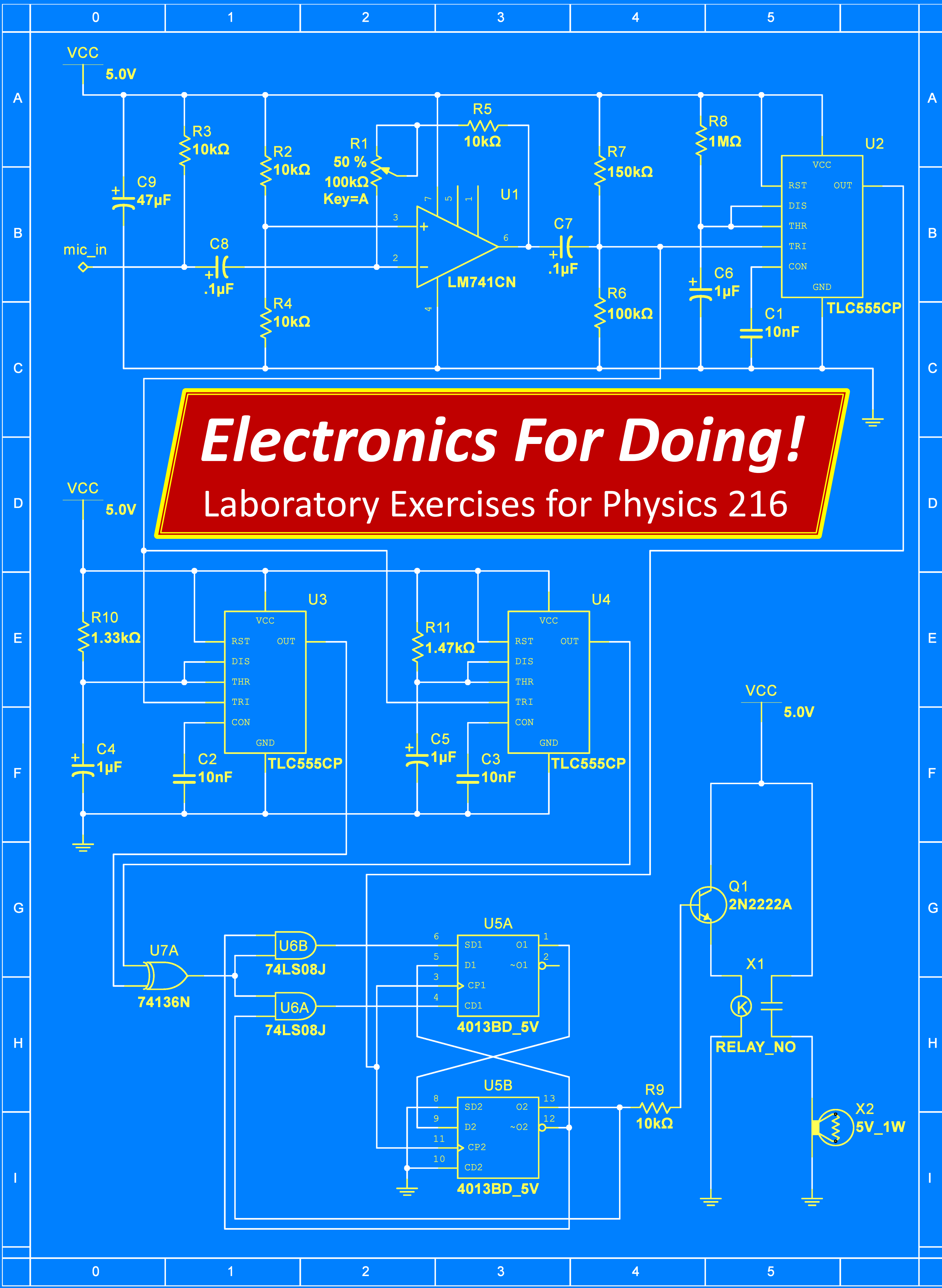
Electronics For Doing!

Laboratory Exercises for Physics 216



Cover art: The circuit diagram from the final class project of a student who took this course in a previous year. By the end of this course, you will understand all of the components in this diagram and will be able to design and build similar circuits yourself.

Electronics For Doing!
Laboratory Exercises for Physics 216

Matthew L. Trawick
Department of Physics, University of Richmond, VA

January 7, 2017

Welcome to Electronics!

The exercises in this manual are for a course in electronics that emphasizes active learning. Sitting passively and listening to lectures is boring, and doesn't help you learn very well either. Instead, this manual will guide you through a series of exercises designed to help you learn things by yourself as you go along, interrupted by me only occasionally to explain some of the tricky parts.

As you work, think about what you're supposed to be learning with each exercise. If an exercise shows you a circuit that does X, then you should walk away from the lab knowing exactly how to build a circuit that does X. Presumably, you'll be asked to do so on an exam soon enough!

Your written work will be done in a separate lab notebook. For each numbered part, you should document your work with a quick circuit diagram or note about what you did, and include any data you take and any conclusions you draw. Also, any sentence in this manual that ends with a question mark needs to be answered by you in your notebook.

Electronics is fun! Don't be afraid to experiment! The worst that can happen is you light some components on fire. (No worries—they're not yours!)

Enjoy the ride and be excellent,

—Matt Trawick

Contents

Lab 1 Getting to Know Your Equipment

Note: good preparation for this lab is looking over the introduction and chapter 2 of Paynter. This will help you to identify what's what in your kits.

1. Identify the values of some of the resistors in your kit, using the resistor color band code on page 54 of Paynter. What is the "tolerance" of most of your resistors, and what does that mean?

2. Use your digital multimeter (DMM) to measure the resistances of a 1 kΩ and 1 MΩ resistor from your kit. How close are they to their stated values?

3. What is the accuracy of the DMM for the resistance for the readings above? (Look in the manual for the DMM to find this out.) In general, which is more accurate: the color bands on the resistors, or a measurement from your DMM?

4. Measure the resistance of yourself. Does it make a difference if you lick your fingers?

5. What does the "range" button do on your DMM? What is "auto ranging" and how do you get it to do that?

6. What does the min/max/avg button on the DMM do?

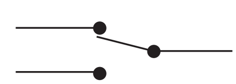
7. What does the yellow button on the DMM do, both in general, and specifically when you're measuring resistances? (What does the "))) " symbol mean, and when might that be useful?)

8. Use what you just found in part 7 to figure out which holes on your "breadboard" are connected to which other holes.

9. What is the resistance of connections made using your breadboard (both of the wires and of the contacts between wires and holes). If you can't measure it, can you at least set an upper limit?

10. The black banana jack on your proto boards is labeled with a triangle, which typically means "signal common." Often, this is also connected to "ground," that is, the Earth. Is this actually connected to the Earth? How can you test it for sure? What else on your proto boards is connected to "signal common?"

11. A "single pole double throw" (SPDT) switch has three terminals, as shown in the diagram to the right. The switches S9 and S10 on your proto boards each have eight little holes for wire connections. Which of those holes correspond to each of the three terminals? Which holes are connected to each other when the switch is up, and when it is down?



12. Use your DMM to measure the resistance of the switch when it is closed. If you can't make an accurate measurement, can you at least set an upper limit on the resistance?

13. How close to 5.0000000 volts is the 5 V power supply on your proto boards? Which is more accurate: the "5 V" printed on the proto board, or a measurement by your DMM?

14. Use your DMM to measure the actual range of the variable supplies $+V$ and $-V$ on your proto boards. Are their ranges exactly 1.3 to 15 volts when you turn the knobs?

15. Set your function generator to a 1.0 kHz sine wave, and measure its frequency and amplitude using your DMM. Hook it up to the speaker for fun. What's the highest frequency you can hear? (How does that compare with the highest audible frequency for your teacher, who spent way too much time playing in really loud, bad rock bands in college?)

Lab 2 Power Dissipation

Extra Apparatus

- digital multimeters (2)
- DC power supply
- various resistors
- banana cables and alligator clips

1. Your DMM has three different ranges for DC current measurement, and two different input jacks to plug in the leads for these measurements. How do you know which input jack and which ranges to use? What happens if you get it wrong?

2. Get several extra resistors (of different sizes) from the front of the room. All of these should have a resistance of about 100 Ω. Measure their resistances precisely. For each resistor, make a table in your lab notebook like the one below. Now hook each one up to the big DC power supply with alligator clips (NOT using your breadboards), and slowly increase the voltage of the power supply. At each voltage setting, measure the current through the resistor and the voltage across the resistor precisely, using two separate DMMs. Your actual voltage values will be slightly different from 1.0, 3.0, etc. From your measurements, calculate the actual resistance, and the power dissipated in the resistor at each setting. Also feel the resistor carefully to see if it is getting hot. When you are done with each resistor, let it cool and measure its resistance one last time with the DMM. Was there any permanent change?

(a) What is the difference between the physically large and physically small resistors?

(b) How do your observations compare with the nominal power rating for each resistor?

(c) Is the temperature coefficient of each of these resistors positive or negative? (That is, does the resistance increase or decrease as a function of temperature?)

Resistor: (describe value, physical size and appearance):					
ΔV(nominal)	ΔV(measured)	I	P	$\Delta V/I$	How hot?
0.3 volts					
1 volt					
3 volts					
10 volts					
30 volts					

Be sure to check in with the instructor after completing the table for your first resistor, to catch any misunderstandings before doing all of them!

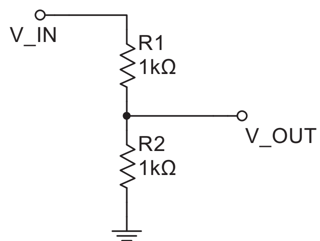
Possible Exam Questions:

- Describe a specific set of current and voltage measurements you could make of a resistor to determine whether it had a positive or negative coefficient. Give an example of numbers for a set of current and voltage measurements that would be consistent with a negative temperature coefficient.

Lab 3   Voltage Dividers

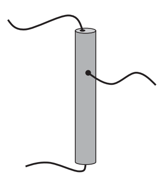
For this lab and from now on, you'll want to make your circuits on your breadboards, using the internal power supplies on your PB-503 Proto-Boards.

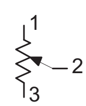
1. A voltage divider is a simple circuit made out of two resistors where the output voltage is a fraction of the input voltage. What is the ratio V_{OUT}/V_{IN} for the voltage divider shown below? Make a prediction, then build it and test it.



2. Design and build a voltage divider that will take an input of 5 V and give an output of 1 V. (It's fine to combine multiple resistors from your kits in series or parallel to get whatever equivalent resistance you need.) What is the power rating of your resistors? Calculate whether they can handle the power demands of your circuit before turning it on.

3. A trimpot (short for "trimming potentiometer," which nobody actually says in real life) is a three-terminal variable resistor. You can imagine a cylinder of an electrically resistive material, with fixed contacts on the top and bottom, and a third contact on the side that can slide up and down. In your kit, the trimpots are small ($\sim$ 1 cm) blue squares with white dials on them. Use your DMM to measure the resistances between each pair of terminals of your trimpot marked "5 0 2" as you turn the dial. Which of the terminals on your trimpots correspond to "1", "2", and "3" in the figure below?





4. Bonus challenge: can you crack the secret code to decipher how the markings like "5 0 2" and "1 0 3" relate to the resistance of each trimpot in your kit?

5. Use a single 5 kΩ trimpot to replace all of the resistors in the circuit you built in part 2. Build it and test it. What is the full range of output voltages you can get from it?

6. Put some additional fixed resistors in series with a 1 kΩ trimpot to build a voltage divider that will use an input voltage of 5 volts and provide a variable output voltage that is adjustable between 1.0 and 2.0 volts. (That is, those are the two output values when the trimpot is in its two most extreme positions.)

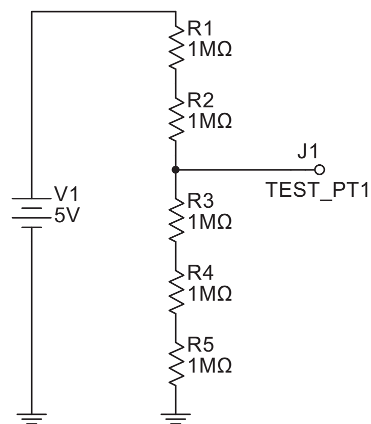
7. Suppose you want to build a voltage divider with a V_{OUT}/V_{IN} ratio of exactly 0.50000. If you use only a trimpot, then it can be very sensitive and difficult to set exactly. On the other hand, if you use only fixed resistors, their values aren't guaranteed to be exact, and you may end up with a ratio of 0.510. Use the same basic idea as in part 6 to design and build a voltage divider using two fixed 10 kΩ resistors and a trimpot, so that you can guarantee a ratio of exactly 0.500, provided the two fixed resistors are within their stated tolerances. What value trimpot should you use? Choose the smallest trimpot value possible for this. (What's the downside if the trimpot value is too big?)

Possible Exam Questions:

- Design a voltage divider using a trimpot and two fixed resistors (100 kΩ and 200 kΩ, both 1% tolerance) that will guarantee that $V_{OUT}/V_{IN} = 1/3$. What value trimpot is needed?

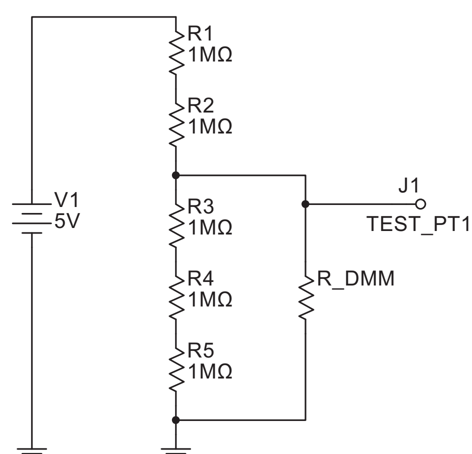
Lab 4 Input Impedance and Output Impedance

1. Make a prediction for the voltage at the test point indicated in the circuit below. Now build the circuit and test your prediction by measuring the voltage with your DMM. Are you surprised?

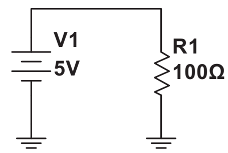


2. The reason for your surprise has to do with the input impedance of your voltmeter. (For now, you can take the word "impedance" to be a synonym for "resistance." In fact, you'll learn in lab 14 that "impedance" is actually a little more general.) What would be the input impedance of an ideal voltmeter? What's the actual input impedance of your non-ideal DMM when it measures voltage? (To find this out, you'll have to check in the manual for your DMM, under "specifications".)

3. The internal resistance of your DMM can affect the circuit you are trying to measure. To see how, consider the circuit at the right, which includes the actual input impedance of your DMM when it's used as a voltmeter. Use the value of R_{DMM} you found in part 2 to calculate the voltage at the test point in the figure below. Does it agree with the voltage you measured in part 1?



4. Make a prediction for the current through the 100 Ω resistor in the circuit below. Now build the circuit and test your prediction by measuring the current with your red DMM, using its mA scale. Are you surprised?



5. The reason for your surprise (possibly less so than before) again has to do with the input impedance of your DMM. What would be the input impedance of a (mythical) ideal current meter? What's the actual input impedance of your DMM when it measures current? Again, you'll have to check its manual. They use the funny term "burden voltage." (What are the units of that burden voltage, exactly?)

6. Again, the internal resistance of your DMM can affect the circuit you are trying to measure. Use the value you reported in part 5 to calculate the actual current through the 100 Ω resistor while you were measuring it with your DMM. Does your result agree with what you measured in part 4?

7. In this exercise, you will determine the internal resistance, or "output impedance" of your function generator. Set your function generator to a large amplitude, 1 kHz sine wave.

- First, use your DMM to measure the generator's AC output voltage when it is not hooked up to any load other than the DMM. This is called its "open circuit" voltage.
- Next, use your DMM to measure the generator's AC output current when it is connected to ground through your DMM. This is called the "short circuit current".
- Use the two measurements above to calculate the generator's internal resistance. Compare your result to the output impedance listed in its instruction manual.

8. The short-circuit current measurement in the last part is okay for a relatively high impedance device like your function generator, but it is potentially dangerous for low impedance devices that can produce very large currents. (Don't try short circuiting a car battery, for instance, or you could melt all your wires and cause the battery to explode!) A safer method is to put a known load resistance R_{LOAD} across the output terminals, and measure the voltage drop across R_{LOAD}. By doing this, you have essentially made a voltage divider circuit using R_{LOAD} and $R_{INTERNAL}$. Try this for your function generator. What size resistor should you choose for R_{LOAD}? How do your results compare with the previous part?

9. When your DMM measures resistance, it does so by putting a small "excitation current" through the resistor and measuring the voltage. How big is this excitation current? (If it's not in the specifications, you could always measure it using another DMM.)

10. Because the excitation current of the DMM is small, it's not very good at measuring resistances less than about an ohm. Use your 5 V power supply in series with a 100 Ω resistor to put a larger "excitation current" through a single, long jumper wire on your breadboard, and

measure the voltage across the wire to determine its resistance. What's the smallest resistance you can reasonably measure this way? Milliohms? Micro ohms?

Possible Exam Questions:

- You have a 5 V source, and you would like to build a simple voltage divider using two equal resistors to give an output of 2.5 volts. The 2.5 volt signal will go to an amplifier with an input impedance of about 50 kΩ. What value resistors should you use for your voltage divider: 1 kΩ, 50 kΩ, or 1 MΩ?
- You want to measure a current which you expect to be about 40 mA. That amount of current can be measured using either the 400 mA range or the 10 A range. What would be the advantage of using the 400 mA range? What would be the advantage of using the 10 A range? (Yes, there is a specific advantage for each one.)

Lab 5 The Oscilloscope is Our Friend!

1. Turn on your oscilloscope, and hook a probe into the BNC connector of the scope marked "CH1" (Channel 1). Use it to measure the output of a 1 kHz sine wave from your function generator, with the amplitude cranked up high (a maximum amplitude of about 10 or 12 volts). What does the VOLTS/DIV knob do? What does the "Div" mean?

2. Your probe has a little alligator clip at the end of it, which you probably connected to ground. Try disconnecting it. Why does it not matter here?

3. There's a tiny little switch on your probes, marked "1X" and "10X." (I'm NOT talking about the menu item on the scope screen, but the switch at the end of the actual probes.) What does this switch do?

4. The menu on your scope's screen should show various options, including "coupling," "BW Limit," "Volts/Div," and "Probe." (If it doesn't show that, probably because you were already playing with other buttons, hit the "Ch1 Menu" button.) What does hitting the button next to "Probe" do? How is this related to the switch on the side of your probes?

5. What does the SEC/DIV knob do? Measure the period of the sine wave by counting divisions on the screen. Is it consistent with a 1 kHz sine wave?

6. Sketch the shape of the waveform from your function generators for a triangle wave and a square wave. For a square wave, use your scope to measure about how long it takes for the voltage to rise from its negative value to its peak positive value. (Is it really instantaneous?)

7. Go back to a sine wave, and hit the "measure" button. Hit the first menu button (to the right of the screen), and then under "Type" have it measure "Mean." Hit the "Back" option, and set up the second menu button to measure peak-to-peak voltage. Set up the third measurement for RMS, a fourth one for "MAX," and a fifth one for "MIN." Show in a sketch which parts of the sine wave these are measuring. Which of these correspond to the measurements your DMM makes when it measures AC voltage?

8. How do the values of those measurements change for triangle and square waves? What the heck is RMS, anyway?

9. What do the knobs marked "position" do? (Vertical and Horizontal.) Notice on screen how you can tell when they're back to zero.

10. Use your second probe to measure the function generator output marked "TTL." Plug it into channel 2, and hit the blue "Ch 2 menu" button to turn on the second display. How is the TTL output affected when you adjust the amplitude and frequency of the sine wave (or triangle or square wave)?

11. Set your function generator back to a nice big 1 kHz sine wave, and hit the "trig menu" button. ("Trig" stands for "Trigger.") Cycle through all of the options under "Source" and describe what you see. What the heck does "triggering" mean? Back on "Ch1" for source, play with the "level" knob, and describe what happens.

12. With the level adjustment back on zero, adjust the amplitude of your function generator down to a point where the triggering really flips out and can't give you a decent looking clean sine wave. At this point, when you would normally despair of ever being able to get a clean measurement, go to the trigger menu, and select "Ch 2" under source. You may also need to adjust the trigger level slightly. Why is it better to trigger off of the TTL output than a sine wave?

13. Under trigger sources, after "Ch1" and "Ch2," what does "Ext" mean? (Hint: unplug the probe from channel 2, and plug it into the jack marked "Ext Trig.")

14. Plug the 2nd probe (measuring the TTL output) back into channel two, and go back to using channel 1 as a trigger. Crank up the amplitude to get a clean signal. Now hit the "Ch 2" menu, and start playing with the on-screen options. What does "Invert" do? What does "Coupling" do?

15. While you have pretty traces of ch1 and ch2 on your screens, hit the Cursor button, select "Voltage" under Type, and be sure the "Source" is set to either Ch1 or Ch2, whatever you have on your screen. Try twiddling the vertical position knobs. Cool, eh? What do the various other options do?

16. Use AC coupling to take a careful look at the output of your 5 Volt DC power supply. If you crank up the sensitivity of the scope using the volts/div knob, you should be able to see in great detail the small AC part that lies on top of the 5 volt DC signal. Adjust the time scale to get a good picture of what the AC part is really doing. What is the characteristic frequency (or frequencies) of this AC part? Where might they be coming from?

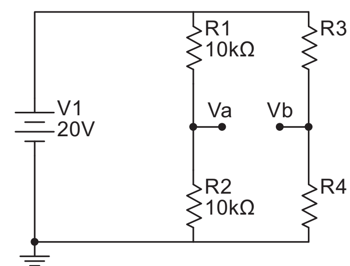
Possible Exam Questions:

- A signal varies sinusoidally between +4 volts and +6 volts. Describe what the oscilloscope trace would look like for AC coupling, DC coupling, and ground.
- Describe how to use the TTL output of your function generator to get good triggering for a precise measurement of a small signal. What inputs would you use, and how would you trigger?

Lab 6 A Bridge Circuit

1. Measure the resistance of a 10k resistor, and see if you can detect any change in its value if you heat it up between your fingers, being careful not to short circuit the leads. (You probably can't see any change.) Be sure you are not accidentally shorting out the resistor with your fingers as you do this. What is the smallest change in resistance that you could detect in principle, given the precision of your multimeter?

2. Now construct the bridge circuit shown below. For V1, use your variable DC power supply, $+V$, cranked up as high as it will go. Let R1 and R2 be 10 kΩ resistors. If R3 is 100 kΩ, what should R4 be so that the measured voltage ΔV_{ab} is zero?



3. Draw a circuit showing how how you can insert a trimpot in between R3 and R4, as you did in Lab 3, to get $\Delta V_{ab} = 0$ exactly.

4. Now see if you can detect a change in ΔV_{ab} if you warm up R1 with your fingers. How much does ΔV_{ab} change by?

5. What is the smallest change in ΔV_{ab} that you *could* detect, based on the precision of your DMM?

6. Based on your answer to part 5, what is the smallest change in the resistance of R1 that you *could* detect using this bridge circuit? How does this compare to what you found in part 1? (This bridge circuit is a good example of a fundamental and powerful principle of experimentation: that doing a difference measurement is usually a lot more sensitive than doing a single measurement directly.)

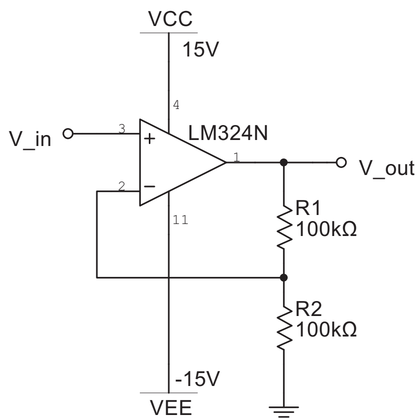
7. A typical temperature coefficient for a regular-old, smelly-old metal film resistor might be about 0.01% per degree Celsius. (That is, the resistance changes by 0.01% of its value for every 1 °C change in temperature.) From your previous answers, what is the smallest change in temperature that you could detect this way?

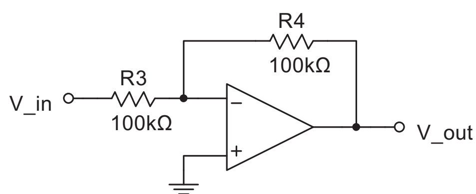
8. Suppose you actually wanted to analyze your bridge circuit taking into account the input impedance of the voltmeter. Is the voltmeter in parallel with another resistor, in series with another resistor, or neither? Show how you can use a "Delta to Wye conversion" (Paynter, pages 170–172) to redraw your bridge circuit as something you could actually analyze.

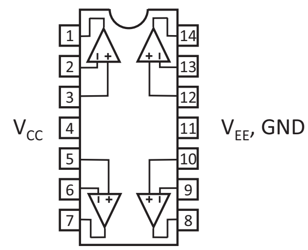
Keep your bridge circuit wired up on your breadboards. We'll be using it again at the end of Lab 7.

Lab 7 Some Basic Op-Amp Circuits

1. The circuit on the left, below, shows a non-inverting amplifier with a gain of 2. (Note that in the circuit diagram below, the four-way cross is NOT an actual electrical connection.) If $V_{in} = 2$ volts, what are V_+, V_-, and V_{out}? Modify this circuit to have a gain of 5, and test it out.





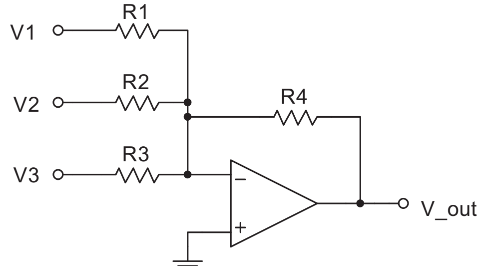


2. The circuit on the right, above, is an inverting amplifier with a gain of -1. (The power supplies V_{CC} and V_{EE} have been omitted from the diagram for clarity, as they often are, but they still have to be connected or it won't work.) If V_{in} is 2 volts, what is V_- (at the negative terminal), and what should be the current through R4? Modify the circuit to have a gain of -5, and test it out.

3. The circuit below is a summing amplifier, where the output is given by

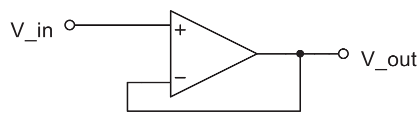
$$V_{out} = -R_4 \left(\frac{V_1}{R_1} + \frac{V_2}{R_2} + \frac{V_3}{R_3} \dots \right).$$

One place where such a circuit is useful is as a digital to analog ("D-to-A") converter. Suppose you have four digital inputs, whose voltages are all either +5 volts (for a logical "1") or 0 volts (for a logical "0"). These inputs represent the "ones place," the "twos place," the "fours place," and the "eights place" of a single number expressed in base two, so that "0001" is 1, "0010" is 2, "0011" is 3, and so on. Show how to use a summing amplifier as part of a 4-bit D/A converter, where the analog output ranges from 0 to +15 volts, corresponding to the numbers 0 through 15. Note that the output of your summing amplifier will be a negative voltage; how can you make it positive? (Also: you'll use this circuit again in part 8 of this lab, so don't destroy it when you're done.)



4. Set your function generator to output a 1 kHz sine wave with an (open circuit) RMS voltage of about 100 mV. (Measure it with your oscilloscope.) What happens to that RMS voltage when you connect the function generator to your speaker? From this, can you calculate the input impedance of the speaker? (This is similar to what you did in Lab 4, part 8.) What is the average power being dissipated in the speaker? Is it audible?

5. The circuit below is called a "buffer." What is its output, and why the heck would you ever want to build such a thing? Insert this buffer into your circuit for part 4, with the output of the function generator as V_{in}, and the V_{out} connected to the speaker. Now what is the RMS voltage across the speaker? And the average power dissipated in the speaker? What are the apparent input and output impedances of this amplifier? On a scale of 1 to 10, how cool is this?

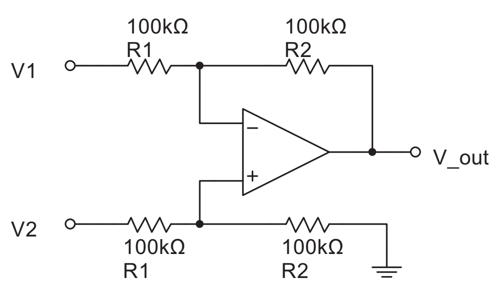


6. While monitoring both the V_{in} and V_{out} on your oscilloscope for the circuit in part 5, gradually increase the amplitude of the signal generator. How do V_{in} and V_{out} change? What is the maximum current through the speaker? Is this consistent with the value listed on the data sheet of the LM324?

7. The circuit below shows an example of a differential amplifier; for the general case, the output is given by

$$V_{out} = \frac{R_2}{R_1}\left(V_2 - V_1\right).$$

Use a differential amplifier like the one pictured below to create a triangle wave that sweeps between −8 and 0 volts. You can use the output of your function generator (set to whatever amplitude you want) and the output from your 5V DC supply, and you can change the values of the resistors R_1 and R_2 below to whatever you like.



8. (This part is intended to be long and complicated!) Your job is to design, build and test a "proportional controller," which could in principle be used to maintain a hotplate at a constant temperature.

- The temperature desired by the user (the "setpoint temperature") will be represented by an adjustable voltage which varies between 0 and 1.5 volts, set by a D-to-A converter like you already made in part 3 of this lab. (You can imagine that in a more sophisticated implementation, those four bits might actually be set by some kind of digital keypad or something.)

- The actual temperature will be read by a bridge circuit such as you built in Lab 6. The difference ΔV_{AB} from the bridge should be amplified to a level that also ranges between roughly 0 and 1.5 volts as you warm one of the resistors with your fingers.
- The output of your circuit should be proportional to the difference between those two voltages representing the setpoint temperature and the actual temperature. (A proportionality constant of 1 is fine.) Though you won't do it here, this output could be sent to a heater to maintain a constant, desired temperture. Incidentally, the cruise control in a car works the same way, pressing the gas pedal down by an amount that is proportional to the difference between the desired speed and the actual speed.

Hint: you'll be in trouble if you hook up V_A from the bridge circuit directly to the differential amplifier you made in part 7, because the amplifier's input impedance is too low. How can you correct this?

Possible Exam Questions:

- This is the first lab where you have been introduced to a truly new component (an op-amp) and several new kinds of circuits you can build with them. In this lab, you were shown examples of each circuit and asked to measure or modify them. On an exam, you won't have your hand held so much. You are responsible for knowing every circuit that you have built here.

  - If you are asked, "Draw a non-inverting amplifier with a gain of 10," you will need to draw one from memory, choosing appropriate values for any resistors you use.
  - If you are asked, "Design an 8-bit D-to-A converter," you will need to come up with a design similar to what you did in part 3 of this lab.
  - Alternatively, you may be shown any of the circuit diagrams from this lab, and asked "What does this do?" or "What is its input impedance?" or "What is the current through resistor R_2."

- Which has a greater input impedance: an inverting amplifier or a non-inverting amplifier?
- Estimate the output impedance of a non-inverting amplifier.
- When you build a non-inverting amplifier, you generally want to use resistors in the neighborhood of 10 kΩ to 100 kΩ. What could go wrong if you use values that are way too high, like 100 MΩ? What could go wrong if you use values that are way too low, like 10 Ω?
- From memory, what might be typical approximate values of an op-amp's (a) input and output impedances, (b) intrinsic gain, (c) maximum output current, and (d) slew rate?

Lab 8 An Introduction to Digital Electronics

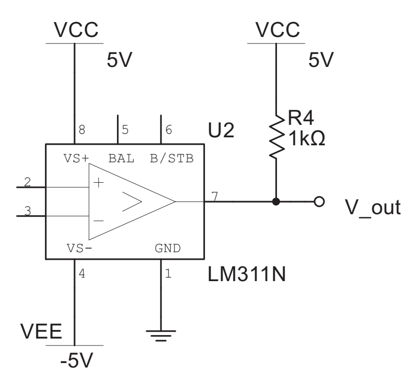
1. A 74LS00 chip has four separate NAND gates on it. (A pinout diagram is included in Appendix A.) To use this chip, you will need to connect pin 14 (V_{CC}) to +5 volts, and pin 7 to ground. Use the logic switches (labeled S1, S2, etc.) on your proto boards as inputs into pins 1 and 2, and connect pin 3 to one of the logic indicator lights. Does your chip produce the correct "truth table" for all four possible combinations of inputs?

2. Suppose that you need an inverter (a "NOT" gate) for a circuit, but you happen not to have one in your kit. Show how you can build your own inverter using a NAND gate. Also show how you can build an OR gate out of two NOR gates.

3. Build a simple controller that might control a set of security lights outside a home. The lights should go on when motion is detected nearby, but they should not come on during daytime hours. The lights should also go on if the homeowner pushes a "panic" button. If the lights come on due to motion outside, the homeowner should be able to turn them off by pushing a "clear" button. There are four digital inputs: motion, daytime, panic, and clear. Design and build a logic circuit with a single digital output that is high or low to control the lights based on these four inputs. Connect the output of your circuit to one of the logic indicators on the right of your circuit boards.

4. Logic circuits can be used to do arithmetic on binary (base two) numbers. Design and build a circuit that can add two 2-bit numbers. Each of the inputs can be either 0, 1, 2, or 3, represented by two input bits as 00, 01, 10, or 11. The output will be the binary representation from 0 to 6, using three output bits. Test your circuit, using the logic switches on your board to control the inputs, and the logic indicators to view the outputs. Keep good notes on how you build this circuit; you will use it again in a future lab.

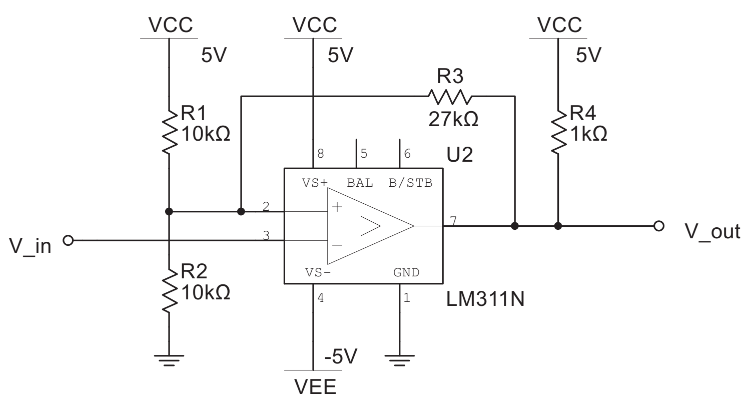
5. Sometimes you want to create a digital signal based on whether one voltage is larger than another. For example, you may want an alarm to be switched from OFF to ON if some temperature, represented by a voltage, rises above some predetermined limit. The drawing to the right shows how to use an LM311 comparator to do this. In the circuit below, if $V_+ < V_-$, then pin 7 gets connected to pin 1 inside the chip, so that $V_{OUT} = 0$. Otherwise, $V_{OUT} = 5$ volts. Use your LM311 comparator to build a circuit whose output is 5 volts if an input voltage is less than 2.5 volts, and the output 0 volts if the input voltage is greater than 2.5 volts. As a test input for your circuit, use a 1 kHz sine-wave from your function generator with an amplitude of 4 volts.



6. What happens to your output when you increase the amplitude of your test input to 10 volts? What determines the limitation on your input voltage?

7. If you look carefully at the output of your circuit in part 5 on a very fast time scale, you will see that each time the output switches, it actually makes multiple transitions, switching back and forth several times before settling down. Why would it do this?

8. To get only a single clean transition, you can make the modification of your circuit, shown below. What is the voltage V_+ at the positive input if $V_{OUT} = 0$ volts? What is V_+ if $V_{OUT} = 5$ volts? Predict the values of the V_+ in each case, and test your prediction by measuring it with your friend, the oscilloscope. How does the resistor R_3 prevent the multiple transitions you observed in the previous part? (This clever little trick is called a "Schmitt Trigger.")



9. In the previous lab, you built a D/A converter. Here, you will build a 2-bit A/D converter, to convert a variable input voltage of 0 to 3 volts to a set of two digital signals representing the numbers 00, 01, 10, and 11. (0.00–0.99 volts gives digital 00; 1.00–1.99 volts gives digital 01, etc.) The way it works is this: start by comparing the input signal to a 2 volt reference, and set the twos bit accordingly. If $V_{IN} < 2$, then compare it two a 1-volt reference, and set the ones bit. If $V_{IN} > 2$, then subtract off 2 volts from V_{IN}, and then set the ones bit by comparing $V_{IN} - 2$ volts to the 1-volt reference. Build and test your circuit, and sketch another circuit showing how you would extend this idea to build a 4-bit A/D converter.

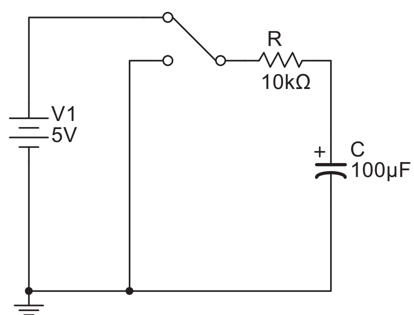
Possible Exam Questions:

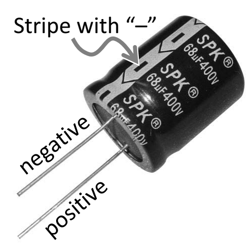
- Design a Schmitt trigger that gives a high output for $V_{IN} < 1.1$ volts, and a low input for $V_{IN} > 0.9$ volts. Yes, you really have to remember how to build one of these from scratch!
- Show how you can build an XOR gate using only OR and NAND gates.

Lab 9 Charging and Discharging a Capacitor

1. Identify a range of different capacitors in your kits. What are the largest and smallest values that you have? How do you read the value of the capacitance on the small ones? Use you DMM to measure some of their values, noting the uncertainty.

2. Build the circuit shown below. (*Be sure to connect the positive and negative ends of the capacitor as shown. If you reverse the polarity of the capacitor, it can blow up in your face and hurt you.*) Use your oscilloscope to measure the voltage across the capacitor V_C as you move the switch back and forth. You will want your scope to be scanning slowly at about 1 second per division. Sketch what you see. Show on your graph where your capacitor is fully charged, and where it is fully discharged.





3. Examine the graphs you have just drawn, and predict where on the graphs the current is a maximum, and where it is a minimum. Now insert your DMM as a current meter between R and C, and test your prediction. Were you correct?

4. When your capacitor is fully charged, throw the switch to discharge it, and measure the time it takes for V_C to drop to about 1/3 of its original value (that is, from 5 V down to about 1.7 V). This is called the "time constant" τ for this circuit.[1]

5. Make a prediction: if you increase the value of R, will the time constant increase or decrease? Write your prediction first, then test it out.

6. Make another prediction: if you increase the value of C, will the time constant increase or decrease? Again, write your prediction first, then test it out.

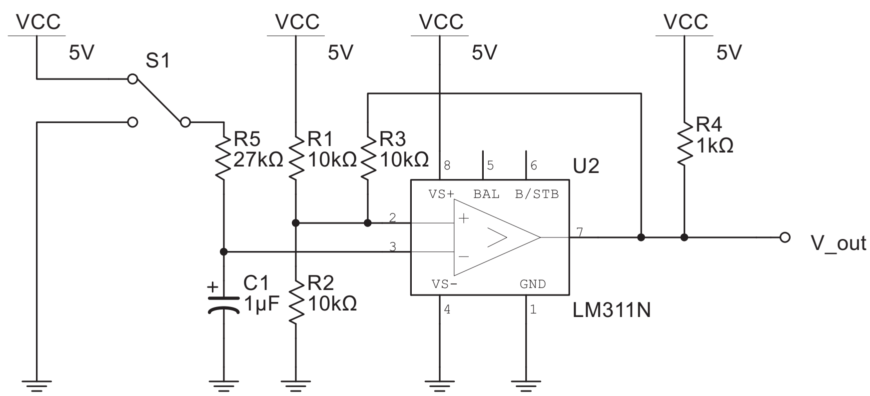
7. Now that you have several values of R, C, and τ, care to hazard a guess about an equation relating those three?

8. Make a circuit like the one above with a time constant of 100 microseconds. To verify the time constant with your oscilloscope, you will need to mess with the triggering for your scopes, so that the scope triggers only once as soon as you throw the switch, and doesn't trigger again. You will need to change the trigger mode from "auto" to "normal." (Look in your manuals. What do these two modes mean?)

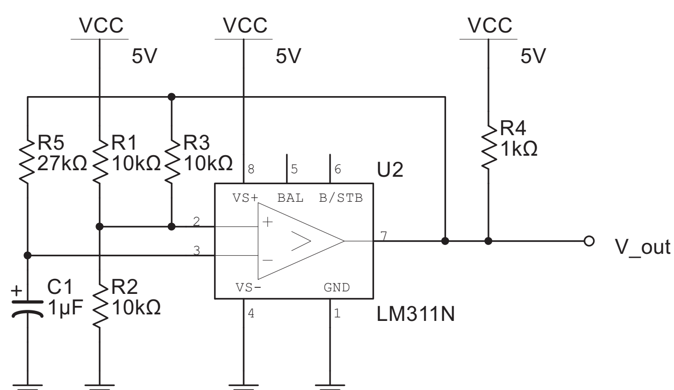
[1] More precisely, τ is the time for the voltage to fall to $1/e$ of its original value, where $e \approx 2.71828$.

9. You have now seen the behavior of V_C as a function of time for a discharging and charging capacitor. Can you guess the functional form of $V_C(t)$ in each case?

10. The circuit below is similar to the Schmitt trigger you made in a previous lab, but instead of an arbitrary V_{in}, the negative input of the comparator is connected to C_1 and R_5. For the component values shown, what is the voltage at the positive input if V_{out} is 0 volts? What is the voltage at the positive input if V_{out} is 5 volts? If the switch is moved from V_{CC} to ground, how long should it take V_{out} to change? How about when the switch is moved the other way?



11. Modify the circuit slightly, replacing the switch with feedback from the output as shown below. Draw three graphs of V_+, V_-, and V_{out}, all versus time, showing how they are synchronized with each other.



12. What capacitors and resistors can you change in order to increase the frequency of the output of the oscillator above? What's the fastest output you can reasonably produce with this circuit? Does it still look like a pretty square wave? Which component of your circuit is the limiting factor in the speed of the output?

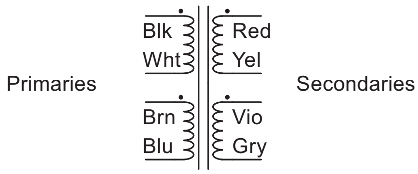
Possible Exam Questions:

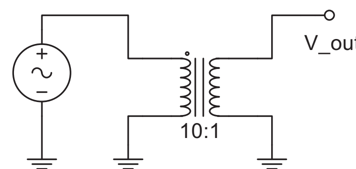
- A 200 nF capacitor charges through a 2 kΩ resistor, starting at zero volts. Graph both the current through the capacitor and voltage across the capacitor as a function of time. At what time is the capacitor voltage reach 90% of the supply voltage?

- What do “Auto” and “Normal” triggering modes mean on your oscilloscope?
- For the circuit drawn in part 11, graph voltage vs. time for each of the input terminals of the comparator.
- Use an LM311N comparator, as well as any other passive components you need (resistors, capacitors, etc.), to design a square wave generator with a frequency of about 150 kHz.

Lab 10 Transformers: More Than Meets the Eye

1. Grab a transformer and examine the wiring diagram on the left below. Notice that there are actually two primaries and two secondaries. Wire up your transformer so that the two primaries are in parallel and the two secondaries are in series, and show what color wires are connected where to make this happen. *Note that the small dots in the picture mark an arbitrary "positive" polarity. This matters because you wouldn't want current going clockwise in one primary coil and counter-clockwise in the other primary coil. Besides not working, that would blow out your transformer.*





2. Hook up your "primary" (actually two primaries) to your signal generator, set to about 60 Hz, and measure the ratio of the primary voltage to the secondary voltage. (The circuit above on the right shows what it would look like with just one primary and one secondary.) What "turns ratio" does your measurement suggest?" Is this a "step down" or "step up" transformer?

3. For this part, you will keep the wiring of the transformer the same: the side that's in parallel stays in parallel, and the side that's in series stays in series. But switch it so that the signal generator is connected to the other side of the transformer (that is, what was formerly the "secondary" of the transformer is now functioning as the primary, and vice versa). In this new configuration, is this a "step down" or "step up" transformer? What is the turns ratio? Is this consistent with your measurement in the previous part?

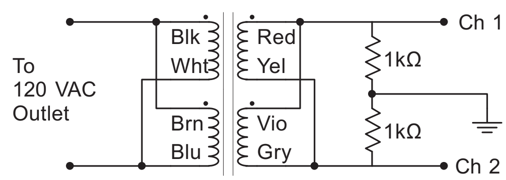
If a transformer steps the voltage UP, it has to step the current DOWN and vice-versa. (If it stepped both of them up, then it would be stepping up the power. If it could do that, then our nation's energy problems would be solved.) The way the transformer "steps up the current" is by effectively having a lower output impedance than the original source. Although this can't buy you power for free, you'll see in this next part that transformers can help you match impedances to use the power you have more efficiently.

4. Use your transformer to step down the voltage of your signal generator, and put the output from the transformer to the speaker. (Use a frequency of about 300 Hz, so you get an easily audible tone.) Does using the transformer make the tone louder or quieter? Calculate the average power dissipated in the speaker both with and without the transformer.

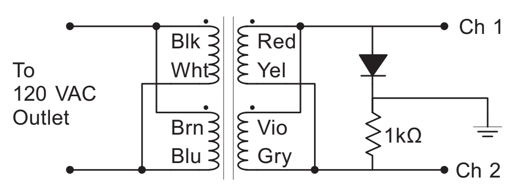
5. Now hook the transformer's primary up to an AC power cord, using two wire nuts, with the two sets of primary leads wired in parallel. Tie all the wires together in a knot when you are done, to prevent mechanical stress on the wire connections. FOR SAFETY, YOU MUST HAVE YOUR WIRING INSPECTED BY THE INSTRUCTOR BEFORE PLUGGING IT IN. Measure the output of your transformer's secondary. Based on the turns ratio you found in parts 2 and 3, what is the actual voltage of the AC outlet? Is that a peak voltage, or RMS?

Lab 11 Diodes: Basic Properties, and Clipper Circuits

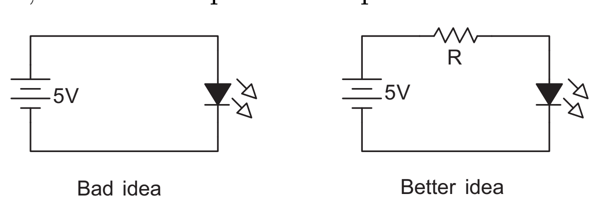
1. The circuit diagram below shows your transformer connected to two resistors, with the voltage across each resistor monitored by the oscilloscope. (Both the primaries and secondaries are connected in parallel!) Under the "display" menu of your scope, select a format of "XY" instead of the usual "YT." Also, under the channel 2 menu, select "invert on." In "XY" format, the scope will display a graph of channel 2 vs. channel 1, as opposed to showing them both vs. time as it usually does. Since the two resistors in your circuit are identical, the display here should show a 45 degree diagonal line, the graph of $y = x$. What does your display show if you disconnect either the channel 1 or channel 2 probe?



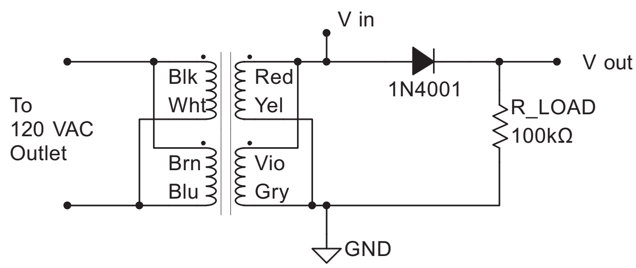
2. If you replace the first resistor in your circuit with a diode, your oscilloscope shows you a graph of current vs. voltage for the diode. Channel 1 (The X coordinate) is the voltage across the diode, and Channel 2 (The Y coordinate) is the voltage across the resistor, which is proportional to the current. Use this to measure the IV curve of the following diodes: 1N4001, 1N4733, 1N4739, 1N34 (which may have no label, but is identifiable by two black lines), and each of your color LEDs. What are their forward and reverse breakdown voltages, V_F and V_R? What kind of diode is each of these? (Silicon, Germanium, Zener....)



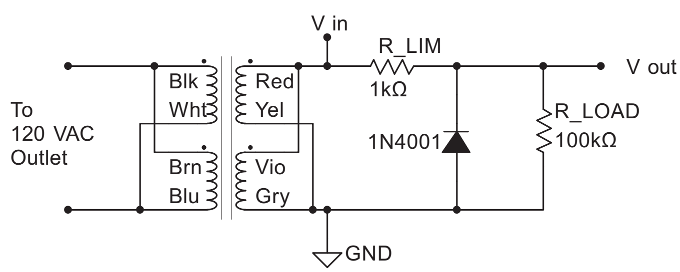
3. You have now seen that when a diode is forward-biased (that is, when it has current going through it in the direction of the arrow) that the voltage drop V_F across it is fairly constant, independent of current. ($V_F \approx 0.7$ volts for a silicon diode, $V_F \approx 1.8$ volts for a red LED.) To make your red LED light up, you want a current through it of about 10 mA; much more than that would blow it out. But if you hook the LED directly to a known voltage source, the actual current through it is hard to control precisely. Instead, you typically connect the LED to a power supply in series with a resistor, as shown below. In the circuit below, what value of resistor would yield a current of 10 mA through the LED? Test it out, and make changes if you don't measure 10 mA. Also, what is the power dissipated in the resistor and in the diode?



4. Return your scope to "YT" mode, and *do not forget to turn the Ch2 invert OFF*. Build the "series clipper" circuit pictured below, using a 1N4001 diode and a load resistance of 100 kΩ. Make a sketch of the input waveform and the output waveform. Make precise measurements of the peak positive and negative input voltages, and the peak positive and negative output voltages. Are they exactly the same? (You may want to shift the vertical positions of your traces and increase the scale on your scopes for more precise measurements.) Also predict how your graph will change if you reverse the polarity of the diode, and test your prediction.



5. Build the "shunt clipper" circuit picture below, using a 1N4001 diode. Use a 1 kΩ resistor for R_1 and a load resistance of 100 kΩ. Again, make a sketch of the input waveform and output waveform, and carefully measure the peak positive and negative input and output voltages. What is the maximum instantaneous current through the diode?

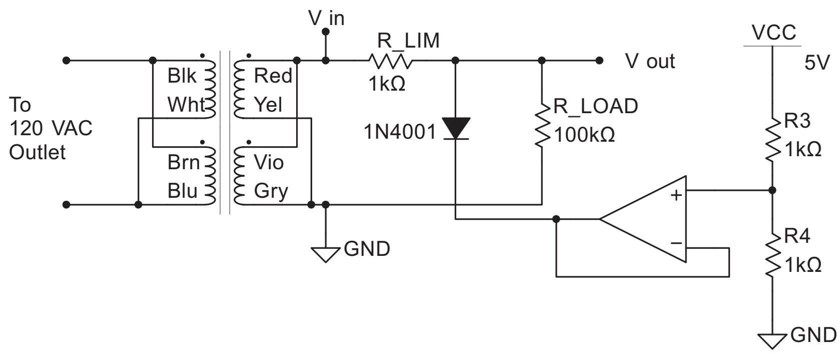


6. Think carefully about the role of R_{LIM}. Your transformer has a very low output impedance of about 2 Ω. If the 1 kΩ resistor were removed from the circuit, it would be as if you set $R_{LIM} = 2\ \Omega$. In that case, what would be the maximum instantaneous current through the diode? (Hint: consider both extremes, $V_{IN} = \pm V_{MAX}$.) What would be the maximum instantaneous power dissipated in the diode? What would be the effect of all of that current and power on the diode? The job of R_{LIM} is to limit the power and current in the diode.

7. Predict how the output voltages of the "series clipper" and the "shunt clipper" would be affected if the load resistance R_{LOAD} were changed to 1 kΩ. Check your predictions by building the circuits and testing them. What can you conclude about which design is better for producing high currents?

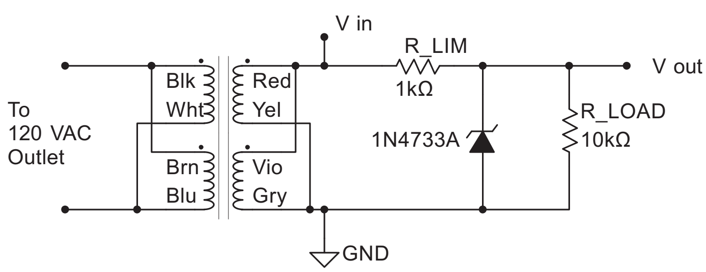
8. The circuit below is called the "biased shunt clipper." The diode's polarity has been reversed (just for variety), but more importantly it is now connected to a non-zero reference voltage instead of ground. Build the circuit and sketch the resulting output waveform for V_{OUT}. What is the relationship between V_{OUT} and the voltage at the junction between R_3 and R_4? Also, why is it necessary to use an op-amp in this circuit rather than connecting the diode to R_3 and R_4

directly? If you did omit the op-amp, how would you have to adjust the values of your resistors?

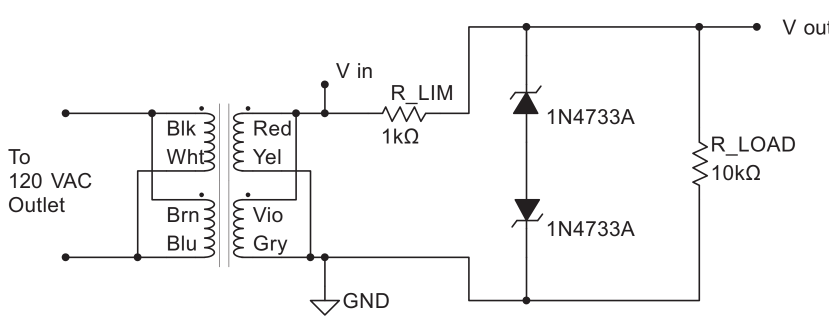


9. Show how you could modify the biased shunt clipper above so that the output voltage never dips below 0.0 volts, as opposed to −0.7 volts for the unbiased shunt clipper you built in part 5. (Incidentally, there's a very slick way to guarantee exactly the voltage you need at the op-amp input V_+ without having to calclate precise resistor values. Can you find it?)

10. Another way to build clippers that work at a variety of voltages is to use Zener diodes with specific values of V_R. Predict the maximum and minimum output voltages of the circuit below, which uses a 1N4733 Zener diode. Build the circuit, and test your predictions. What is the maximum instantaneous power dissipated by the diode? (Consider both $V_{Diode} = V_F$ and $V_{Diode} = V_R$.



11. The circuit below uses two Zener diodes in series. Again, predict the output of this circuit, then test your prediction. How would you modify this circuit to make a shunt clipper that gives $V_{-PEAK} = -9.7$ V and $V_{+PEAK} = +5.7$ V. (Hint: You don't need a biased clipper for this, because you can replace the 1N4733 with any kind of diode you want. In fact, if you look back at your measurements from part 2, you'll find one diode that's just *perfect* for what you need to do here.)



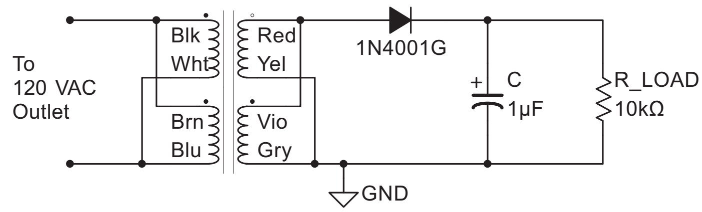
Possible Exam Questions:

- For the circuit drawn in part 8, calculate the maximum instantaneous current in the diode.
- Draw a graph of current vs. voltage for (a) a yellow LED, (b) a germanium diode, and (c) a Zener diode, including both positive and negative voltages.
- Design a biased shunt clipper that limits the output voltage to a maximum of 4.5 volts, regardless of the input voltage.
- For the circuit drawn in part 10, calculate the maximum instantaneous power dissipated in the resistor R_{LIM}.
- What are the relative advantages and disadvantages of a series clipper vs. a shunt clipper?
- As with all previous labs, you should be able to recreate any circuit in this lab from memory, choosing appropriate values of components as needed to get any desired behavior. You should also be able to calculate the value of voltage or current for any component given a diagram of any circuit you built here.
- In Lab 7 part 8 you built a proportional controller, where the output voltage was proportional to the difference between a setpoint temperature and a measured temperature. The idea was that the output voltage could go to a heater, which would heat just enough to maintain a constant temperature on something like a hotplate. But if you think about it, there was one small problem: the heater would also receive current (albeit negative) if the measured temperature was *higher* than the setpoint, which would make the hotplate heat up even more! Show how you could integrate the clipper circuit you built in part 9 of this lab into your circuit from Lab 7 part 8 to prevent this problem.

Lab 12 DC Power Supplies: Rectifiers, Regulators, and Ripples

The purpose of this lab is to show how to convert an AC voltage, for instance from a standard wall socket, into a stable DC voltage that you can use as a power supply for all of your circuits.

1. The circuit below shows the basic first steps. You use a transformer to step down the voltage somewhat, and then use a diode as a series clipper, which is also known as a "half wave rectifier." Make a sketch showing how the voltage waveform across the load resistance changes when the capacitor is connected or disconnected from the circuit. With the capacitor connected as shown, what are the minimum and maximum voltages across R_{LOAD}? How long is the rise time? How long is the fall time?



2. Predict qualitatively what will happen to the minimum and maximum voltages across R_{LOAD} if you double the value of C. Test your prediction with a quantitative measurement.

3. Predict qualitatively what will happen to the minimum and maximum voltages across R_{LOAD} if you increase the load resistance to 100 kΩ. Test your prediction with a quantitative measurement.

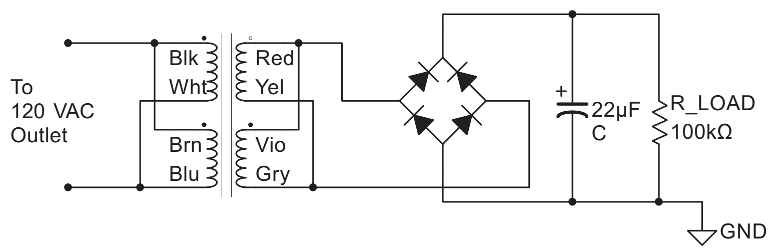
4. The difference between the minimum and maximum voltages across R_{LOAD} is called the "ripple voltage," and generally you'd like it to be small. In this part, you'll step through how to calculate the expected size of V_{RIPPLE}. Suppose $C = 2\ \mu$F, and $R_{LOAD} = 100$ kΩ.

- Based on the maximum voltage you measured in part 3, what is the maximum charge Q on the capacitor? What is the maximum current I through the load?
- How much charge ΔQ flows through the load in one cycle, assuming the current I through the load is approximately constant for the whole cycle?
- By how much will that change ΔQ lower the voltage across the capacitor?
- Does your calculation match with the measurement you made part 3?

5. Follow the same steps as in part 4 to calculate a general result for V_{RIPPLE} in terms of R_{LOAD}, C, V_{MAX}, and the frequency f.

6. Calculate the size of capacitor required to lower the ripple voltage to 100 mV for a load of 100 kΩ. Test your calculation. (As we continue to decrease the ripple voltage, you may find it handy to use the average function on your scope, under the acquire menu, to improve your measurements.)

7. Replace the half-wave rectifier with a full-wave bridge rectifier, shown below, using 1N4001 for all the diodes. As before, sketch the waveform both with and without the capacitor.

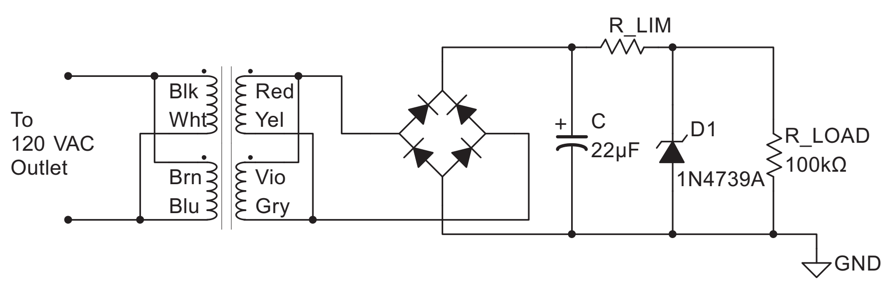


8. With the capacitor in place, you can measure that the maximum voltage across R_{LOAD} is slightly less than with the half wave rectifier. (How much less? And why is that?)

9. You should also see that the ripple voltage is about half what it was before. Why is that? Which part of your calculation in part 5 has changed by a factor of two?

10. Replace the four diodes with the integrated bridge rectifier from your kits, and confirm that it works as predicted. How are the leads labeled to indicate AC input and DC output?

11. To cut down the ripple voltage, you could of course make the capacitor much bigger, but that gets too large and too expensive pretty quickly. A better way is to use a Zener shunt clipper, as shown below. (The 1N4739 shown has a reverse breakdown voltage of $V_R = 9$ volts.) What value resistor is required for R_{LIM} if we want to protect the diode by limiting the reverse current in the diode to about 4 mA? (You should calculate that it's around 1 kΩ.) Build the circuit and measure V_{RIPPLE} and V_{DC} across the load, as well as the maximum and minimum of the voltage V_C across the capacitor.

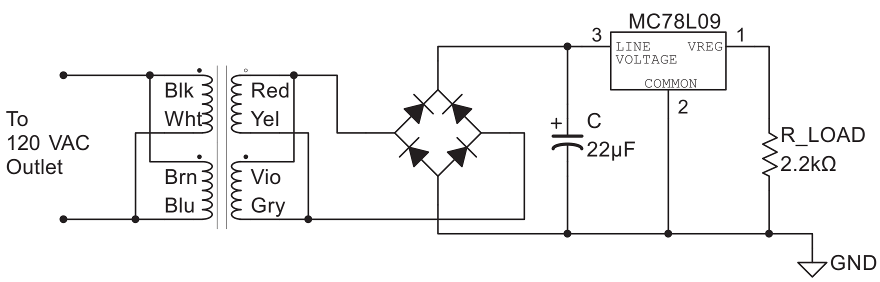


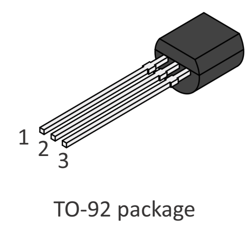
12. The circuit you just built should work fine as long as the value of R_{LOAD} isn't too small. But now what happens to your output when you change the load to $R_{LOAD} = 2.2$ kΩ? Why is your output no longer a steady 9 volts?

13. You can still make your circuit work with $R_{LOAD} = 2.2$ kΩ if you lower the value of R_{LIM}. What is the largest value of R_{LIM} that will make your circuit work? Fix your circuit using an "extra safe" value of $R_{LIM} = 330\ \Omega$ and redo the measurements you took in part 11.

14. Why is the output ripple voltage you found in part 13 larger than you found in part 11?

15. You have now seen the limitation of using a Zener shunt clipper for a DC power supply: for loads that draw large output currents, you need to lower R_{LIM} so that the output voltage never falls below V_R of the diode. But small values of R_{LIM} draw more current, which leads to larger ripple voltages, and would eventually lead to currents that would damage the diode. And fundamentally, the output ripple voltage is *always* going to be a problem for large currents, because the voltage across the diode isn't exactly 9.00000 volts—it always depends a little on the current in the diode. So you can't win! Fortunately, there's a terrific solution to all of these problems, and it comes in a little chip that costs just pennies and can happily output 100 mA. Build the circuit shown below, and verify that it produces a smooth 9 V output.





16. What happens when you replace the capacitor with $C = 10\ \mu$F? Why this huge change? (Hint: measure the input voltage to the 7809, and look up the "dropout voltage" on its data sheet. Be sure you're using the data sheet for the small MC78L09, in the TO-92 package.)

17. What capacitance C would you need if your load resistance were small enough to draw 100 mA?

18. Using $R_{LOAD} = 2.2$ kΩ and $C = 22\ \mu$F, measure the ripple voltage across R_{LOAD}, and compare that to the ripple voltage at the input of the 7809. By what factor has this ripple voltage been reduced? How does that compare to the value stated on the data sheet, which is given in dB?

19. You also have in your kits a 7805, in a larger TO-220 package. How much output current are these capable of? What is their "ripple rejection" ratio, in dB? How big would V_{RIPPLE} be if you used this in your circuit above?

Possible Exam Questions:

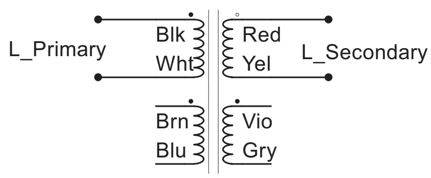
- Use a transformer, four silicon diodes, one capacitor, and a 7805 regulator to design a power supply to design a power supply that delivers 5 volts at 300 mA with 0.1 mV ripple. The 7805 has ripple rejection of 75 dB and a dropout voltage of 1.7 volts. Specify the capacitance and transformer turns ratio required.
- Derive the expression $V_{RIPPLE} = V_{MAX}/2fR_{LOAD}C$.

Lab 13   Introduction to Inductors

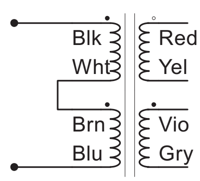
Note: before starting this lab, be sure both channels of your oscilloscope are set to DC coupling. Also set both of your probes to 10× to minimize the effects of probe capacitance.

1. Use your DMM to measure the inductance (in Henries) of several of the inductors in your kits. You should have eight of them to choose from. What can you deduce about the color code for the bands?

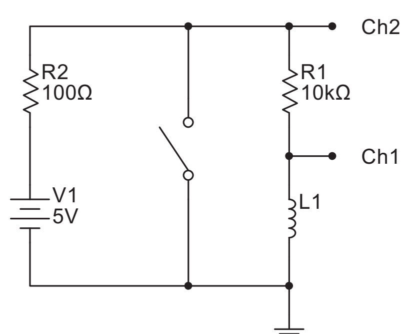
2. An inductor is basically a series of wire loops, usually around an iron or ferrite core. Your transformer consists of four sets of wire loops around a common iron core: two primaries and two secondaries. Make a prediction: which has a greater inductance: one of the primaries (say, white and black leads) or one of the secondaries (say, red and yellow). Test your prediction.



3. Make another prediction: given what you just measured for the inductance of one of the primaries, what value do you predict for the inductance of both primaries connected in series, as shown. Test whether your prediction was correct, and explain why it wasn't.



4. Build the circuit shown below, with channels 1 and 2 of your scope hooked up as shown, using the two transformer primaries in series as your inductor. Note that the switch is kind of funny here in that it shorts out the power supply. To measure the voltage across the resistor V_{R1}, you will need to push the "math" button on your scope to get it to display the difference Ch2 − Ch1. It's hard to get the switch to open and close cleanly every time; for best results, try to slide the switch quickly but silently. Sketch graphs of V_{R1} and V_L versus time as you open and close the switch. Also sketch the current I vs. time. What is the time constant τ of this circuit? (It should be in the neighborhood of about a millisecond.)



5. Sanity check: Does $V_{R1} + V_L = 5$ volts when the switch is open? What's $V_{R1} + V_L$ when the switch is closed?

6. In the water pipe analogy, we can think of power supplies as pumps, resistors as constrictions in the pipes, and capacitors as flexible rubber membranes that block the pipes. Take a look at your graph of current versus time, and think of an analogy for the inductor in your circuit. (Hint: something with a lot of mass or inertia.)

7. Make a prediction: what will happen to the time constant if you replace L with only a single primary coil of your transformer, instead of two primaries in series? Check it and see.

8. Make a prediction: what will happen to the time constant if you replace R_1 with a 5 kΩ resistor? Check it and see.

9. Figure out a mathematical relationship between R, L, and the time constant τ.

10. From the graphs of $V_L(t)$ and $I(t)$ that you found in part 4, can you find an explicit relationship between $V_L(t)$ and $I(t)$? (Hint: one of them is proportional to the time derivative of the other. Bonus points if you can find the constant of proportionality.)

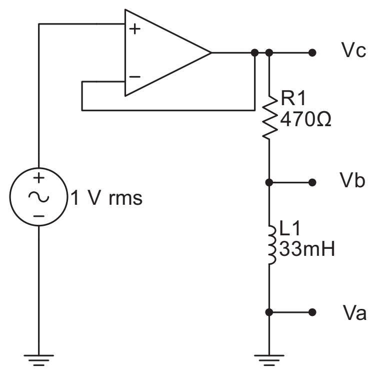
11. Compare your answer in the previous part to the situation for the RC circuit you studied in Lab 9. (Remake and remeasure the circuit in part 2 of that lab if you don't remember how it worked.) Make graphs of $V_C(t)$ and $I(t)$ as the switch is opened and closed. Can you find an explicit relationship between $V_C(t)$ and $I(t)$, where one of them is proportional to the time derivative of the other? Again, what's the constant of proportionality?

Possible Exam Questions:

- The primary of a 10:1 step-down transformer has an inductance of 1.4 Henries. What is the inductance of the transformer's secondary?
- An inductor ($L = 35$ mH) and a resistor (200 Ω) are connected in series to a 10 V power supply at time $t = 0$. Graph the current through the inductor and voltage across the inductor as functions of time. Find the time at which the voltage across the resistor is 4 volts.

Lab 14 Inductors and Capacitors in AC Circuits

1. Construct the circuit below. Use your DMM (not your oscilloscope) to measure all three RMS voltages V_{AB}, V_{BC}, and V_{AC}. Do it for frequencies of 20 Hz and 20 kHz. Is there more current in this circuit at high frequencies, or low frequencies?



	20 Hz	20 kHz
V_{AB}		
V_{BC}		
V_{AC}		

2. Think of the inductor as part of a voltage divider circuit. How does the inductor's "resistance" (actually its "impedance," Z_L) depend on the angular frequency ω, where $\omega = 2\pi f$?

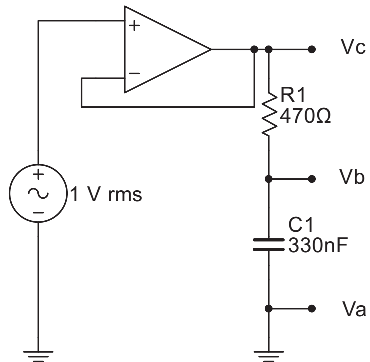
3. Adjust the *frequency* of the generator so that the RMS value of V_{AB} is about 0.3 volts, which should be at about $f \approx 1$ kHz. Predict what will happen to V_{AB} if the 33 mH inductor is replaced by 10 mH. Test your prediction.

4. Based on your previous answer, how does the inductor's "resistance" (actually its "impedance," Z_L) depend on its inductance L? Take a stab at writing a single equation relating Z_L to both ω and L.

5. (Today's shocker!) With the 33 mH inductor back in place, adjust the frequency of the generator so that the RMS value of $V_{AB} = V_{BC}$, which should be around $f \approx 2$ kHz. Is it still the case that $V_{AB} + V_{BC} = V_{AC}$? How can this be? Use your oscilloscope to measure all three voltages versus time, and demonstrate that Kirchoff's loop rule is not being violated here. (Use channel 1 for V_B, channel 2 for V_C, and the difference Ch2$-$Ch1 for V_{BC}.)

6. Graph the voltage across the inductor and the current as a function of time. What is the relative phase shift between them? Does the current "lead" the voltage, or does the voltage "lead" the current?

7. Now construct the circuit below. Again, use either the DMM or your oscilloscope to measure all three RMS voltages V_{AB}, V_{BC}, and V_{AC}. Do it for frequencies of 20 Hz and 20 kHz. Is there more current in this circuit at high frequencies or low frequencies?



	20 Hz	20 kHz
V_{AB}		
V_{BC}		
V_{AC}		

8. Think of the capacitor as a resistor in a voltage divider circuit: how does its "resistance" (actually its "impedance," Z_C) depend on the angular frequency ω, where ω, where $\omega = 2\pi f$?

9. Adjust the *frequency* of the generator so that the RMS value of V_{AB} is about 0.1 volts. Predict what will happen to V_{AB} if the 330 nF capacitor is replaced by 100 nF. Test your prediction.

10. Based on your previous answer, how does the capacitor's "resistance" (actually its "impedance," Z_C) depend on its capacitance C? Take a stab at writing a single equation relating Z_C to both ω and C.

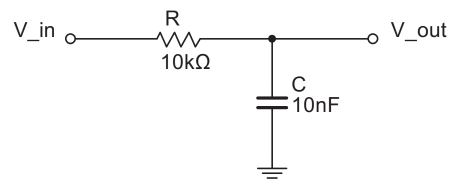
11. Adjust the frequency of the generator so that the RMS value of $V_{AB} = V_{BC}$. Again, is it the case that $V_{AB} + V_{BC} = V_{AC}$? Again, use your oscilloscope to measure all three voltages versus time, and demonstrate that Kirchoff's loop rule is not being violated here.

12. Graph the voltage across the capacitor and the current through the capacitor as a function of time. What is the relative phase shift between them? Does the current "lead" the voltage, or does the voltage "lead" the current?

13. You probably came up with laws for $Z_L = \omega L$ and $Z_C = 1/\omega C$, which is close to being correct. We'd like for the impedances and to follow something like Ohm's law, where $V_L = IZ_L$, and $V_C = IZ_C$. If $I(t) = I_0 \cos(\omega t)$, do the equations $Z_L = \omega L$ and $Z_C = 1/\omega C$ really give the correct voltages for the inductor and the capacitor? Why not? Think about your graphs for part 6 and part 12, and think carefully which graphs represents $I(t)$.

Lab 15 Fun With Filters

1. The circuit below is called a filter, because it "filters out" some frequencies but not others. Connect V_{IN} to your function generator, and measure V_{OUT}/V_{IN} for different frequencies. (Use 10 Hz, 30 Hz, 100 Hz, etc., up to 100 kHz .) Plot V_{OUT}/V_{IN} vs. the frequency f using Excel, and include a printout of the graph taped into your notebook.



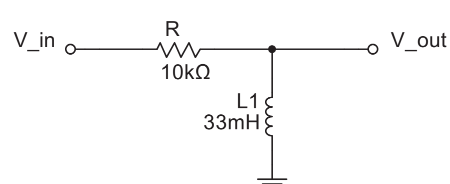
2. A low pass filter filters out high frequencies, allowing low frequencies to "pass through." A high pass filter filters out low frequencies, allowing high frequencies to pass through. Which kind of filter did you just build?

3. Plot your data from part 1 for V_{OUT}/V_{IN} in decibels vs. f, where f is plotted on a log scale axis. Show on the graph the frequency at which $V_{OUT}/V_{IN} = -3$ dB, called the "cutoff frequency," f_C. Is your value of f_C consistent with the expression you derived for V_C in your homework? Check your graphs with your instructor now to make sure you're on the right track.

4. Measure the phase shift ϕ between V_{OUT} and V_{IN} as a function of f, and make a rough plot of it. (You don't need many data points for this, just enough to give some idea of what ϕ does for $f \ll f_C$, $f = f_C$, and $f \gg f_C$.)

5. Design and build a *high pass* filter by switching the positions of the capacitor and the resistor in your previous circuit. Change the value of the capacitor so that the cutoff frequency is 100 Hz. Test your filter by recording V_{OUT}/V_{IN} vs. f and plotting the result in decibels on a logarithmic frequency scale as before.

6. You can also build filters using resistors and inductors. Is the filter pictured below a high pass or a low pass filter? Predict its cutoff frequency, and then test your prediction with a measurement.



7. Sketch how you would build a low pass filter using an inductor. If you wanted your filter to use a 10 kΩ resistor, what value of L would be required for a cutoff frequency of 10 Hz? What would be the physical size of such an inductor? By contrast, how big a capacitor would you need to make it work? (This is why in real life most filters are made using capacitors instead of inductors. Filters made using inductors are good for blocking very high radio frequencies called RF; for this reason inductors are sometimes called "RF chokes.")

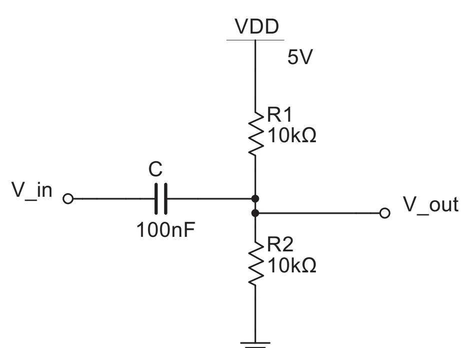
8. When you set the "coupling" on channel 1 or channel 2 of your oscilloscope to "AC" instead of "DC," you are actually filtering the input. Does "AC coupling" apply a low-pass filter or a high-pass filter? Take some measurements to determine the exact cutoff frequency of this filter. You should find that it's in the neighborhood of about 1 Hz.

9. Design a "band pass filter" that passes frequencies between 20 Hz and 20 kHz (roughly the audio range of human hearing). Do this by building a high-pass RC filter and a low pass RC filter and hooking the output of one to the input of the other. (Hint: you will want your first resistor to be about 10 kΩ, and your second resistor to be about 100 kΩ. Why?) Test your filter by making a graph of V_{OUT}/V_{IN} vs. frequency.

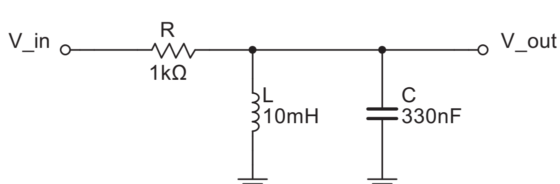
10. Use your scope to look carefully at the output of your DC 5 volt power supply. In particular, switch to "AC coupling" and crank up the scale to look at small deviations of the voltage. What does it look like? Is there a single dominant frequency in this "noise"? What else do you see in the signal? (You did this before, in Lab 5. This is just a reminder.)

11. Design a low-pass filter that will effectively filter out as much of this unwanted "noise" and interference as possible, to produce something closer to a clean DC output. Build it and test it, and draw what the output looks like. What should be the cutoff frequency of this filter? *Important note:* for this exercise, you will need to ground the end of your filter to a CLEAN grounding point. It turns out that the "ground" on your circuit boards is probably noisy as hell. (Measure it with the scope, in fact. How noisy is it?) Use a ground on your oscilloscope instead.

12. The circuit below shows an easy way to add a DC offset (Also called a "DC bias") to an AC signal. Use the principle of superposition to analyze this circuit, shorting out V_{IN} and V_{DD}. In each case, what is V_{OUT} for both very high and very low frequencies? Build it and test your predictions.



13. The circuit below is called a resonance filter. Make a plot of V_{OUT}/V_{IN} vs. f, paying close attention to the response at $f \approx 2.5$ kHz. Also plot the phase shift ϕ between V_{OUT} and V_{IN} as a function of f.



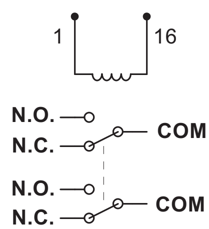
14. How would you modify this circuit to give a resonance frequency of 5 kHz? Build it and test your prediction.

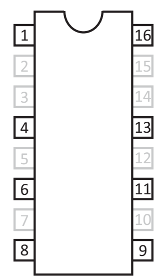
Possible Exam Questions:

- Design a band pass filter between 400 Hz and 1 MHz, using one capacitor and one inductor, plus resistors.
- Design a resonance filter with a resonance frequency of 1 kHz.
- Design a low-pass filter using a capacitor and a resistor with a -3 dB point of 4.7 kHz.
- Show how to use a capacitor and two resistors to add a 2 volt DC offset to an AC signal. What values of R and C are required for your circuit to work well at 3 kHz?
- A low-pass filter uses a 22 nF capacitor and a 4.7 k resistor. What is the ratio V_{OUT}/V_{IN}, in dB, at a frequency of 80 kHz?

Lab 16   Relays: One Circuit Controling Another Circuit

1. Find the relay in your kit. (It's a little blue boxy thing, about 1 × 1 × 2 cm, with 8 leads.) The switch connections are either "normally open," "normally closed," or "common." To actuate the switch, you need to put 5 V across the coil leads, which are pins 1 and 16. Figure out how the rest of the relay is wired up, by examining the mouse-sized writing on its underside and measuring it with your DMM. Draw a circuit diagram in your notebook similar to the one on the left below, including pin numbers for all eight leads.





DPDT relay

2. The switches inside your little relays can safely handle currents of about 1 Amp. Slightly larger relays, perhaps one cubic inch in volume, can handle currents of 10 or 20 Amps. By contrast, how large is the current in the coil of your relay? (Measure it! It should be a few tens of mA.)

3. Use your 5 V power supply to actuate the relay, opening and closing a circuit that connects your function generator to your speaker. Draw a good looking diagram of this circuit.

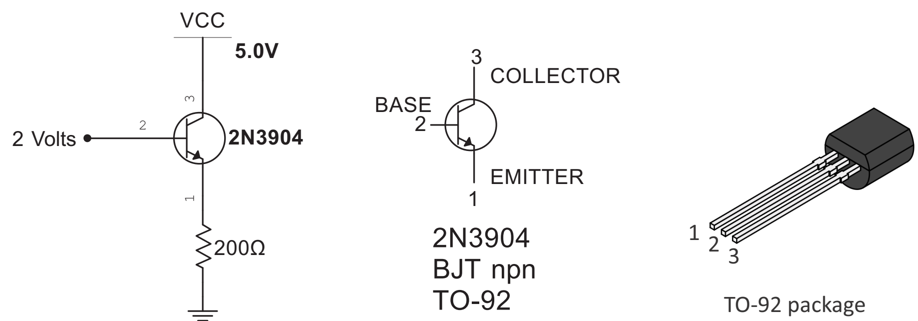
4. Change your circuit so that the relay opens and closes a circuit that connects a 15 VDC power supply to a 10 kΩ load resistor. Use your oscilloscope to monitor both the voltage across the relay coil and the voltage across the 10 kΩ resistor. Once you apply 5 VDC to the relay coil, how long does it take for the N.O. switch to close? How long does it take for the N.C. switch to open? (You should measure a few milliseconds for both.)

Possible Exam Questions:

- A digital "AND" gate is supposed to have a response time of 10 nanoseconds to any change in input. Describe how you would use an oscilloscope to test this response time. (How would you trigger, what coupling would you use, etc.)

Lab 17 Introduction to Bipolar Junction Transistors

1. Build the circuit drawn below, which includes a 2N3904 npn bipolar junction transistor. Measure the DC current through the base, collector, and emitter leads of the transistor. (To minimize the effect of your DMM on the circuit, use the "mA" scale where possible.) What is the ratio of I_E/I_B? Where is all the current in the emitter coming from?

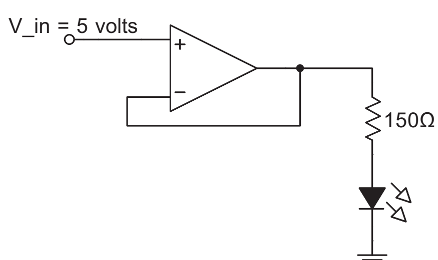


2. What happens to the voltage across the 200 Ω resistor as you raise the voltage of the base (carefully) to 3 volts? 3.5 volts? 4 volts? What is the voltage difference $V_B - V_E$?

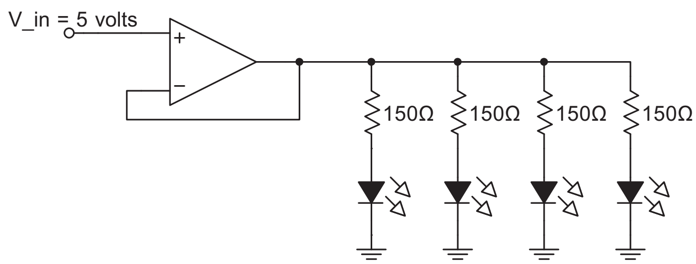
3. By connecting the base of the transistor to a DC voltage as you did in parts 1 and 2, you are effectively using the transistor as a switch. That is, by turning the DC base voltage on and off, you control current flow through the collector and the emitter. How fast does current turn on in the emitter when you connect the base to a DC voltage? (Measure it!) How does this compare with the switching speed of the relay switch you measured? (Note: You may not be able to get an accurate measurement of the delay time, but you should at least be able to set an upper limit.)

4. How does the size of the current I_B compare to the current in the coil of the relay switch from the Lab 16?

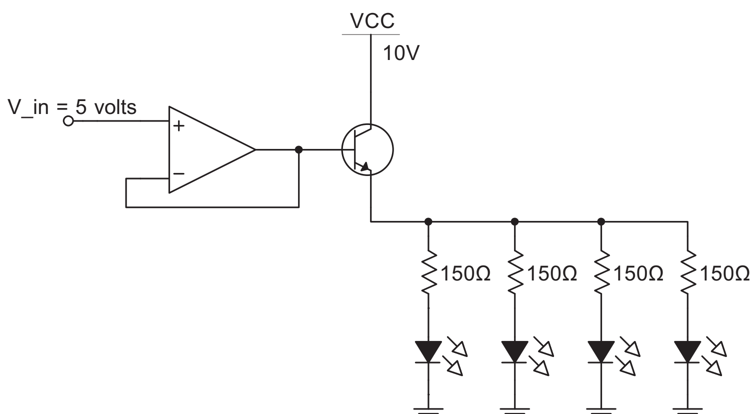
5. In the circuit below, the output of an op-amp is being used to light an LED. (Perhaps this is the last stage of some other more complicated project.) This circuit should work just fine. Build it and verify that V_{out} and I_{out} are what they should be.



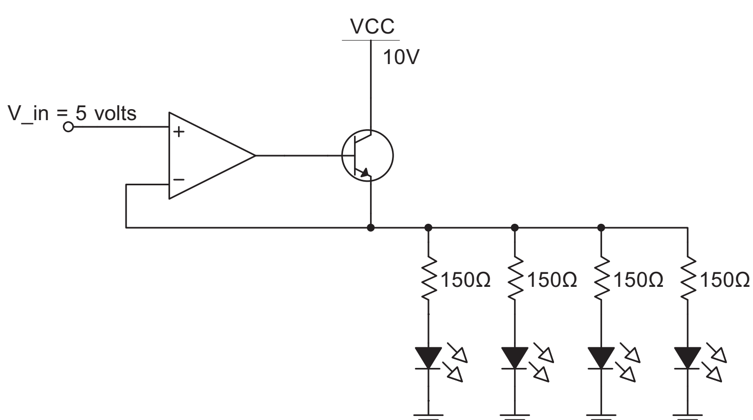
6. Now suppose that you want to use the op-amp's output to power three additional LEDs in parallel, as shown below. What is the maximum output current of the LM324? How is the current through the first LED affected by the addition of the other three?



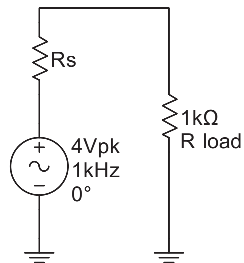
7. The circuit below shows one way to fix the problem you just found, using a transistor as an "emitter follower amplifier." Now what is the current through each LED? What is the voltage at the emitter, V_E? What is the current I_B?



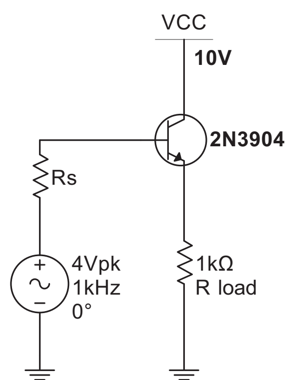
8. The circuit above has a problem, that the emitter of the transistor is always about 0.7 volts lower than the base, so that the voltage across the load is always 0.7 volts lower than you really want it to be. The circuit below shows an improved version of what you just built. In the circuit below, what is V_E, and what is the V_{OUT} of the op-amp? (Clever, eh?)



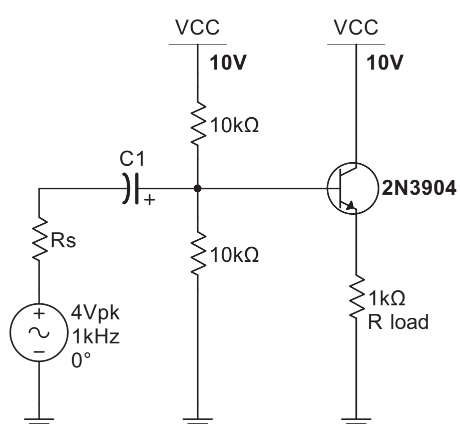
9. Set your signal generator to output a 1 kHz sine wave with a 4 V peak voltage with no load. Now hook it up to a load resistance of 1 kΩ and record the actual peak voltage across the load resistor. Is this consistent with the internal resistance of the signal generator? (See Lab 4.)



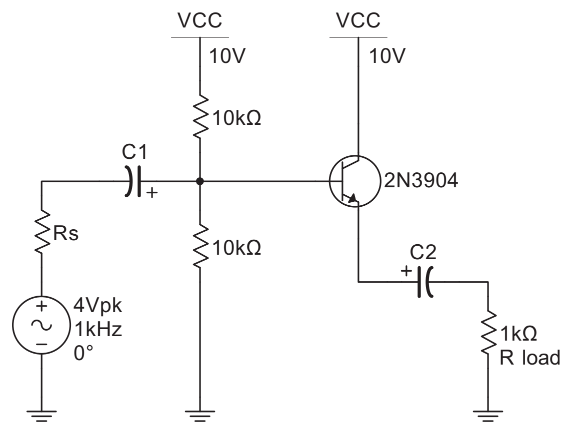
10. We've solved the impedance mismatch of the previous part once before, using a op-amp buffer. Let's see if we can do it with just a transistor! Connect your signal generator up to the base of the transistor, and connect the load resistor to the emitter. Sketch the waveform across the load resistor. Is this a good way to get a 4-volt peak sine wave across the load resistor?



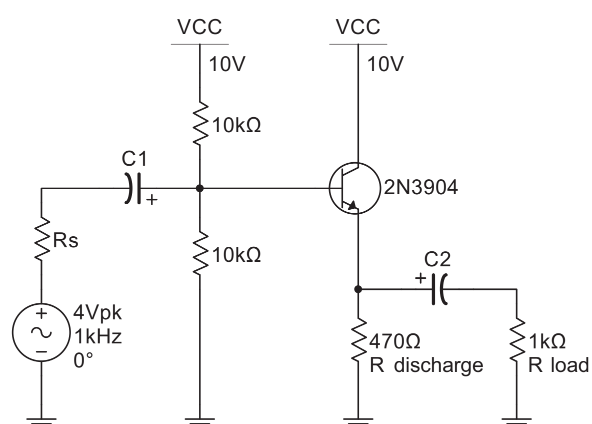
11. The circuit above is a lousy amplifier for AC signals, because it cuts off the bottom half of the input signal. The circuit below shows how to "bias" the input signal with a DC voltage so that the base of the transformer is always positive. What is the voltage at the base of the transistor in the circuit below (AC and DC components)? What must be the value of the capacitor for the cutoff frequency to be 100 Hz? (That is, at 1 kHz, you want the AC amplitude at the base to be the same as the AC amplitude of the source.)



12. To eliminate the DC offset on the output, your first instinct should be to use a high pass filter, as shown below. Measure the signal across the load resistor. Is it consistent with what you expect?



13. From the measurements you have just made, you have probably found that the circuit above is totally useless. It doesn't work, because current can never flow backwards across the transistor; the junction between the base and emitter is a diode, and current only flows across it forwards. Once the capacitor charges up to a maximum voltage, there is no way for it to discharge. To provide a path for it to discharge, you will need to use a second resistor in parallel with the load resistor, as shown below. The value of the discharge resistor has to be less than half of the load resistance for this to work properly. What values of C_1 and C_2 are required to make this amplifier work well over the entire frequency range of human hearing, from 20 Hz to 20 kHz? (Be careful: is each of these a high-pass or a low-pass filter?)



Possible Exam Questions:

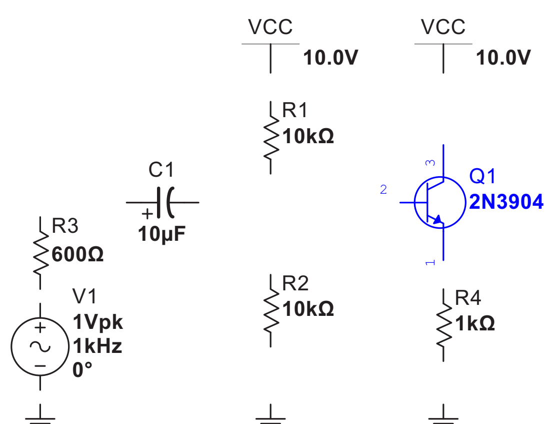
- In the drawing in the very last part of this lab, the upper 10 kΩ resistor is changed to 30 kΩ, and the capacitor is 100 nF. What is the cutoff frequency of this amplifier?
- If you want to use a voltage to turn the current in another circuit on or off, is it better to use a relay or a transistor? Discuss the advantages and disadvantages of each.

Lab 18   Multisim: How to Not Get Your Hands Dirty

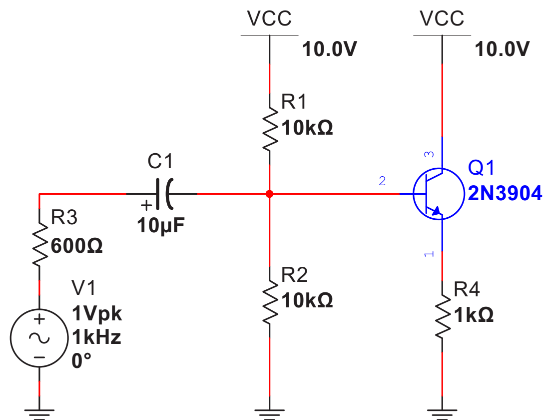
1. Open the program Multisim 14.0 on your computers (under Programs ⟶ Physics Applications). Under the "Place" menu, select "component" to add a component to a circuit. In the window that opens, under the "Group" menu, select "Basic", and under the list of "Families," select "RESISTOR." In the long list to the right, scroll down and select "10 k" to add two 10 kΩ resistors to the circuit diagram. After placing the resistors on the diagram, you will need to right click on each one, and select "Rotate 90° clockwise" to rotate them so that they look like the figure below.

2. Continue adding all of the components in the figure below, mimicking the circuit you built for a previous lab.

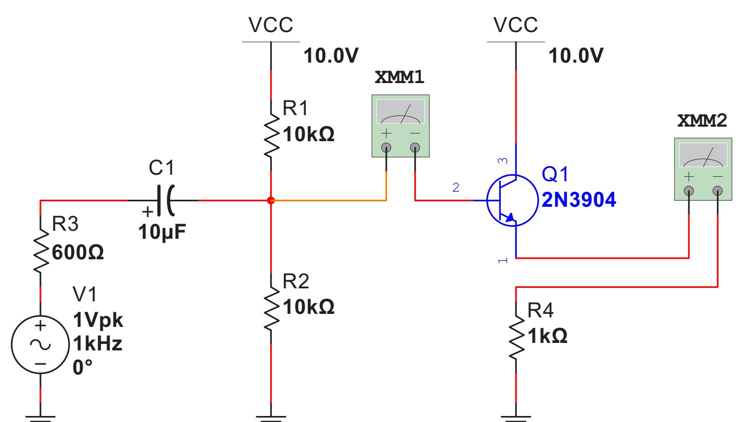
- The capacitor is under the group "Basic" in the family "CAP_ELECTROLIT."
- The 2N3904 transistor is under the group "Transistors" in the family "BJT_NPN"
- The ground symbol is under the group "Sources" in the family "POWER_SOURCES"
- The power supply VCC is also in the family "POWER_SOURCES." To set it to 10 volts instead of the default value of 5, right click and select "properties."
- The signal generator is under the group "Sources" in the family "SIGNAL_VOLTAGE_SOURCES"



3. Connect your circuit together. If you simply click on a terminal of one of the components, Multisim will automatically connect a wire to it. (Be sure you get a red dot at the four way junction, showing that all four wires are connected there.) When you finish, your drawing should look like the figure below. You can always move stuff around after wiring it if you need more room. Now would be a good time to save your work to your Box account or some other place you can get to later. Continue to save your work often throughout this lab, and make printouts as needed for your lab notebook (not for this step!).

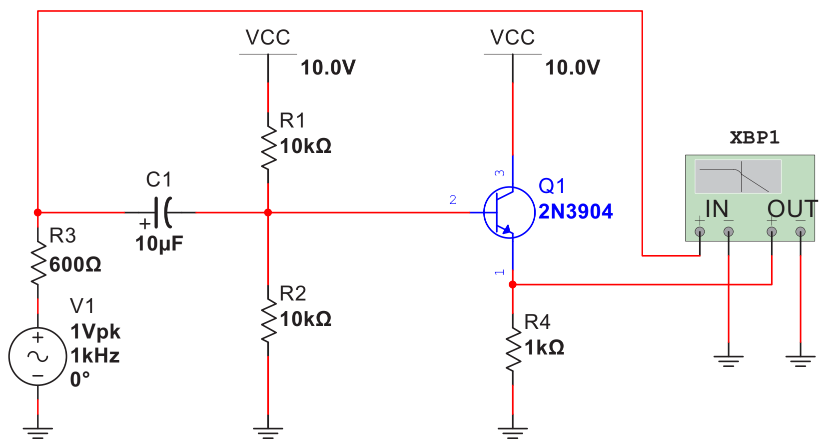


4. Making pretty pictures is all well and good, but Multisim also allows you to simulate the behavior of the circuit. Remove two connecting wires, and add two multimeters, as shown below. (Multimeters are the top item on the toolbar on the right edge of the screen.) Double click each one to set them to Amps DC. With the pop-up windows for each meter still open, choose "Simulate" then "Run" from the main menu. What is the apparent value of h_{FE} for this transistor? You can also start, pause, and stop the simulation by clicking the little icons (▶ ❚❚ ■) in the toolbar.



5. Add an oscilloscope to your circuit to measure the voltage across the load resistor. (The scope is the fourth tiny icon down on the toolbar on the right side of the screen). The positive lead to input A of the scope will go to your circuit; for the negative lead, you can place another "ground" symbol and connect it to your scope. Before you start simulating, make a prediction of what you should see (both AC and DC parts!). Is your prediction correct?

6. One of the coolest things Multisim can do is to show you the behavior of your circuit at different frequencies. Add a "bode plotter" to your circuit as shown below. (It's the sixth icon down on the right hand toolbar.) This tool gives you a graph of as a function of frequency. Zoom in on the graph by changing the maximum and minimum frequencies, and find the cutoff frequency for this circuit (the −3 dB point) using the little slider on the display.



7. As an alternative to the multimeters, Multisim allows you to add a "measurement probe" to monitor the voltage and current on a particular wire. (In the main menu, select Place ⟶ Probe, or use the little "V" and "A" icons just to the right of the stop/start buttons.) Remove the two current meters from your circuit, and replace them with "voltage and current" probes. What additional information do these give you?

8. Use Multisim to design a common emitter amplifier similar to the one we studied in class, but with a gain of −20. (That is, V_{OUT} is 20 times larger than V_{IN}, but phase shifted 180 degrees.) Connect the input of your amplifier to a 0.1 V AC source through a capacitor, as in the figure in part 6 above. Test your amplifier using Multisim to be certain it has the correct gain. (Hint: it will be easier if you increase the voltage of the DC supply to 20 volts.)

9. Connect the output of your circuit in part 8 to a 200 Ω load, and watch how everything instantly goes to hell! :-) The reason for this is that the load draws lots of current from your amplifier, which throws a monkey wrench into all of the resistor ratios you worked out so carefully. Use trial and error with Multisim to find out how large the load resistance has to be in order to keep the gain within 95% or so of what it is with no load.

10. To avoid the problem you ran into in part 9, connect the output of your circuit in part 8 to a second amplifier stage with a gain of $G = 1$, but with very low output impedance, suitable to drive a 200 Ω load. To keep this second stage from drawing too much input current (which would affect the first stage), you may need to use two transistors in the Darlington configuration. As in the previous lab, you will need to use an additional bypass capacitor on the output to get rid of the DC component of the output.

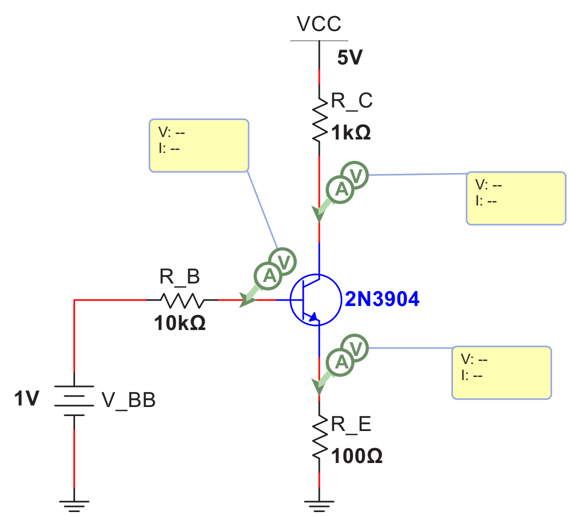
Possible Exam Questions:

- Use bipolar junction transistors to design a multi-stage amplifier with a gain $G = 100$, capable of a 5 V amplitude output signal. Use two stages, each with $G = 10$, with an additional emitter-follower stage in the middle to correct for any impedance mismatch. Use additional capacitors and resistors to correctly bias each amplifier stage so that $V_C > V_B > V_E$.

Lab 19   Saturation in Bipolar Junction Transistors

In this lab, we'll look closely at the behavior of an npn bipolar junction transistor under different operating conditions, simultaneously measuring the voltage and current at all three transistor terminals. Rather than trying to scrounge up six multimeters for the job, we'll simulate the transistor's behavior in Multisim instead.

1. Build a model of the circuit shown below in Multisim. Add voltage and current probes to all three leads of the transistor as shown. By default, the probes will display both the AC and DC components of the signals; to display only the DC values as shown below, click on the gear icon by the probe toolbox ("probe settings") and select "Instantaneous only."



2. Run simulations of the circuit for values of the voltage source V_{BB} = 0, 1, 1.2, 1.5, 2, and 5 volts. For each value of V_{BB}, record in a table the values of all six voltages and currents, and calculate the ratio of I_C/I_B.

3. You learned in Lab 17 that $I_C/I_B = h_{FE}$, where the constant value of h_{FE} is about 150 for the 2N3904. For what values of V_{BB} in your table does I_C/I_B have that approximate value?

4. You also learned in Lab 18 that the amplifier you designed only worked as you intended when $V_C > V_B > V_E$. For what values of V_{BB} in your table is that true?

5. In fact, bipolar junction transistors have three distinct "modes of operation" depending on the relative voltages of their terminals. For an npn transistor,

- $V_C > V_B \leq V_E$: "cut-off mode" (all currents ≈ 0)
- $V_C > V_B > V_E$: "active mode" (e.g. a working amplifier)
- $V_C < V_B > V_E$: "saturation mode"

Label each row of your data table to show which operating mode your transistor is in.

6. We say that a p-n junction is "forward-biased" when the p-type material is at a higher potential than the n-type material. (For a regular single diode, current flows when it is forward biased.) A junction is "reverse-biased" when the n-type material is at the higher potential. For each of the three operating modes above, indicate whether the emitter-base junction is forward- or reverse-biased, and whether the collector-base junction is forward- or reverse-biased. (Remember, the transistor you are using is an npn BJT.)

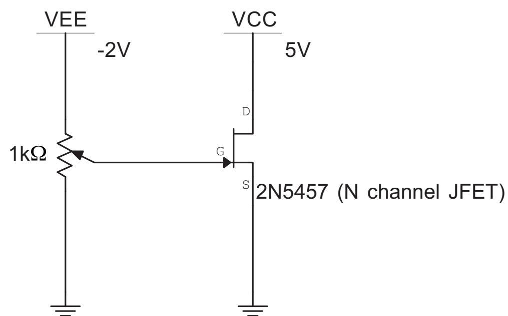
7. If you think about what's happening to V_C as you raise V_{BB}, you'll realize that your single BJT transistor can be used as a simple inverter, or NOT gate, using V_{BB} as the input and V_C as the output. (For this application, you can remove the resistor R_E from your circuit, or just set its value to $R_E = 0\ \Omega$). When you set the input to 0 V or 5 V, how close is the output to being exactly of 5 V or 0 V?

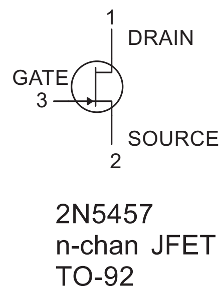
8. Which operating mode is your transistor in when the inverter's output is high? Which operating mode is it in when the inverter's output is low?

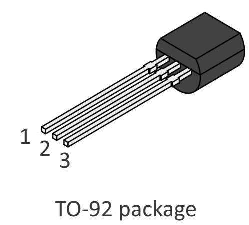
9. If you remove the resistor R_B as well, you can increase the voltage ΔV_{BE} substantially beyond the nominal 0.7 volts. However, when you do so, you'll pull a lot of current through the transistor, mostly from the base, and it will heat up like crazy. Based on its size, about what should be the maximum power you can dissipate in a small transistor like the 2N3904? Place a power probe on the 2N3904 to see how many Watts are dissipated in your transistor for various V_B. What is the maximum value you can set for V_B before the transistor would likely be ruined?

Lab 20 A Brief Introduction to a JFET Transistor

1. Build the circuit drawn below, which includes a 2N5457 transistor. This is a new type of transistor called a "Junction Field Effect Transistor," or JFET, and you've never seen anything like it before. The three leads of this kind of transistor have different names than the BJT transistors you have seen so far. The "drain" is analogous to the BJT's collector, the "source" is analogous to the emitter, and the "gate" is analogous to the base. *Notice that the 2N5457 pins are numbered differently from the previous transistor you used (the 2N3904), so be careful hooking it up!* Measure the current through the source as you vary the gate voltage from 0 to -2 volts. Make a rough plot of I_S vs. V_G.







2. Adjust the potentiometer so that the gate voltage is about -0.5 volts, and note the drain current. Without changing the potentiometer adjustment, move your DMM to measure the current in the gate. What's the ratio of I_S/I_G?

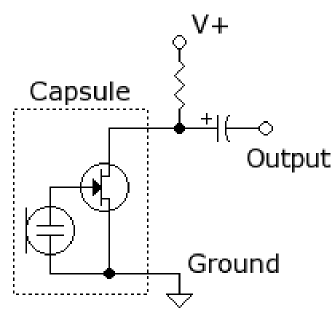
3. If you adjust V_G to increase the magnitude of the current I_G, would that *increase* or *decrease* the current I_S? Is the ratio I_S/I_G a constant?

4. In previous labs, we learned two basic rules for analyzing BJT circuits: $V_E = V_B - 0.7$ volts, and $I_C/I_B = h_{FE}$ (a constant). Does either of these two rules appear to hold true for the corresponding leads of this JFET transistor?

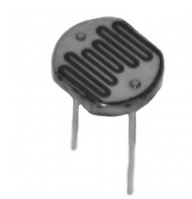
5. Given your answer to part 2, what would be the advantage of using a transistor like this in an amplifier circuit?

Lab 21 Sound and Light! (A Microphone and a Photocell)

1. One place where field effect transistors are useful is in an electret microphone. An electret is a material that holds a permanent polarization, something like the electric equivalent of a permanent magnet. This material is mounted on a movable diaphragm that vibrates in response to sound, so that its vibration induces a small oscillating voltage in a nearby electrode. A convenient way to detect this small voltage is to apply it to the gate of a field effect transistor. Your microphone (the "capsule," on the diagram) already has a transistor built into it. Connecting the microphone to an external power source and resistor and removing the DC bias with a capacitor as shown produces an AC signal that's easy to measure. How big is the resulting AC output when you whistle into the microphone? (Note that the leads of your microphone are polarized; the negative terminal is the one that's visibly shorted to the outside of the capsule.)

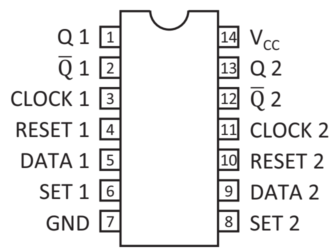


2. As long as we're playing with detectors, your kits also include a photocell, or "photoresistor." (It's the two-terminal thingy with a squiggly line on the top.) This is essentially a resistor made of cadmium sulfide, which is a semiconductor similar to silicon. The resistivity of this material changes in response to light, as incident photons excite charge carriers across the band gap. What is the resistance of this device in the ambient light of the room? When you put your thumb over it? Design and test a circuit that turns a current of 10 mA through an LED on and off when you pass your hand over a photocell.



Lab 22 Flip-Flops: A One-Bit Memory

1. A flip-flop is a device that holds a single bit of memory: either a logical “1” or “0”. The CD4013, shown below, contains two flip-flops (two independent bits) on the same chip. To wire up the flip-flop on one side of the chip, start by connecting V_{CC} and GND to 5 volts and ground. Connect all four inputs on one side (SET, RESET, DATA, and CLOCK) to four digital logic switches on your proto-board, all set to zero. Connect the two outputs Q and $\overline{Q}$ on the same side of the chip to two of the LED logic indicators on the right side of your proto-board. The line over the “Q” here means “not.” Do Q and $\overline{Q}$ give opposite readings like they’re supposed to?



CD4013
dual flip-flop
14-pin DIP

2. Start by playing with the SET and RESET inputs. What happens to the output when you change SET to 1 (actually 5 volts)? Does it change back when you lower SET back to 0? What about when you do the same thing with the RESET input?

3. If the SET input is held at 1, does the RESET input have any effect on the output? How about the reverse?

4. The SET and RESET inputs are one method of controlling the output of the flip-flop, called “asynchronous” operation. You can also change the output in a “synchronous” mode, using a clock signal. With the SET and RESET switches both off, set the DATA switch to 1. Then change the CLOCK switch from 0 to 1. At the moment the CLOCK input rises, the output should change to 1, if it wasn’t there already. What happens when you change the DATA input to 0? Write a rule that describes what CLOCK and DATA do.

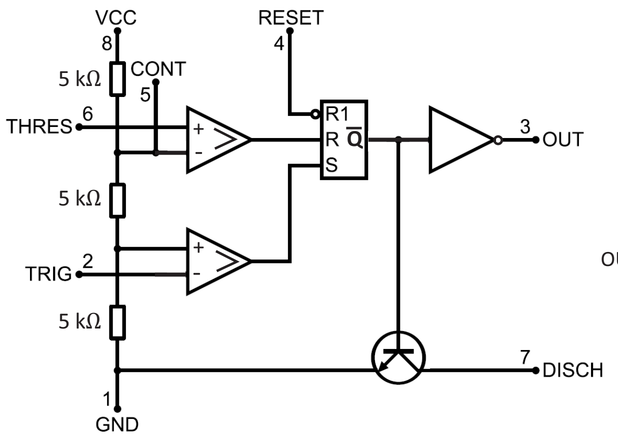
5. Does the CLOCK input have any any effect on the output when SET or RESET is held at 1? Do SET and RESET work when the CLOCK input is held at 1?

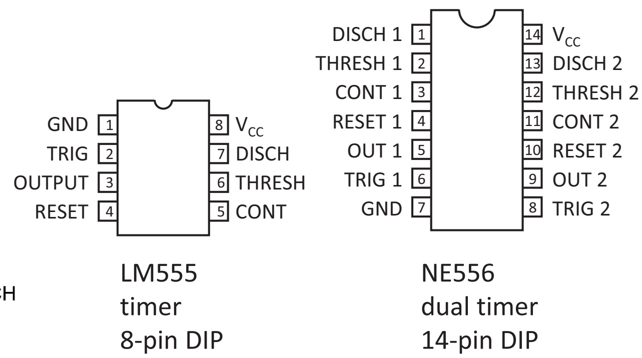
6. [*Optional*] Show how to use the two flip-flops on your CD4013 to wire up a set of buttons and lights that could be used to control a game show. Two contestants are asked a question, and each contestant hits their button (a switch) as soon as they think they know the answer. A light turns on over contestant “A” or “B” to show who buzzed in first. After the contestant answers, the host clicks a third button to turn both lights off. The tricky part here is that

each contestant will hit his or her button only briefly; once their hand comes off their button, the signal from the button turns off again, but you still need to hold that contestant's light on, *AND* prevent the other contestant's button from having any effect—that is, until the host has reset the system for the next question. (Hints: you can use the outputs Q or $\overline{\mathrm{Q}}$ as inputs to the system as well. If one player's "Q" turns to 1, what effect should that have on the other player's controls? You are allowed to use additional digital components like logic gates if you need to, but try to make your circuit as simple and elegant as possible.)

Lab 23   The 555 Timer (and Friends)

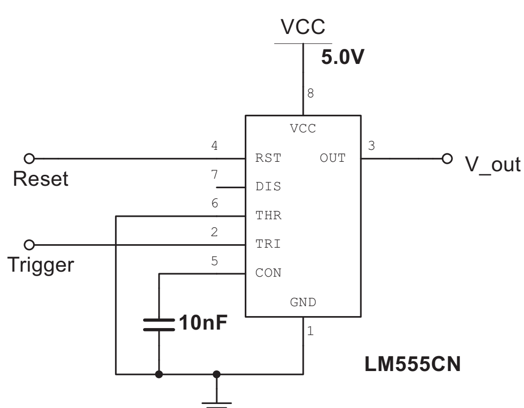
If you were trapped on a desert island and could have only a single integrated circuit with you, what would you choose? One good answer is the 555 timer, which consists of two comparators and one flip-flop, plus a few extra internal resistors and an output stage to increase its utility. As the block diagram below suggests, 555 timers can do anything that flip-flops or comparators can do alone, but they often simplify your circuit by requiring fewer external components.





The 555 timer was first developed in 1971, and is still selling over a billion chips per year. The original was the NE555 made by Signetics, but it is now made by many manufacturers and in countless variations (for high speed, low power, low voltages, extreme temperatures, etc.). It also comes in versions with two or even four devices on one chip. The pinouts for two common versions are shown above. In this lab, we'll consider three common uses for this best-selling chip of all time.

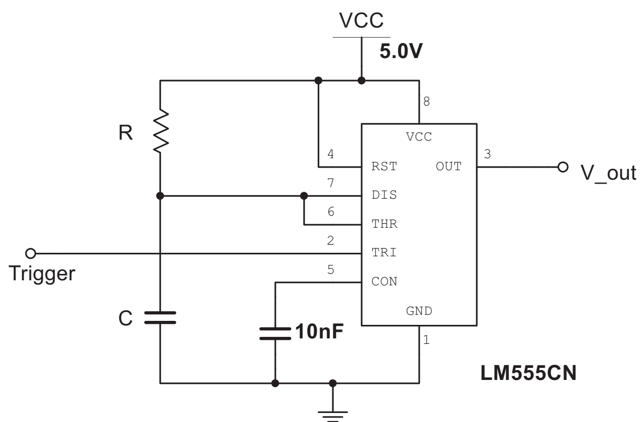
1. **Bistable mode**: When wired as shown below, the circuit acts like a simple flip-flop (but without the fancy clock and data inputs that you get with a chip like the CD4013). The inputs are normally held high at V_{CC}. When the trigger input is momentarily brought to ground, the output swings to V_{CC} and stays there. What happens when the reset is momentarily brought to ground? (Although the CONTROL input is not specifically used in this application, it is common nevertheless to place a 10 nF capacitor between CONTROL and ground, to reduce unwanted interference.)



2. **Monostable mode**: In this mode, the 555 acts as a "one shot" timer. When a trigger signal is given, the output signal jumps from 0 to V_{CC} for a time given by

$$\Delta t = \ln(3)RC \approx 1.1RC.$$

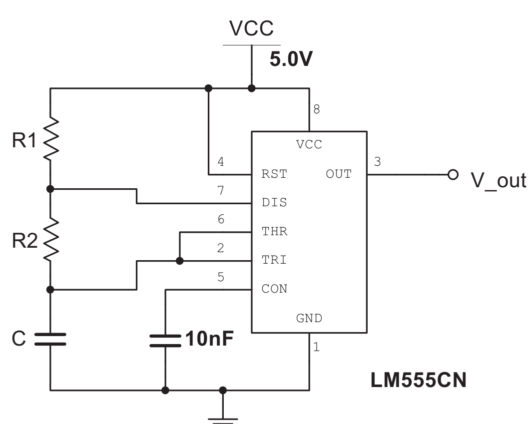
The output then resets to zero and waits for another trigger event. The trigger in this case happens when the input signal at the TRIG input falls below a certain fraction of V_{CC}. What fraction would that be, by the way? You should be able to tell by looking at the block diagram of the chip, especially those three 5 kΩ resistors in series. Make a prediction, then build the circuit to test it out. Aim for a delay time of $\Delta t \approx 3$ seconds.



3. **Astable mode**: In this mode, the 555 acts as a square wave generator, much like the one you built using a single comparator in Lab 9, part 11. In the circuit shown below, when the output swings high to V_{CC}, the capacitor C charges through R_1 and R_2. When the output swings low, the discharge lead (pin 7) is pulled to 0 V, and the capacitor discharges through R_2 only. Because of this, the square wave produced by this circuit is asymmetric, with the dwell times given by

$$\Delta t_{HIGH} = \ln(2)C(R_1 + R_2), \text{ and}$$
$$\Delta t_{LOW} = \ln(2)CR_2.$$

Build the circuit below, choosing values for R_1, R_2, and C that will yield a square wave with a ratio of $\Delta t_{HIGH}/\Delta t_{LOW} = 2$, and a frequency of $f \approx 10$ kHz.



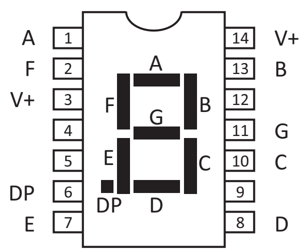
4. The "duty cycle" of a square wave is the percentage of each cycle that the output is at a high voltage. If you want to make a square wave with a duty cycle of less than 50%, there's a neat trick you can use to make the capacitor charge faster than it discharges: place a diode in parallel with R_2, oriented so that R_2 is effectively shorted out during the charging phase. Modify your previous circuit so that the output pulse has a duty cycle of about 20%. (Don't worry about its frequency, just get the duty cycle close.)

5. Use a single 556 timer chip (two timers on the same IC) to build a beeper that beeps a warning at about 1 kHz. The beeper should sound on and off about once per second. For this, you'll want to have one 1 Hz oscillator that effectively turns a second 1 kHz oscillator on and off every second. Looking at the circuit diagram in part 3, what's a good way to turn your oscillator on and off without cutting power to the entire chip?

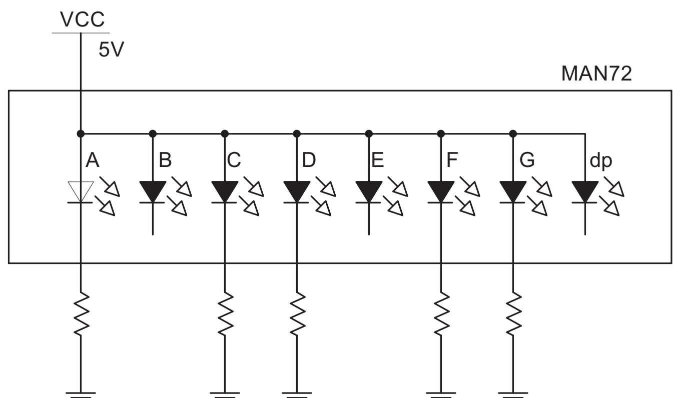
6. You probably noticed that the pitch of your beeper sounds a little unsteady. What's happening is that because of the low impedance of the speaker, the high current drawn from the 556 timer is causing some change in its performance, perhaps due to internal heating. To fix this problem, use a single MJE3055T transistor as an emitter follower amplifier between the 556 timer and the speaker. The pin-out diagram of the MJE3055T is shown in Appendix A. Draw a diagram showing how to wire this up, and build it to confirm that it works as expected.

Lab 24 Counters, Decoders, and Segment Displays

1. Your kit contains several 7-segment displays for displaying numbers. Find one of them with the characters “M72” on the side; this is the MAN72A, shown below. Each of the 7 LEDs on its face has its anode (negative terminal) connected to one of the labeled inputs (A, B, C, etc.), and its cathode (positive terminal) connected to pin 14 and pin 3. (Use either 14 or 3 for the connection; one is sufficient.) This configuration is called a “common cathode” arrangement, since all seven segments share the same positive terminal. What number would the display show as it is wired below?

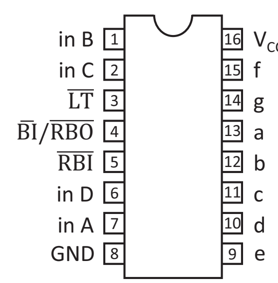


MAN72
7 seg display
14-pin DIP



2. A current of about 10 mA is required to make each segment light up brightly. *To limit the current, you will need to connect each anode to ground through an external resistor as shown in the diagram. If you hook any of the segments directly between 5 volt supply and ground without a resistor, the display will get blown out. (They’re diodes, after all; each is only supposed to have a forward voltage of about 1.8 volts.)* What value of resistor do you need to limit the current to 10 mA?) Draw a circuit showing expressly how to light the “B” and “C” bars for the number 1. Test your circuit, and verify that the current is what you think it is. By the way, which two leads on your chip control the left and right decimal points?

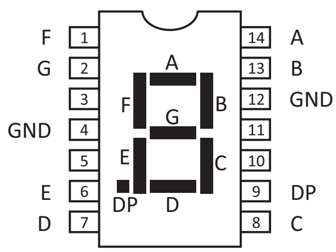
3. Suppose you have a series of digital signals that represent a binary number, as you did when you made a 2-bit adder. (Recall that you had 3 output leads, representing 1, 2, and 4.) Hooking up the 7 segment display directly to the output from your adder would be a royal pain; you’d need a LOT of logic gates to control which segments get illuminated for each of the possible combinations of output bits. Fortunately, you have a chip that does this for you: the 74LS47. Each of the 4 inputs A,B,C,D controls the 7 outputs (a,b,c,d,e,f,g) and can be hooked to the MAN72 chip. Hook the outputs up to your MAN72, and control the inputs directly with four logic switches. (Yes, you will have to use resistors!) Does it behave as you expect? What happens when you give it a binary number greater than 9? (NOTE: any unused input A, B, C, or D has to be grounded; you can’t just leave them hanging open.)



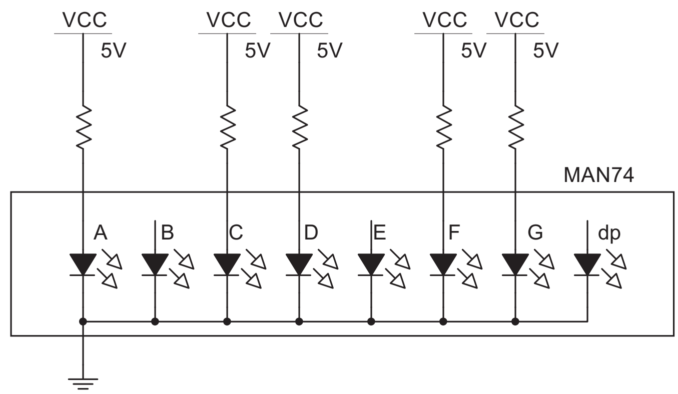
74LS47
4-bit decoder
16-pin DIP

4. What happens when you ground the input $\overline{\mathrm{LT}}$? What about when you ground the input $\overline{\mathrm{BI}}$? The bar over the letter means “not”; in other words, whatever thing “LT” stands for will happen when that input is FALSE (0 volts) as opposed to when it’s TRUE (5 volts).

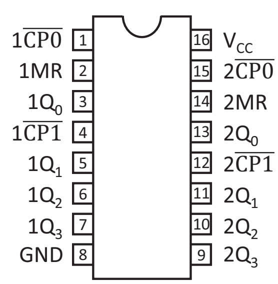
5. For comparison, the MAN74A, which is a “common anode” configuration, is shown below, as it would be wired to display one possible number. Could you use the 74LS47 chip with the MAN74A? (If yes, then build a circuit and show how. If no, then explain why not.)



MAN74
7 seg display
14-pin DIP



6. Your kits contain another chip that counts: the CD74HC390E, whose pin-out is shown here. Connect Vcc to +5 volts, and GND to ground. Connect pin 4, labeled $1\overline{\mathrm{CP1}}$, to the TTL output of your function generator at 1 Hz. Connect the outputs $1Q_1$, $1Q_2$, and $1Q_3$ to three logic indicators on your breadboards. The input 1MR should also be grounded. How high does this chip count (in binary) when it is wired in this configuration?



CD74HC390E
dual counter
16-pin DIP

7. What happens when you disconnect the input 1MR (called the “master reset”)? What happens when you connect 1MR to +5 volts?

8. What condition at the clock input $1\overline{\mathrm{CP1}}$ is required to make the counter increment? (A rising signal? A falling signal?)

9. Now connect the TTL output of your function generator to the clock input $1\overline{\mathrm{CP0}}$, and connect the output $1Q_0$ to the clock input $1\overline{\mathrm{CP1}}$. Use your logic indicators to monitor all four outputs, $1Q_0$ through $1Q_3$. How high does the chip count in this configuration?

10. Predict what will happen if you change the configuration of the previous part so that the master reset 1MR is connected to the output $1Q_3$. How high will the chip count? Test your prediction.

11. Perhaps you've noticed that your counter has a whole second set of inputs and outputs on the other side. Show how you can hook up one of the outputs of the first set to one of the clock inputs of the second set to count from 0 to 99. Connect each set of outputs to a 74LS47 decoder, controlling a 7-segment display. Cool, huh?

12. How can you modify your counter to count from 0 to 39?

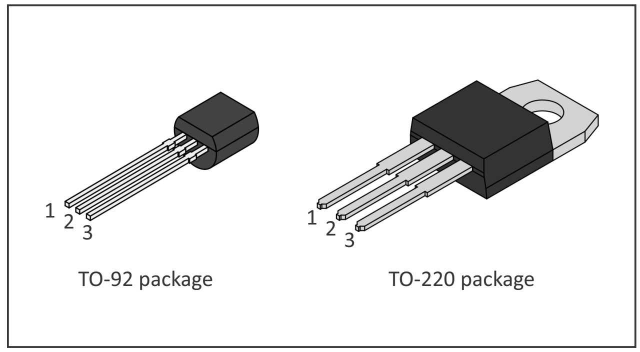
13. What's the highest you could count to if you had three CD74HC390E chips?

Lab 99 Student's Choice

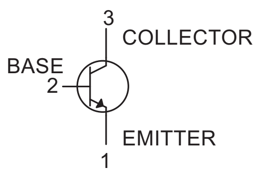
1. Think of something cool that you could build using what you've learned in this class. Maybe even something you've always wanted to build, but didn't know how to begin.

2. Build it.

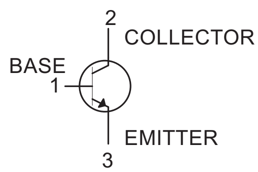
A Pin-Out Diagrams of Common Components



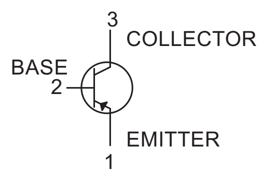
TRANSISTORS:



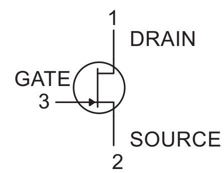
2N3904
BJT npn
TO-92



MJE3055T
BJT npn
TO-220

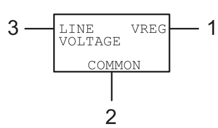


2N3906
BJT pnp
TO-92

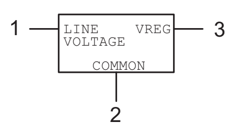


2N5457
n-chan JFET
TO-92

VOLTAGE REGULATORS:

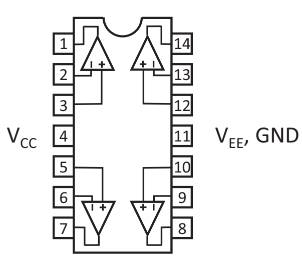


MC78Lxx
regulator
TO-92



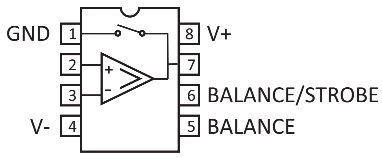
MC78xx
regulator
TO-220

OPERATIONAL AMPLIFIERS:

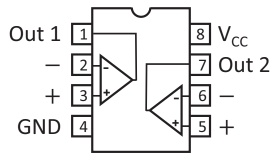


LM324
op-amp
14-pin DIP

COMPARATORS:

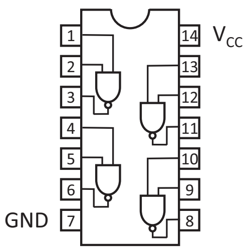


LM311
comparator
8-pin DIP

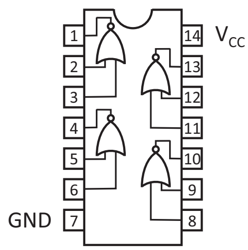


LM393
comparator
8-pin DIP

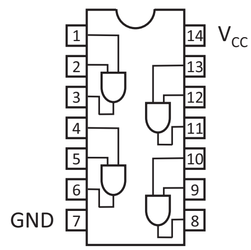
DIGITAL LOGIC GATES:



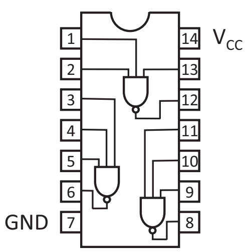
74LS00
4x2 NAND
14-pin DIP



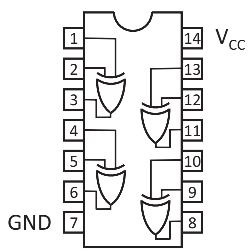
74LS02
4x2 NOR
14-pin DIP



74LS08
4x2 AND
14-pin DIP

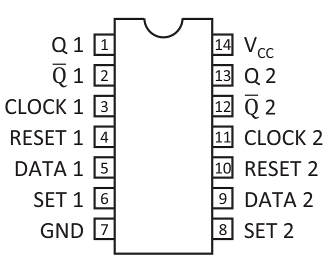


74LS10
3x3 NAND
14-pin DIP



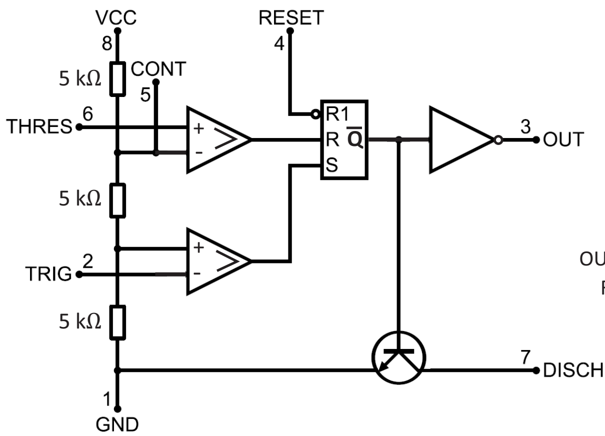
74LS86
4x2 XOR
14-pin DIP

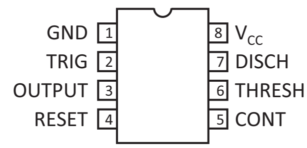
FLIP-FLOPS:



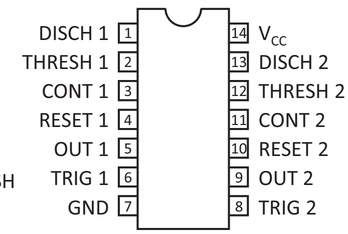
CD4013
dual flip-flop
14-pin DIP

TIMERS:



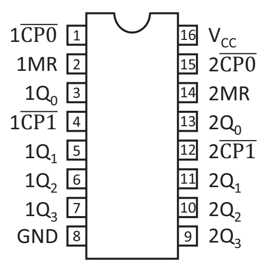


LM555
timer
8-pin DIP

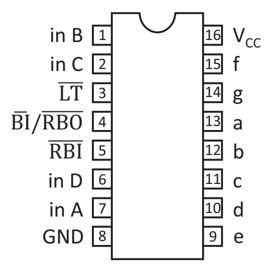


NE556
dual timer
14-pin DIP

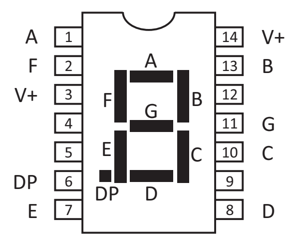
COUNTERS, DECODERS, DISPLAYS:



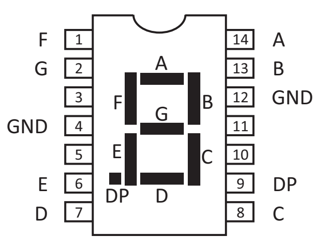
CD74HC390E
dual counter
16-pin DIP



74LS47
4-bit decoder
16-pin DIP



MAN72
7 seg display
14-pin DIP



MAN74
7 seg display
14-pin DIP

B A Short Summary on Handling Uncertainties

Uncertainties in measurements:
When you estimate the uncertainty of a measurement, the actual value should have a *reasonable* probability of lying within your stated range of *best guess* ± *uncertainty.* "Reasonable probability" usually means either 67% or 95%, called the *confidence level.* Sometimes you can only know the uncertainty from reading the manual for the measuring device. Other times you can use your common sense to determine whether you would be willing to bet a cup of coffee on the final result.

Propagating uncertainties:
When a calculated quantity depends on a measurement that has uncertainty, do the calculation using both your *best-guess* value and one of the two *worst-case* values, then take the difference of the results. **Example:** calculating circumference C from measured diameter of (50 ± 2) cm.

- Best guess: $C = \pi D = \pi(50\text{ cm}) = 157.1\text{ cm}$
- Worst case: $C = \pi D = \pi(52\text{ cm}) = \underline{163.4\text{ cm}}$
- Difference of: $6.3\text{ cm.} \Longrightarrow \boxed{C = (157 \pm 6)\text{ cm.}}$

As long as the uncertainties are smallish, you only need to calculate one worst case.

Two or more sources of uncertainty:
When a calculated quantity depends on two or more measurements that have uncertainty, start by finding the uncertainty in the calculated quantity from each ONE of the measurements separately, as above. Then combine those individual uncertainties "in quadrature," like legs of a right triangle. **Example:** calculating $W = F\Delta x$, where $F = (45 \pm 3)$ N and $\Delta x = (100 \pm 9)$ cm.

- Uncertainty in F only: $F = (45 \pm 3)$ N, $\Delta x = 100.000$ cm $\Longrightarrow W = (45 \pm 3)$ Joules.
- Uncertainty in Δx only: $F = 45.000$ N, $\Delta x = (100 \pm 9)$ cm $\Longrightarrow W = (45 \pm 4)$ Joules.
- Combine in quadrature: $\sqrt{(3\text{ J})^2 + (4\text{ J})^2} = 5\text{ J.} \Longrightarrow \boxed{W = (45 \pm 5)\text{ Joules.}}$

Three or more sources of uncertainty combine the same way: $\sqrt{(\ )^2 + (\ )^2 + ... + (\ )^2}$.

Correlated uncertainties:
Combining uncertainties in quadrature as above is what you do when the uncertainties are *uncorrelated*: each measurement could be either high or low, independent of the other one. But if the uncertainties are *correlated* (one measurement high means the other is high too), then calculate a single worst-case scenario with both measurements too high, or both too low. **Example:** You want the difference between two masses, $m_2 - m_1$, each measured on the same scale, which might be miscalibrated by up to 1%. Suppose $m_2 = (400 \pm 4)$ g, and $m_1 = (300 \pm 3)$ g.

- Best guess: $\Delta m = m_2 - m_1 = 400\text{ g} - 300\text{ g} = 100\text{ g}$
- Worst case: $\Delta m = m_2 - m_1 = 404\text{ g} - 303\text{ g} = \underline{101\text{ g}}$
- Difference of: $1\text{ g.} \Longrightarrow \boxed{\Delta m = (100 \pm 1)\text{ g.}}$

You can see right away that adding the uncertainties in quadrature would give (100 ± 5) g, which is crazy. (If they were measured by different scales, then you really could have $m_2 = 403$ g and $m_1 = 298$ g, so $\Delta m = (100 \pm 5)$ g would be reasonable, not crazy.) Also note that for adding the masses, a shortcut is to add the uncertainties directly, not in quadrature: $m_{total} = (700 \pm 7)$ g.

Disclaimer:
There are enough shortcuts, special techniques, definitions, and rigorous justifications to fill a book. If you want them, go find a book. This is just a short summary.