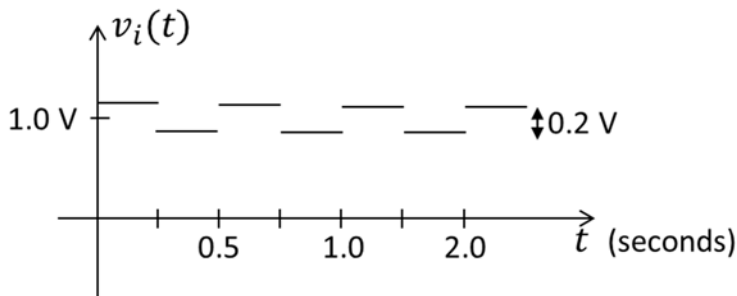

Figure 7. "Noisy" signal.

Consider now connecting $v_i(t)$ in Figure 6 as the input to the circuit of Figure 5. Like the improved volt meter, the input resistance of the comparator is very high. Thus, we do not need to worry about the loading effect.

c) What do you expect the LEDs to do for a pot setting of $F = 0.4$? How about $F = 0.5$? How about $F = 0.6$?

In class, we also talked about the Schmitt trigger (comparator with hysteresis) which reduces "chattering" due to noise. A Schmitt trigger circuit is shown below in Figure 8. Again, assume power rails of ± 5 V.

d) What is the threshold T for the Schmitt trigger as shown?

Note that this threshold is larger than v_{pp} of the noise in the example above. Thus, any chattering in the example should be eliminated by replacing the comparator with the Schmitt trigger. We will verify this in the lab.
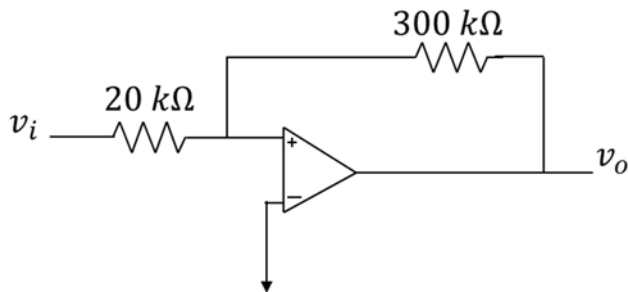

Figure 8. Schmitt trigger.

Prelab Part 3

In this part, we use an RC circuit together with a comparator to build a simple timer. Figure 9 below shows an RC circuit with a step input of 5 volts at time $t = 0$. Assume that the initial capacitor voltage is 0 V.

In Chapter 7, we will learn the theory for RC circuits. For now, just use the fact that $v_c(t) = 5 - 5e^{-t/RC}$ V.
a) Let $C = 100\ \mu F$ and calculate the value of R required so that the capacitor voltage passes through 2.5 V at time $t = 9$ seconds. That is, $v_C(9) = 2.5$ V.
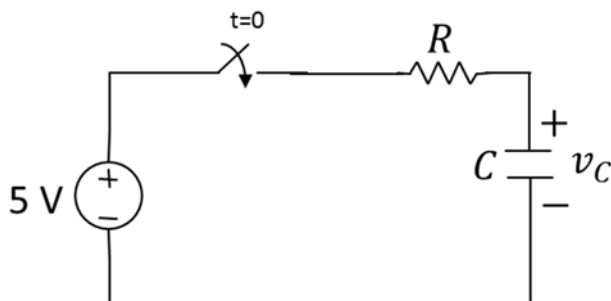

Figure 9. RC step response.

The circuit of Figure 10 below shows the RC circuit from Figure 9 connected so that the

capacitor voltage appears as the input to a comparator. Note that the input current i_p to the Op Amp is very small so that the loading effect on the RC circuit will be negligible. Note also that $v_{ref} = 2.5$ V for the comparator. As usual, 3 diodes are used to model an LED. It should be clear that prior to time $t = 9$, the LED will be off. After time $t = 9$ the LED will be on. Thus, this circuit acts as a simple timer. The time at which the LED turns on can be adjusted by choosing different values for the resistor R.
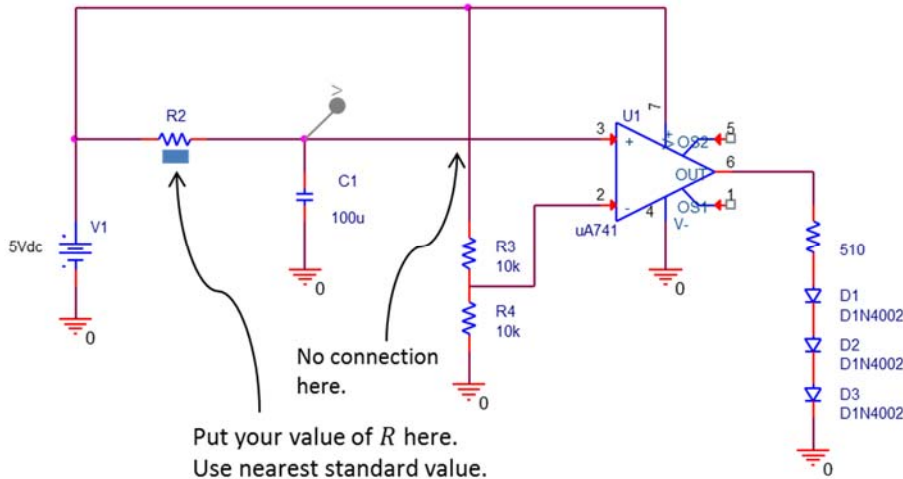

Figure 10. Timer circuit.

We will now do a PSpice simulation of our timer. Start a new project in PSpice and draw the circuit in Figure 10. The part name for a capacitor is C. You will need to set the initial voltage of the capacitor. Double click carefully on the capacitor symbol itself (not the value or the name). The property editor spreadsheet will open. Look for the column labeled IC. This is the initial condition. For a capacitor, this is always voltage. For an inductor, of course, IC would be current. Set the value below IC to 0. Click on the tab labeled Page 1 to return to your schematic.

Click PSpice -> New Simulation Profile. A new window will open. Enter a name and click create. Another new window will open (check the icons at the bottom of the screen). Set the following parameters:
Analysis type: Time Domain (Transient)
Run to Time: 80 seconds
Start saving data after: 0 seconds
Maximum step size: 0.01 seconds
Click OK
Put a voltage marker at the top of the capacitor, and run the simulation. The resulting graph will be the capacitor voltage as a function of time. Make sure that this behaves as expected. When does it cross through 2.5 V? What is the capacitor voltage when $t = 5RC$?

b) Print your graph.

Delete the voltage marker, and add a current marker to monitor the LED current. Run the

simulation again. This time the graph shows the LED current as a function of time, indicating that the LED is off prior to 9 seconds, and on after 9 seconds, as expected.

c) Print your graph.

Prelab Grading

Part 1
a)   2 pts
b)   2 pts
c)   1 pt
d)   1 pt
e)   3 pts
f)   1 pt

Part 2
a)   1 pt
b)   2 pts
c)   3 pts
d)   1 pt

Part 3
a)   2 pts
b)   3 pts
c)   3 pt

_____
25 total pts

Lab 4 Data Sheet

Please follow all rules, procedures and report requirements as described at the beginning of the Lab 1 Data Sheet.

Lab Grading: Data Sheet = 15 pts; Explanation = 10 pts.

Lab Part 1

No experiment

Lab Part 2

Comparator circuit working: Y / N (circle one)
5 pts

Schmitt trigger circuit working: Y / N (circle one)
5 pts

Lab Part 3

Timer circuit working: Y / N (circle one)
5 pts

Lab Experiments

Lab Part 1

No experiment.

Lab Part 2

Online students: due to a limitation in the way that the grounds are connected in the Analog Discovery 2, you will follow a slightly different procedure for Part 2. Please see the appendix for the procedure to follow.

On-campus students:

Build the circuit shown in Figure 11. Use an LM358N op amp. Use power rails for the op amp of ± 5 V. Use the function generator for the square wave "noise" $n(t)$. Set it for $v_p = 0.1$ V ($v_{pp} = 0.2$ V) and a frequency of $f = 2$ Hz. Monitor the voltage v_i with the scope as you vary the pot setting. Make sure you have the scope coupling in DC mode. Verify that v_i behaves as expected (see Figure 7 and the related discussion in the prelab). Verify that the LEDs behave as you expect for different pot settings. Specifically, you should be able to find pot settings where only LED1 (or LED2) is on. You should also be able to find a setting where the LEDs blink on and off (alternating). This is "chatter."
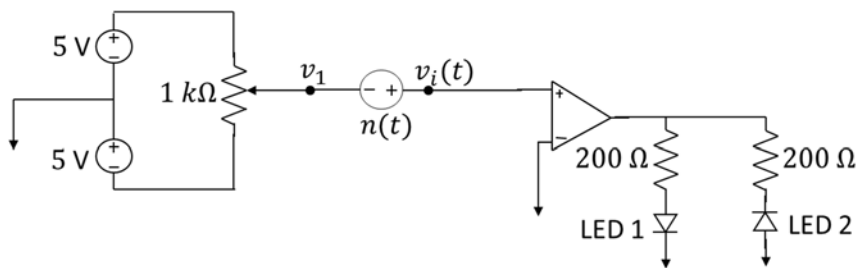Have your TA initial that your circuit is working.

Figure 11. Comparator with noise.

Modify your circuit to that shown below in Figure 12, which uses a Schmitt trigger in place of the comparator. Repeat your inspection of the LEDs for different pot settings. Raise and lower the pot setting several times to ensure that all chatter has been eliminated.
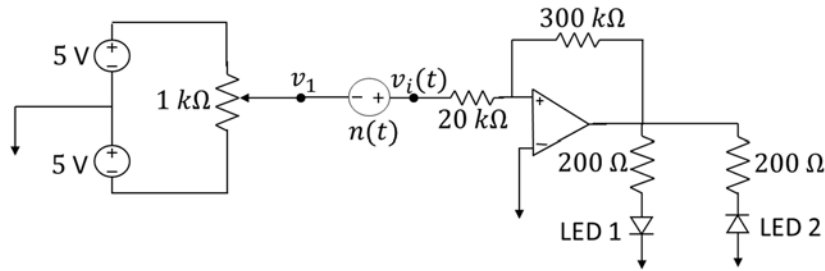Have your TA initial that your circuit is working.

Figure 12. Schmitt trigger with noise.

Lab Part 3

Build the circuit shown in Figure 13. Use the value of R that you calculated in the prelab. Use a piece of wire for the switch. The initial position of the switch ensures that the capacitor is discharged so that $v_C(0) = 0$ V. Note the slightly different symbol shown for the capacitor in this figure. This indicates a "polarized" capacitor that should only be charged in one direction. The straight line indicates the "+" terminal, while the curved line indicates the "−" terminal. Charging a polarized capacitor in the wrong direction can destroy it. The capacitor that we will use for this experiment is polarized. The positive terminal (the "top" terminal in Figure 13) is the one with the longer of the two wires. Be very careful when you hook up your circuit, and **as always, wear your safety glasses.**

At some time of your choosing ($t = 0$), move the switch to the other position. 9 seconds later, the LED should turn on.
Have your TA initial that your circuit is working.
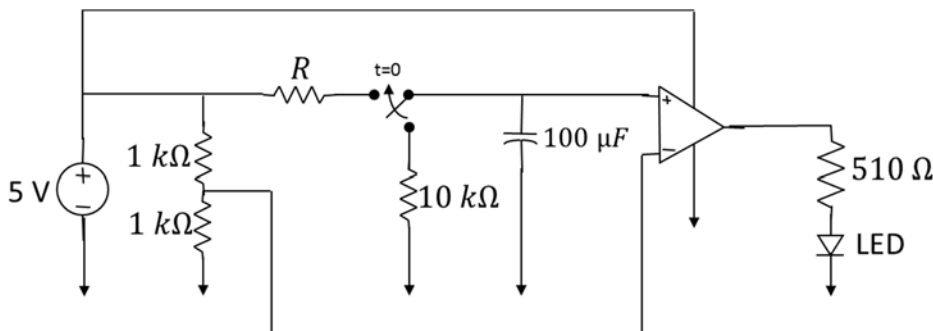*On-campus students: make a video to demonstrate that your circuit is working.*

Figure 13. Timer circuit.

APPENDIX: Lab Part 2 for Online Students

Build the circuit shown in Figure 14. Use an LM358N op amp. Use ± 5 V for the power rails of the op amp. In Figure 14, we are using the function generator in place of the circuit from Figure 6 of the pre-lab to generate the "noisy" signal $v_i(t)$. We will employ a DC offset in the function generator to create the "noiseless" signal v_1. Set the function generator to output a square wave with $v_p = 0.1$ V ($v_{pp} = 0.2$ V) and a frequency of $f = 2$ Hz.

Set the DC offset of the function generator (v_1) to various values, while monitoring the noisy voltage v_i with the scope. Make sure you have the scope coupling in DC mode.

Verify that v_i behaves as expected (see Figure 7 and the related discussion in the prelab). Verify that the LEDs behave as you expect for different settings of v_1. Specifically, you should be able to find settings where only LED1 (or LED2) is on. You should also be able to find a setting where the LEDs blink on and off (alternating). This is "chatter."

On-campus students: make a video to demonstrate that your circuit is working.
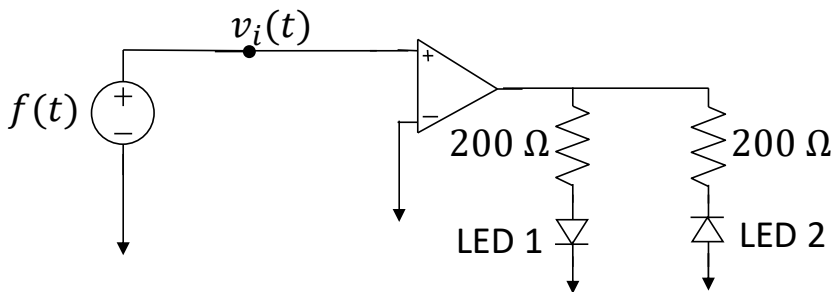

Figure 14. Comparator with noise.

Modify your circuit to that shown below in Figure 15, which uses a Schmitt trigger in place of the comparator. Repeat your inspection of the LEDs for different values of v_1. Raise and lower v_1 carefully to ensure that all chatter has been eliminated.

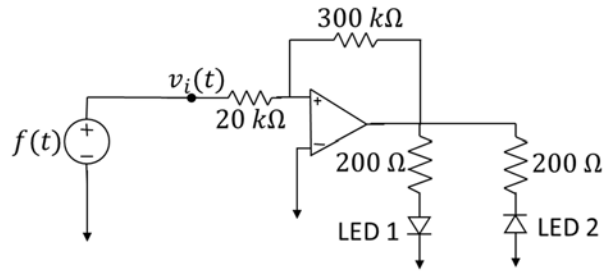*On-campus students: make a video to demonstrate that your circuit is working.*

Don't forget to go back and do Part 3.

Figure 15. Schmitt trigger with noise.