

ECE 220 Laboratory 1

Wheatstone Bridge, Voltage Divider, the Loading Effect, and Dimmer Circuit

Michael W. Marcellin

The first portion of this document describes preparatory work to be completed in advance of the laboratory session. This portion of the work is referred to as the "Prelab." The second portion of this document contains a description of the experiments to be performed during the laboratory session. That portion of the document is referred to as the "Lab."

When you do the lab:

- 1) **Always wear your safety glasses.**
- 2) Bring a printed copy of the lab data sheet (appears later in this document).
- 3) On-campus students - bring your Cat Card and wear closed toed shoes.

Please see the course website (weekly assignments page) to determine the dates on which the Prelab Report and Lab Report are due. Read the submission instructions at the top of that page carefully. **Failure to follow submission instructions will result in a 25% penalty on your score.**

Prelab Report Requirements

For your Prelab Report, turn in all calculations/answers requested, together with all PSpice printouts requested in each part. Within each part, the items to be turned in are labeled with the letters a), b), c), etc.

This does not need to be a formal report. The calculations/answers can be hand written. The level of detail should be similar to homework. Please include your lab day and group (A or B) under your name on all Prelab reports.

Please circle your answers. Please give numeric answers using at least 4 significant digits. Do not leave your answers as fractions.

Prelab Part 1

A copy of the resistor color code can be found on the course webpage. Look under the link "Lab Resources." Resistors can only be purchased (inexpensively) for certain standard values. A list of standard resistor values, as well as other parts available in the lab, can also be found under "Lab Resources."

Give the color code for each of the following resistor values:

- a) $36\text{ k}\Omega \pm 5\%$
- b) $47\ \Omega \pm 10\%$
- c) $680\ \Omega \pm 10\%$
- d) $1\text{ M}\Omega \pm 5\%$

Give the resistance and tolerance for resistors that have the following color codes:

e) brown, black, red, gold

f) yellow, black, red, silver

g) green, blue, red, silver

h) green, blue, green, gold

Prelab Part 2

In class, we learned (or will learn) that the Wheatstone Bridge shown in Figure 1 below is balanced when $R_1/R_2 = R_3/R_4$. When the bridge is balanced, $v_5 = 0$ regardless of the value of v_s or R_5 .

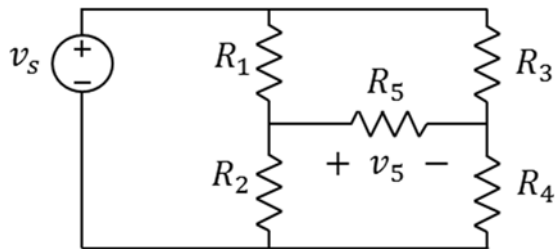


Figure 1. Wheatstone bridge.

Let $R_1 = 27\text{ k}\Omega$, $R_2 = 3\text{ k}\Omega$, and $R_3 = 5.6\text{ k}\Omega$.

a) Calculate the value of R_4 required to balance the bridge.

Prelab Part 3

Consider Figure 2 as shown below. From class, we know that the $1\text{ k}\Omega$ and $3\text{ k}\Omega$ resistors form a voltage divider and that $v_2 = v_1 = \frac{1\text{ k}}{1\text{ k} + 3\text{ k}} 4 = 1\text{ V}$.

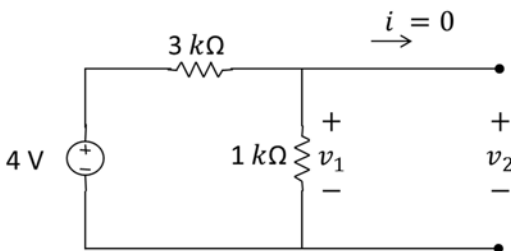


Figure 2. A voltage divider.

Consider now Figure 3 below. The circuit in Figure 3 is the same circuit from Figure 2, but with two $560\ \Omega$ resistors added on the right. Note that the $1\ k\Omega$ and $3\ k\Omega$ resistors no longer form a voltage divider due to the presence of the $560\ \Omega$ resistors. In particular, the $1\ k\Omega$ and the $3\ k\Omega$ resistors are no longer in series. That is, they do not have the same current flowing through them. The $560\ \Omega$ resistors “load” the original circuit which changes (reduces) the value of v_1 to some new value v'_1 . The reduction in v_1 is called “the loading effect.” It is worth noting that the two $560\ \Omega$ resistors *do* form a voltage divider, so that $v'_2 = \frac{560}{560+560} v'_1 = \frac{1}{2} v'_1$.

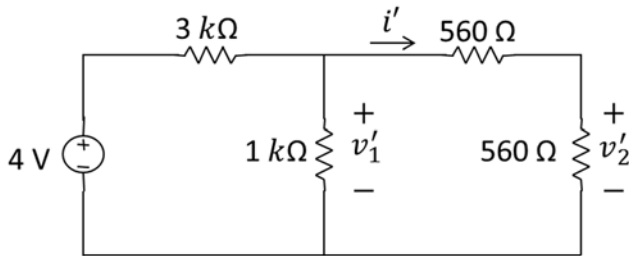


Figure 3. Illustration of the loading effect.

a) Calculate v'_1 , v'_2 , and i' .

Now consider Figure 4 below, which is the same as Figure 3, but with the $560\ \Omega$ resistors replaced by $560\ k\Omega$ resistors.

b) Calculate v''_1 , v''_2 , and i'' .

In this case, we see that $v''_1 \approx v_1$. The loading effect is negligible in this case due to the fact that $i'' \approx 0$, and so nearly the same current flows through the $1\ k\Omega$ and $3\ k\Omega$ resistors.

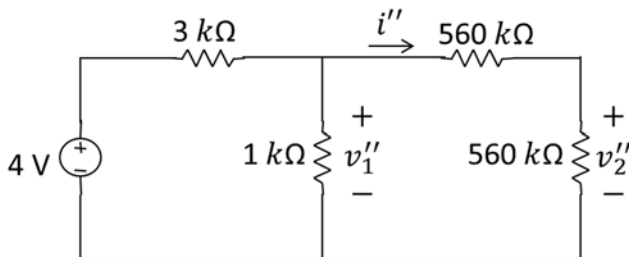


Figure 4. Negligible load current.

Prelab Part 4

In this part, we will use a light emitting diode (LED). In general, diodes are non-linear devices. The circuit symbol for a diode is as follows:

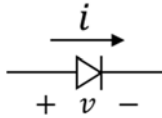


Figure 5. Circuit symbol for a diode.

A diode can be thought of as a “one-way valve.” That is, current can only flow in the forward direction, as shown in the figure above. Current only flows when voltage is positive according to the polarity in the figure above. No current flows if a negative voltage is applied. The terminal of the diode where current flows in (the left side in the figure above) is called the anode. The terminal where the current flows out (the right side in the figure) is called the cathode.

Recall from class that resistors are linear, and are governed by Ohm’s Law which is given by $v = iR$, as shown in the following figure.

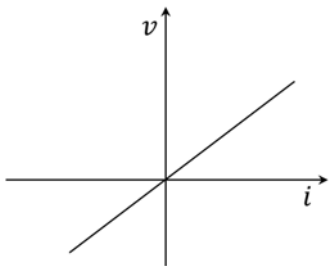


Figure 6. Resistors are linear.

In contrast, over the range of currents of interest here, a diode has a non-linear voltage-versus-current characteristic as sketched in Figure 7 below.

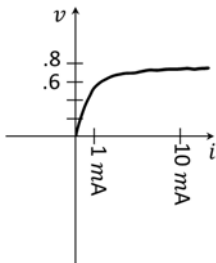


Figure 7. Voltage-versus-current for a D1N4002 diode. This is a standard diode (not an LED).

You will learn more about diodes in future courses. In ECE 220 we make the following simplifying assumption:

If current is flowing in the forward direction (as shown on the circuit symbol in Figure 5), the “forward voltage” is fixed at $v = 0.7$ V. The voltage-versus-current characteristic corresponding to the simplifying assumption is shown in Figure 8 below. Note the similarities and differences between Figures 7 and 8.

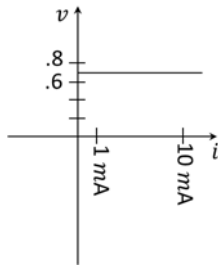


Figure 8. Simplified voltage-versus-current for D1N4002.

For our lab experiments, we will use an LH3330 LED, which has a similar voltage-versus-current characteristic but with a forward voltage that we will approximate by $v = 2.1$ V (rather than 0.7 V).

Using what we have learned above, we can analyze the LED circuit given in Figure 9 below.

Assuming that $v_{LED} = 2.1$ V, we can use KVL to find that $v_R = 2.9$ V. Thus, $i_{LED} = i_R = \frac{2.9}{200} \approx 15$ mA, which is a safe value for an LH3330 LED. By contrast, if we change the 5 V supply to -5 V, $i_{LED} = 0$ (one-way valve). So, $v_R = 200i_R = 200i_{LED} = 0$. Then, by KVL, we can find that $v_{LED} = -5$ V.

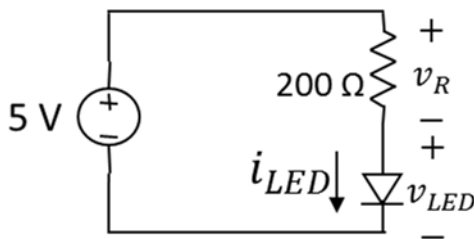


Figure 9. A simple LED circuit.

From the discussion above, it should be clear that the resistor in the circuit of Figure 9 is used to set the value of the current that flows through the LED. Don't forget this. It is a favorite question used by potential employers during job interviews!

Simple Dimmer Circuit

We will now explore circuits to control the intensity of light emitted by an LED. This intensity varies as a function of i_{LED} .

We begin with a simple design using a potentiometer (pot). A pot is a 3-terminal device. It is a resistor with a variable tap. The circuit symbol for a $10\text{ k}\Omega$ pot is shown below.

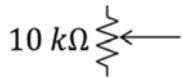


Figure 10. Circuit symbol for a $10\text{ k}\Omega$ potentiometer (pot).

The $10\text{ k}\Omega$ pot above is equivalent to the following:

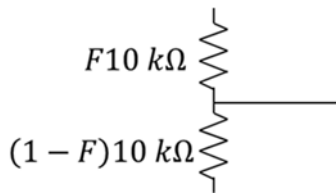


Figure 11. Equivalent pot circuit.

The value F can be varied between 0 and 1 by turning a knob or by moving a slider, depending on the design of the pot. In the figure above, $F = 0$ corresponds to the slider being all the way up. Similarly, $F = 1$ corresponds to the slider being all the way down. Of course, $F = 0.5$ is half-way, etc.

The circuit shown in Figure 12 below is then a voltage divider with output voltage $v_1 = \frac{(1-F)10k}{(1-F)10k+F10k} 5 = (1-F)5\text{ V}$. So, by turning the knob on the pot, the output voltage v_1 can be adjusted between 0 and 5 V.

a) Repeat the calculation for a $1\text{ k}\Omega$ pot to show that the same expression is obtained for v_1 . That is, $v_1 = (1-F)5\text{ V}$.

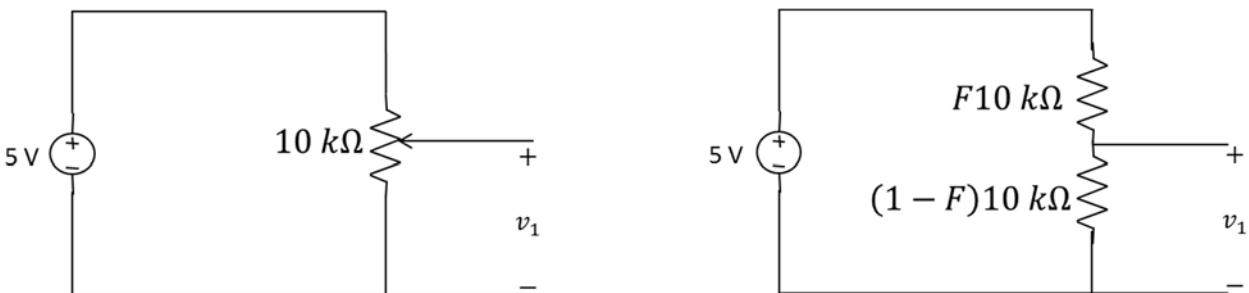


Figure 12. Voltage divider using a pot. The two circuits in this figure are equivalent.

Connecting our $200\ \Omega$ resistor and LED to a voltage divider (formed using a $1\ k\Omega$ pot, rather than a $10\ k\Omega$ pot) results in the simple dimmer circuit shown in Figure 13 below. As F is varied from 1 to 0, v_1 varies from 0 V to 5 V. Ideally, v_1 (and thus i_{LED}) would vary linearly with the movement of the slider on the pot. That is, the value of v_1 would remain as given above ($v_1 = (1 - F)5$). Specifically, when the slider is at the bottom, v_1 would be 0. Moving the slider one quarter of the way up would yield 1.25 V. Moving the slider one half of the way up would yield 2.5 V, etc.

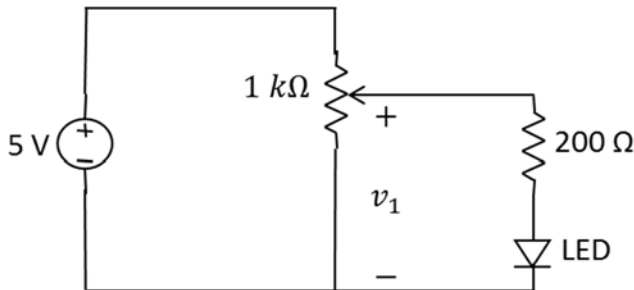


Figure 13. Simple dimmer circuit.

Unfortunately, the $200\ \Omega$ resistor and the LED look like a load to the voltage divider (as formed by the pot). As a result, we end up with v_1 (and thus i_{LED}) varying in a non-linear way as a function of the pot setting. Indeed, the LED current as a function of the pot setting F is sketched roughly in Figure 14 below. As expected, the graph is not a straight line (non-linear) due mostly to the loading effect. We will verify the shape of the graph in Figure 14 via a PSpice simulation as described below.

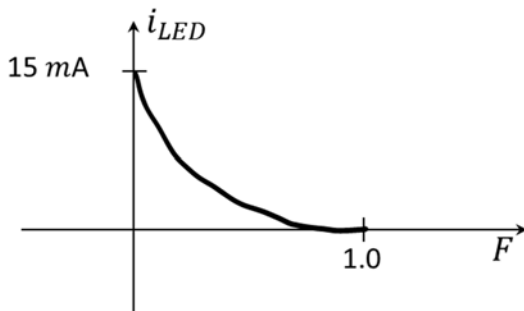


Figure 14. Non-linear LED current as a function of pot setting.

PSpice Simulation of Simple Dimmer Circuit

To begin, start PSpice (actually, Capture) as described in the PSpice homework.

If necessary, review in the PSpice homework how to start a new project and how to place and wire parts. Useful parts for this exercise include:

VDC (voltage source) – ignore /Design Cache, /ANALOG, /BREAKOUT, etc. These are library names.

R (resistor)

POT (pot)

D1N4002 (diode) – this is a standard diode, not an LED

2N2222 (transistor)

PARAM (simulation parameters)

Remember: In PSpice, every circuit must have a ground, and that only grounds named 0 will work.

Draw the circuit shown on the left side of Figure 15 below. Be sure to draw it on the left side of your screen and save room to draw the figure on the right side at a later time. For now, ignore the current marker (it looks a bit like a dart). To change the value of the voltage source, double click on the value. Be careful to click only on the item you wish to change. The voltage source has a name (V1) and voltage value (0Vdc). Double click on 0Vdc. A new window will open where you can change the value to 5Vdc, as required in the circuit. You can change the value of the resistor in the same way. Check that the pot value is 1K. If not, change it in the same way. By default, the pot set point is 0.5. We will change this to a variable later.

Note: Compare the circuit on the left of Figure 15 with the circuit of Figure 13. The student version of PSpice does not have an LH3330 LED (forward voltage 2.1 V). So, here we use three D1N4002 diodes in series (each with forward voltage 0.7 V) to emulate one LH3330.

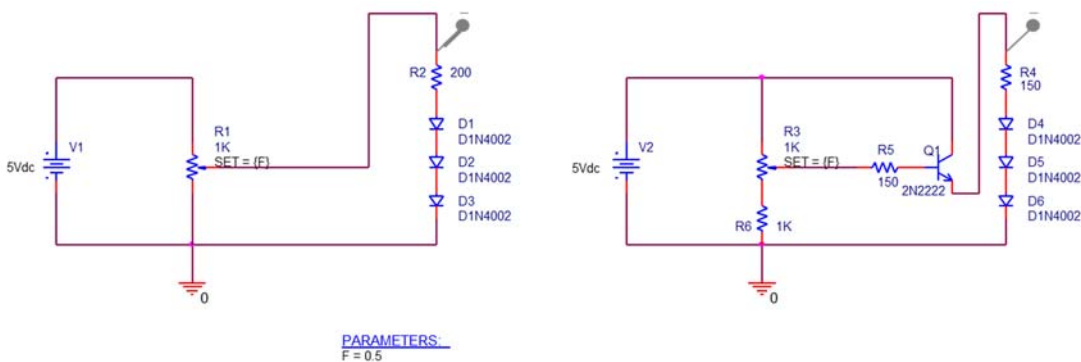


Figure 15. PSpice dimmer circuits.

Our goal in this simulation is to plot a graph of the LED current as a function of the pot setting. To this end, we will create a variable and name it F, consistent with our previous discussions. If you have not already done so, place the “Part” PARAM, which appears as PARAMETERS below the left circuit in Figure 15. Double click on PARAMETERS. This will open a Property Editor Spreadsheet. Click the “New Property” button in the upper left portion of the screen. A new screen will open. Enter F as the parameter name. Below that enter a Value of 0.5. This is the default value. We will create a “sweep” of values for F via the instructions below. Click “OK” Click on the F Heading (that we just created) in the Property Editor Spreadsheet. You may need

to scroll to the right or left to find this heading. Now click the button above labeled "Display..."
A new window will open. Click the button for Name and Value and click "OK"
Return to the schematic by clicking the tab marked "PAGE1"
Below the PARAMETERS part, you should now see F=0.5 as in Figure 15.

We now need to associate our parameter F with the pot. Double click carefully on the pot setting parameter (SET = 0.5). A new window will open. In this window, change the value 0.5 to {F} Don't forget the {} braces. Click ok. On the circuit symbol for the pot, you should now see SET={F} as in Figure 15.

We are almost there!

Here we will specify the parameters to "sweep" the value of F from 0 to 1 in steps of 0.01. Click PSpice -> New Simulation Profile

Enter a name for this profile (e.g., dimmer sweep)

Click Create

A new window will open. You might have to look at the icons at the bottom of the screen to find it.

In the drop down menu entitled Analysis type:, choose DC Sweep.

Enter the following parameters:

Sweep Variable: Global parameter

Parameter name: F (no curly braces this time)

Sweep type: Linear

Start value: 0

End value: 1

Increment: 0.01

Click "OK"

Last step!

We will now add a current marker. Click

PSpice -> Markers -> Current Into Pin

Rotate the marker once, and then place it at the top pin of the 200 Ω resistor as shown on the left side of Figure 15. If you have any wire extending above the resistor, make sure to put the probe right on the pin of the resistor. Look carefully at the circuit in Figure 15 for an example.

This marker measures the current **into** a pin. Since the current through the resistor is the same as the current through the diode(s), this marker will measure the current through the (emulated) LED. You could place the marker at the top of the first diode. The danger here is that you could accidentally place it at the bottom of the resistor. You would then get the current **into** the **bottom** of the resistor, which would be the negative of the current **into** the **top** of the resistor.

Here we go.

Click

PSpice -> Run

A new window will open. You may need to look at the icons on the bottom of the screen to find it. This window will contain a graph of the LED current versus F . If the graph does not look like Figure 14, something has gone wrong. Check your work carefully and try again. If the simulation fails to run, and you don't get a graph at all, again check your work. Common errors include: no ground, wrong ground (see previous discussion on grounds), wires/pins not connected properly (in this last case, PSpice usually will draw some circles on your schematic to highlight the problem).

b) Print the graph of LED current versus F (click File -> Print). You will turn this graph in as part of your prelab report.

Transistor Dimmer Circuit

We now explore a more sophisticated circuit that uses a transistor to buffer (isolate) the load from the voltage divider. The symbol element for a transistor is shown below in Figure 16. As shown there, a transistor is a 3-terminal device. The three terminals are called base, collector, and emitter. These terminals are labeled b , c , and e in the figure, respectively. Like a diode, a transistor is generally non-linear. You will learn more about transistors in future classes.

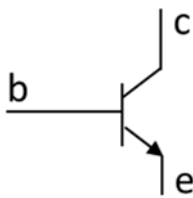


Figure 16. Transistor circuit symbol.

An LED dimmer circuit that uses a transistor to reduce the loading effect is shown below in Figure 17. To avoid clutter, analysis of this circuit is included in an appendix at the end of this document. **Make sure you read the appendix. There you will gain some appreciation for why we study dependent sources.** The basic idea is that i_b is very small (a few hundred micro-amps) and so the loading effect on the voltage divider is negligible. The transistor acts as a current amplifier to bring the LED current to the desired range. From the appendix, $i_{LED} \approx (1 - F)15 \text{ mA}$, which is sketched in Figure 18 below, revealing that the desired linear relationship is (approximately) achieved by this more sophisticated circuit.

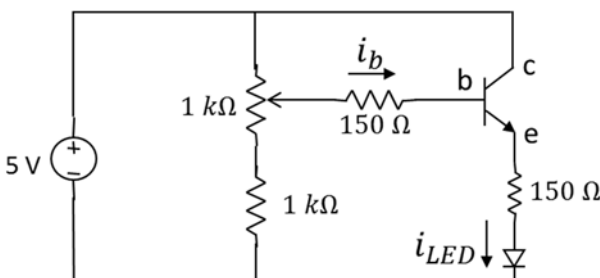


Figure 17. Transistor dimmer circuit.

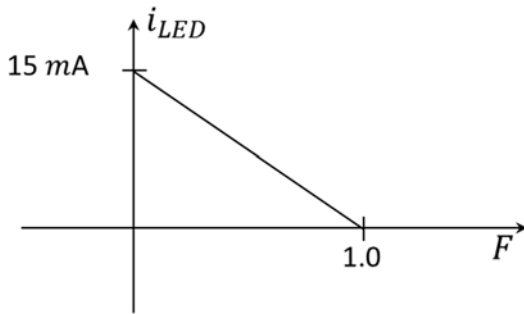


Figure 18. Approximate i_{LED} versus F for the circuit of Figure 17.

We will verify that the dimmer circuit of Figure 17 indeed operates roughly as shown in Figure 18 via the following PSpice simulation.

For convenience, Figure 15 is repeated here as Figure 19.

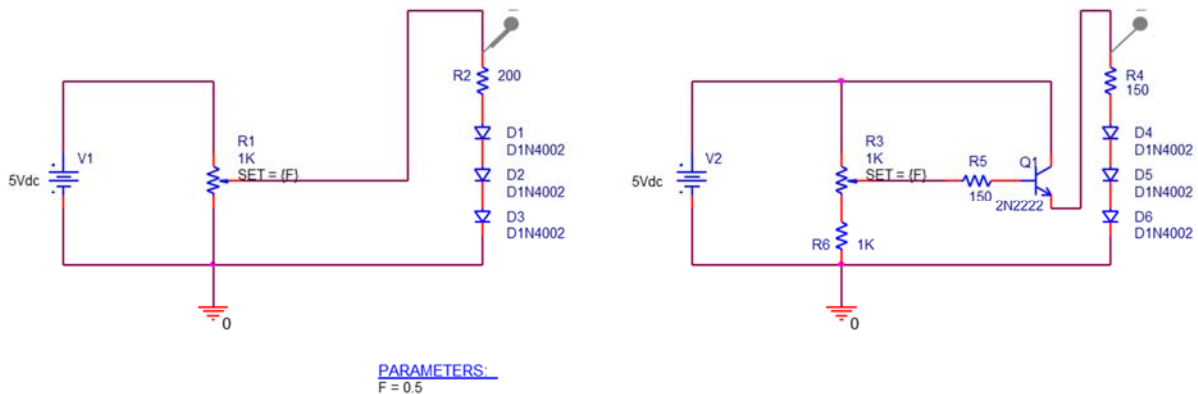


Figure 19. PSpice dimmer circuits.

PSpice Simulation of Transistor Dimmer Circuit

Add (draw) the circuit on the right of Figure 19 to your PSpice schematic. You do not need to do anything with the PARAMETERS, nor the simulation profile. You have already taken care of all of that. Hint: You can copy/paste the entire circuit from the left, and then edit it. If you choose to draw it from scratch, don't forget to associate the parameter F with the pot, following the instructions given above.

Add a current marker on the right circuit and remove the current marker on the left circuit (carefully click the marker and press delete on the keyboard). Run the simulation again. The graph of LED current versus F should now look roughly like Figure 18.

c) Print this graph of LED current versus F and turn it in with your prelab report.

Prelab Grading

Part 1

a) – h) 8 pts (1 pt each)

Part 2

a) 1 pt

Part 3

a) 3 pts (1 pt each item)

b) 3 pts (1 pt each item)

Part 4

a) 2 pts

b) 4 pts

c) 4 pts

25 total pts

Lab 1 Data Sheet

Print and bring this sheet to lab and record your experimental data below. You must get your TA to initial each part before you move on to the next part.

Online students: you do not need initials from a TA. Separate instructions for you will be given at the appropriate places in the lab instructions.

Turn in a brief typed report. Your report should explain your experimental results and any differences from anticipated results (from calculations and/or simulations in your prelab).

Please include your lab day and group (A or B) under your name on all Lab reports.

A scanned copy of your data sheet must be included as the last page of your report.

Online students: put any photos that you are asked to make into your report after the datasheet. Any videos that you are asked to make can be turned in as separate files.

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

Lab Part 1

No data

Lab Part 2

$R_1 =$	$R_2 =$	$R_3 =$
$R_4 =$	$R_1/R_2 =$	$R_3/R_4 =$

1 pt each

Lab Part 3

$v_1 =$		
$v_1' =$	$v_2' =$	$i' =$
$v_1'' =$	$v_2'' =$	$i'' =$

1 pt each

Lab Part 4

Dimmer circuit works correctly: Y / N (circle one)

½ pt

$F = 1$	$i_{LED} =$
$F = 0$	$i_{LED} =$
$F = 0.5$	$i_{LED} =$

½ pt each

Lab Experiments

Lab Part 1

No experiments for this part.

Lab Part 2

Build the circuit shown in Figure 20 below. Use the same resistor values for R_1 , R_2 , and R_3 as given in Prelab Part 2. Use $R_5 = 10\text{ k}\Omega$ and $v_s = 5\text{ V}$. Use a $1\text{ k}\Omega$ pot as a variable resistor for R_4 as shown in the figure. Use the digital multi-meter (DMM) to measure v_5 . Adjust the pot until $v_5 = 0$.

Online students: take a photo of your circuit and DMM measuring 0 V.

Take the circuit apart being careful not to disturb the pot setting. Use the DMM to measure the actual values of R_1 , R_2 , R_3 , and R_4 . Calculate R_1/R_2 and R_3/R_4 . Record these six values on the Lab 1 Data Sheet and have your TA initial your results.

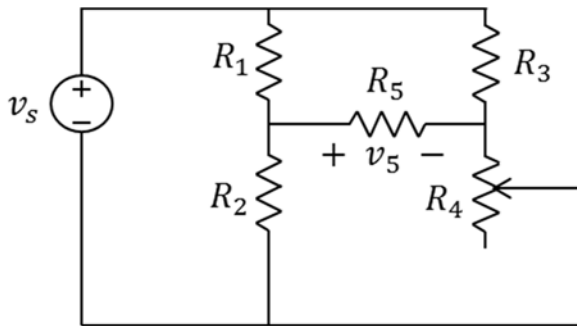


Figure 20. Balancing a Wheatstone bridge.

Lab Part 3

Build the circuit shown in Figure 2 of the Prelab. Use the DMM to measure v_1 .

Build the circuit shown in Figure 3 of the Prelab. Use the DMM to measure v_1' , v_2' , and i' .

Build the circuit shown in Figure 4 of the Prelab. Use the DMM to measure v_1'' , v_2'' , and i'' .

Record these 7 values on your Lab 1 Data Sheet and have your TA initial your results.

Online students: take a photo of each circuit with your DMM showing the values recorded on your data sheet.

Lab Part 4

Build the circuit shown in Figure 17 of the Prelab. Use a 2N2222 transistor (it is ok to substitute 2N2222A), and an LH3333 LED. The 2N2222 transistor is packaged in a small metal can with 3

pins. The pin diagram is shown in Figure 21 below. The longer of the two wires on the LED is the anode.

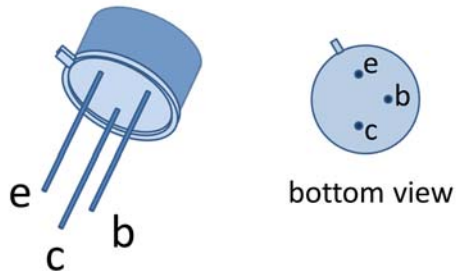


Figure 21. 2N2222 transistor pin diagram.

Verify that the LED is effectively controlled from dark to bright by varying the pot setting. Set the pot to an endpoint so that the LED is dark. Use the DMM to measure i_{LED} . Set the pot to the other endpoint so that the LED is at maximum brightness, and measure i_{LED} . Set the pot at the 50% point (estimate as best you can) and measure i_{LED} .

Have your TA initial your Lab 1 Data Sheet.

Online students: take a video that shows the LED brightness changing with the pot setting.

Appendix: Analysis of Transistor Dimmer Circuit

We assume a simple model for a transistor, which uses a current controlled current source and an independent voltage source, as shown below:

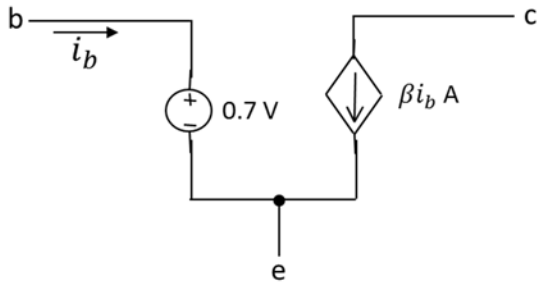


Figure 22. Transistor model.

Changing the $1\text{ k}\Omega$ resistor to $1.2\text{ k}\Omega$ in Figure 17, and then substituting Figure 22 for the transistor, we get the circuit shown below in Figure 23. The resistor substitution will be explained at the end of this appendix.

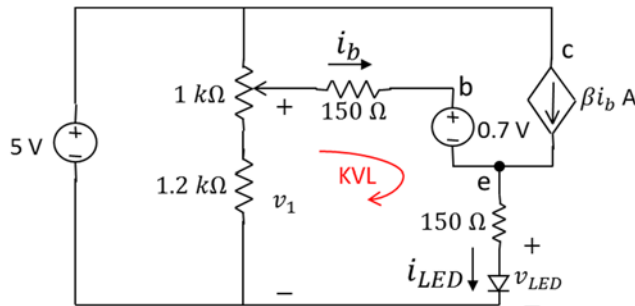


Figure 23. Dimmer circuit model.

In this circuit, i_b is very small (a few hundred micro-amps). Thus, the loading effect of this current on the voltage divider is also very small. Ignoring the loading effect, we then have $v_1 = \frac{1.2k + (1-F)1k}{1.2k + 1k} 5$. Simplifying yields

$$v_1 = 5 - 2.2F \quad (1)$$

Writing a KVL equation around the loop indicated in the figure yields

$$-v_1 + 150i_b + 0.7 + 150i_{LED} + v_{LED} = 0. \quad (2)$$

Writing a KCL equation at the node labeled e gives

$$i_{LED} = i_b + \beta i_b = (1 + \beta)i_b. \quad (3)$$

Substituting (1) and (3) into (2), setting $v_{LED} = 2.1$, and solving for i_b yields

$$i_b = \frac{2.2(1 - F)}{150(2 + \beta)}. \quad (4)$$

Substituting (4) into (3) gives

$$i_{LED} = \frac{(1 + \beta)}{(2 + \beta)}(1 - F)(0.015).$$

The parameter β is known as the DC gain of the transistor, and varies from transistor to transistor, even for the same part number from the same manufacturer. However, β is generally large. For example, for the 2N2222 transistor, β is guaranteed to be at least 75. Thus, the fraction $(1 + \beta)/(2 + \beta) \approx 1$, and so

$$i_{LED} \approx 15(1 - F) \text{ mA},$$

which has the desired linear form.

Final note: I originally designed the transistor dimmer circuit using the $1.2 \text{ k}\Omega$ resistor as described in this appendix. Due mostly to the approximation of Figure 8 vs. Figure 7, the LED current would not go all the way to 0 mA . I substituted $1 \text{ k}\Omega$ to get closer.