Laboratory Physics I
Electronic Lab

Experiments and Instructions

Guido Mueller, Andrew Rinzler

Textbook:

Basic Electronics for scientists and engineers

by Dennis L. Eggleston

Course web site: http://www.phys.ufl.edu/courses/phy4802L/spring15/

Simulation site: www.circuitlab.com

Other recommended texts:

The Art of Electronics by Horowitz and Hill

Student Manual for the Art of Electronics by Hays and Horowitz

Spring 2015, 4802L

Department of Physics

University of Florida

2015

Lab 1:

Equipment, tools and simple resistor circuits

The experiments in this Lab use mostly resistors and voltage sources to introduce the most fundamental laws (Kirchhoff's, Ohm's), circuit diagrams and simulation tools as well as the equipment which you will use throughout the course.

CIRCUIT DIAGRAMS AND SIMULATION TOOLS

We will make extensive use of circuit diagrams. You will need to learn how to read them and also how to draw them. You will also be asked to simulate your circuits. Many simulation tools for electrical circuits exist and can help you throughout this course. We will use www.circuitlab.com. It has a very simple user interface, provides simple simulation tools and should still be free for UF students if you sign up with your UF email address. Most of the circuit diagrams in this lab manual are drawn with Circuitlab and you can use it to draw your circuit diagrams for your lab reports. It also provides several simulation tools and you can specify parameters for many of the more complex components to study how they influence the response or behavior of your circuit. Set up your account at Circuitlab! **The general rule is that you**
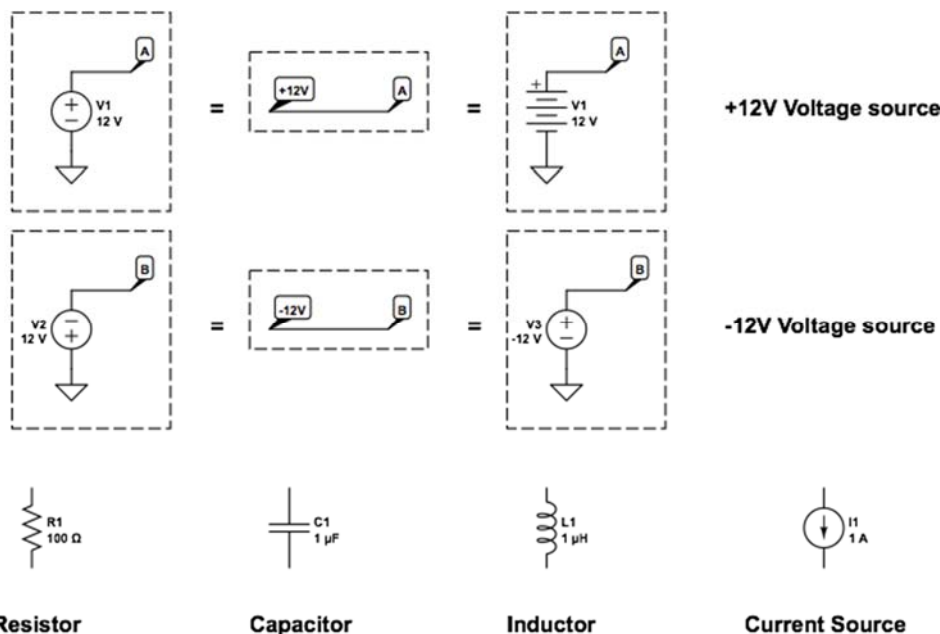

Figure 1: Different symbols are often used in circuit diagrams to describe similar things. Note here the different ways voltage sources are shown.

should simulate all circuits prior to, or latest in parallel with, doing the experiments.

Note that different books use slightly different symbols for some devices and this can be confusing. Even Circuitlab provides several options to include a voltage source in your experiment. Figure 1 shows three different ways to represent a +12V voltage source (top row) in a circuit diagram. The middle row shows different ways to represent a -12V source. The bottom

row shows the standard representations for a resistor (R), a capacitor (C), an inductor (L) and a current source.
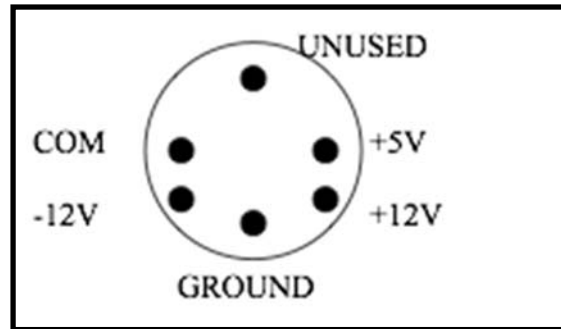
DC-POWER SUPPLY:

Figure 2: DC-Power supply connections

The standard power supply you will use for most of your experiments is the "power brick" sitting on your desk. This power supply is plugged into the connector on the breadboard platform. The wires coming out of the back end of the breadboard connector are shown in Figure 2.

GROUND is a direct connection to the Earth through the ground terminal on the three-prong wall plug. COM stands for Common Ground. It is the reference point from which the other voltages (+5V, ±12V) are measured. COM is a "floating" reference point. "Floating" means it is not connected to Earth ground.

All four terminals (+5V, ±12V, and COM) are isolated by a transformer, whose secondary windings power the circuitry that generates the output voltages. In fact, COM can be biased by a different power supply to any potential (up to 30V or so) and the three output voltages will likewise follow. The key thing to note is that the output voltages are relative to the floating COM terminal and not to the GND terminal. **Thus, for all the labs it is recommended to connect the COM terminal to the GND terminal to ensure that all supply voltages generated by the "power brick" are referenced to GND.**

Each voltage from the DC power brick (+5V, ±12V) is sent through a separate fuse on the breadboard mount. Make sure you only make connections to the supply voltage after the fuse. In the case of a wiring error, the fuse will blow, thus saving the power supply from possible damage. **We do not provide an unlimited supply of replacement fuses.** Please check your circuits very carefully before connecting to power!

TUNABLE VOLTAGE/CURRENT SOURCE:

The HP E3616A is a tunable voltage or current source (shown in Figure 3). There are two knobs: Voltage and Current. These are used to adjust the constant voltage output by the device or the constant current output by the device. There are little green LEDs (CV and CC) near the display which indicate if the power supply is voltage or current limited, if it operates in voltage or current mode.

For constant voltage mode, the current knob must be turned to a setting above any that need be supplied and then the voltage knob is set for the desired voltage. For constant current mode, the voltage knob must be turned to a setting above any that need be supplied and then the current knob is set for the desired current. The potential at the red terminal is always above that at the black terminal by the amount shown on the readout, but this potential difference is, like the power pack, floating. If the black terminal is wired to the green ground the red terminal will be at the potential given on the readout. If the red terminal is wired to the ground, the black terminal

Figure 3: Tunable voltage/current supply, also called the HP power supply.

will be at a potential equal to the negative of the readout.

The supplies may already have the black terminal grounded (with an easy to miss wire between them) for using the red terminal as a positive supply voltage.

MULTIMETER:

The multimeter allows you to measure current through it, voltages or potential differences between two points in the circuit, and the resistance of resistors (and some measure the capacitance of capacitors). We will use different multimeter models and won't to go into further detail here, but two things are worth pointing out:

A voltage measurement is always done between two points. The internal resistance of the voltmeter is high and will be parallel to the part of the circuit between those points.

A current measurement requires that all the current flow through the ammeter. The internal resistance of the ammeter is low and will be in series with the part of the circuit in which the current is measured.

The ideal ammeter acts like a low resistance wire in your circuit. Novices have a tendency to place it between points in a circuit which have a significant potential difference. This will cause a large current through the ammeter, which will blow the fuse. Some of the fuses (like the one in our Fluke multimeter) cost several dollars each, so the cost can add up quickly. Avoid this!
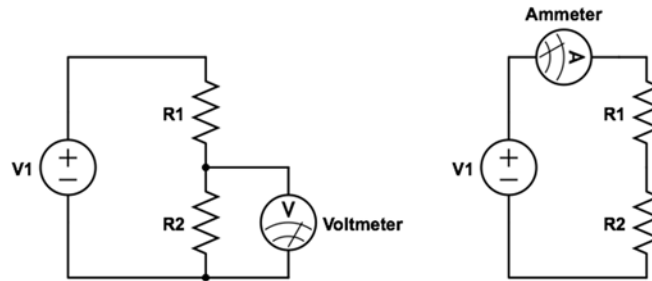
Figure 4: A Voltmeter has to be connected in parallel to the element(s) across which you want to measure the voltage difference. An Ammeter has to be connected in series with the element through which the current flows.

Example:

Assume that the voltage source in both circuits of Fig. 4 provides 10V. Now assume that resistor R1 has a resistance of 10Ω and R2 of 100Ω. The voltmeter in the left circuit will show a voltage of around 10V x 100Ω/(100Ω+10Ω) ~ 9V. The ammeter in the right circuit will show a current of I = 10V/110Ω ~ 90mA. The maximum rating of the Fluke multimeter in the low current setting is 400mA and will be fine.

Now assume you initially measured the current using the right circuit and now want to measure voltage in the left circuit. You disconnect the ammeter, and now connect it as shown in the left circuit, except that you forgot to use the voltmeter connections and settings and connect it as an ammeter across R2. The low resistance in the ammeter will allow the current to bypass R2, which will now be I = V/R1 = 1A, too high for the fuse and the fuse will need to be replaced.

This happens fairly often if you are not careful. This example also shows that connecting a measurement device will change your circuit and you will occasionally need to take this into account.

BREADBOARDS:

Breadboards are normally used to prototype and test electrical circuits before they are soldered on a circuit board. They are fairly standardized and here we want to discuss how to use them. Figure 5 shows a schematic circuit (left) consisting of four 10k resistors in series, connect to 6V at one end and ground at the other.
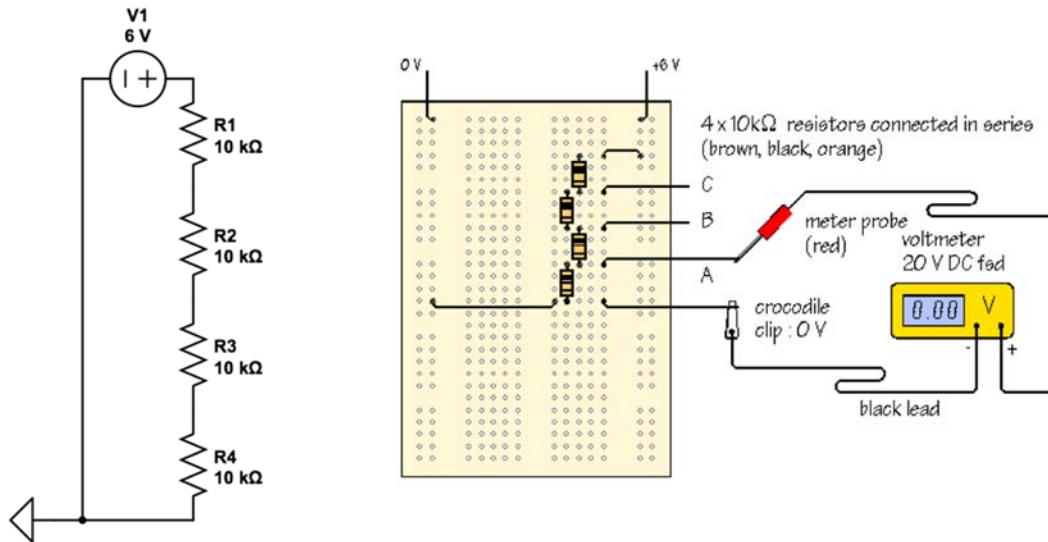
Figure 5: Four 10k resistors in series attached to a 6V voltage source. The left graph shows the circuit diagram, the right graph shows one way to set this up on the breadboard.

Figure 5 (right) shows how to assemble this circuit on a breadboard. The breadboard has four blocks which have two, five, five, and two vertical columns of holes. For the two column blocks all holes in each vertical column are connected to each other. For the five column blocks the 5 holes in each horizontal row are connected to each other.

The 0V (= GND for our purposes) connection of the top left corner ensures that all holes in the second column of the first block are at 0V. The holes in the first column of that block are not connected to 0V; they are not connected to anything and are 'floating'. Similarly, the +6V connection of the top right corner ensures that all holes in that column are at +6V. The short bridge (wire) in the upper right corner connects the 6V to the 4th horizontal row of the third block. This ensures that the five holes in that row in that block are also at +6V. One end of the top resistor is connected to that row, the other end to the 7th row. This is R1 in the circuit diagram. R2 connects row 7 with row 10, R3 row 10 with row 13, and R4 row 13 with row 16. Row 16 is also connected to 0V (= GND). This completes the circuit shown in the circuit diagram: four 10k resistors in series supplied with 6V.

A voltmeter could now be used to measure the voltage at points A, B, or C with respect to GND as shown in the figure on the right. Or measure the voltages across for example R3 by measuring the potential difference between points A and B.

The following two videos will provide more information about the breadboards and how to use them (watch these videos):

• Introduction to Breadboard, Part 1 (8 min): http://www.youtube.com/watch?v=oiqNaSPTI7w

• How Breadboards Work (10 min): http://www.youtube.com/watch?v=lqw6ask5HK0

Initial measurements:

VOLTMETER, POWER SUPPLY, AND BREADBOARD:

The first experiments are meant to help you to set up and become familiar with the breadboard, the DC power supply, and the multimeter.

The DC power supply will provide three different voltages measured relative to COM, the common reference 0-Volt level, and a ground connection. The three 'hot' lines are connected to three fuses on your bread board. These fuses protect the DC power supply from short circuits on your bread board which would draw too much current and could damage the power supply.

Each voltage is then connected to one of the upper horizontal rows on your bread board. We like to keep this consistent between the different boards to simplify trouble shooting and avoid unnecessary short circuits. The top row should be connected to +12V, the next row to -12V, the third row to +5V, and the last row to **COM and to GROUND**. We will usually not work with floating voltages but rather tie returns (COM) to ground. **Once these connections are made use the multimeter to measure the voltage of each of the three supply lines with respect to ground**

The measured voltages will not be exactly what you expect but might be up to 15% higher or lower. This is typical and you should always measure the voltages for each experiment and use the measured voltages to discuss your data.

Below these four rows is the area where you will set up your circuits. Just for practice, set up the four 10k resistors in series shown in Figure 5 and connect one end to 5V (instead of 6V used in Figure 5) and the other end to GND. Measure the voltage drop across each resistor and the voltage at points A, B, and C with respect to GND and compare with the expected values based on the simulation.

OSCILLOSCOPE:

The oscilloscope is the instrument of choice for measuring voltages that vary in time. It can display complex waveforms, permitting you to determine the amplitude, period, and shape visually. The original scopes (short for oscilloscopes) were analog scopes which used the measured voltage to deflect an electron beam vertically while a second internal voltage deflected it horizontally.

Modern scopes are virtually always digital scopes. They use an analog to digital converter (ADC) to measure the voltage at predefined time intervals, then digitize the signal and display it on a monitor as a function of time (or of a second signal when operated in H-V mode). These devices are now mini-computers and you will learn throughout the course how to use at least some of their functions. Connecting a scope to your circuit changes the circuit. The input of a scope acts as a low pass filter with a DC resistance of 1 to 10 MΩ depending on the type of scope. To minimize the impact, scope probes are often used which provide a higher impedance; we will discuss the concept of input and output impedance in more detail soon.

For the beginning, you need to know that you can adjust the vertical axis; this sets the minimum voltage the ADC can resolve and the max voltage the ADC can distinguish; any voltage above it

will saturate the ADC but not damage it unless you exceed the max voltage specifications. You can also adjust the horizontal axis or time base or how fast the scope digitizes the signal. The second important function is the trigger function. The scope does not know a priori which part of the signal the user wants to display. The trigger signal and trigger settings (post, pre, type of trigger) allow the user to select a specific part of the wave form to be displayed. As the trigger, you can use one of the signals themselves or a third signal which you don't need to display on the screen.

The inputs of the scope are BNC connectors and we can use for example BNC to alligator clamp adapters to connect the circuit to the scope. However, the scope itself has a finite input impedance which can distort your measurement. This is one reason why your scope comes with two probes which are essentially high impedance 10:1 voltage dividers (more on this later).

FUNCTION GENERATOR:

The function generator has two outputs. The signal output and the sync output. The main output is the signal output. The function generator allows you to generate different waveforms (sin, square, triangular) at different frequencies up to 30 MHz. You will use these signals as input signals in many of your experiments. You can add DC-offsets to the signals and modulate the amplitude or phase of the signals. Each voltage source like the function generator has a certain output impedance which is in series with your circuit. The output impedance of the function generator we use can be switched from 50Ω to low impedance (or high-Z load). For most of the experiments, you should use the high-Z load mode.

The sync or synchronization signal is a square wave signal of fixed amplitude which is in phase with the signal itself. This can be used to trigger (synchronize) the scope.

Function Generator and Scope:

Set the function generator up to produce a 10 kHz, 1V sin wave. Connect it to channel 1 of the scope with a BNC cable (disconnect the probes and put them aside for now). Set the vertical and horizontal axis of the scope such that you see a few complete oscillations filling the screen. Set the trigger to channel 1 and test the various trigger functions and settings.

Connect the sync output to channel 2 of the scope and observe the signal. Is it well synchronized with the sin wave from channel 1? Then connect the sync output to the trigger input and change the input of the trigger function to external trigger. Is the signal from channel one still well synchronized? It should be. This is what the sync out is meant to do.

Explore the function generator and the scope.

Now let's explore part of what scope probes do. First, connect a BNC to alligator clip adapter to the function generator and then measure the voltage across the alligator clip with a probe. The voltage will be a factor of 10 lower than expected unless you change the settings of the scope to 'compensate' for the factor of 10 voltage divider inside the probe. **Students tend to forget to set the scope correctly for the probe being used,** 10× for measurements with the probe, 1× when using just a BNC cable.

Basic concepts and resistor circuits

Chapter 1 in the textbook discusses Kirchhoff's current law (KCL), Kirchhoff's voltage law (KVL) and Ohm's law. The following experiments will explore circuits which should demonstrate the validity of these laws.

TEST OF KVL:

The circuits shown in Figure 6 allow you to test several of the laws discussed in Sections 1.1 and 1.2 in the textbook including KVL and that resistors add in series. First simulate both circuits to familiarize yourself with Circuitlab; let Circuitlab calculate the voltage at each node and the current through each resistor. Always use measured resistor values and source voltages for your final simulations.

Incorporate the simulation diagrams and results in your lab reports.
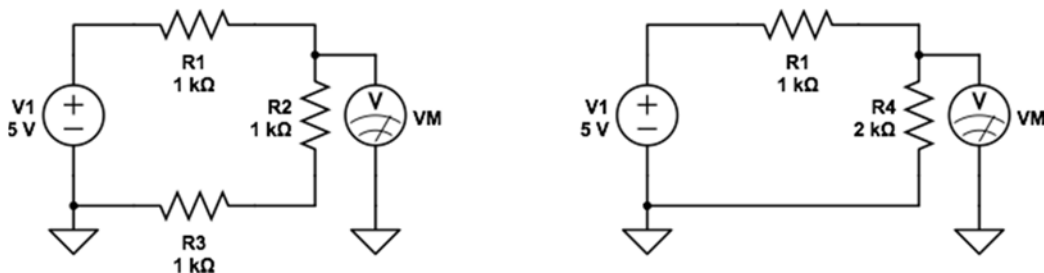
Figure 6: Two circuits which demonstrate that resistors add when used in series.

Test that resistors add in series (Eq. 1.9-1.11):

1. Set up the left circuit on the breadboard following the guidelines shown in Figure 5 (GND and power lines along the vertically connected holes and the circuit in the horizontally connected rows).

2. Measure the voltage supplied by the voltage source. To do this, set the multimeter to volts, connect the - (typically black) voltmeter terminal to ground (from the vertical GND line) and connect the + (typically red) voltmeter terminal to the 5V line (from the vertical 5V line).

3. Measure the voltage between R1 and R2 (left circuit) with respect to ground.

4. Set up the right circuit and measure the voltage across R4.

A few words about terminology:

- Voltages are electrical potential differences and are measured between two nodes in a circuit. One node might or might not be ground. For example if you want the voltage across a resistor you would place one probe on either side of the resistor.

- You will also see terminology like: measure the voltage at node A. What this means is to measure between node A and ground.

5. Compare the voltages.

Test of KVL (Eq. 1.2):

1. Set up the left circuit again and measure the voltage across each of the three resistors. To do this, as opposed to what you did above, you attach the + (typically red) voltmeter terminal to the high voltage side of the resistor and the - (typically black) voltmeter terminal to the low voltage side of the resistor, thus reading the voltage dropped across that resistor.

2. Add up these values and compare to the voltage provided by the power supply.

The multimeter allows you to measure potential differences within a circuit. The disadvantage is that the multimeter cannot display any dynamical (time-varying) signal.

The oscilloscope can measure dynamic or time-varying voltages but it cannot directly measure across a component because the outer cylinder of the BNC connector at the scope **is connected to ground**! Connecting ground somewhere inside your circuit in almost all cases represents a massive change in the circuit and in the best of all cases your data will just be wrong. In the worst case, you can **damage** the components in your circuit and **the equipment**.

Try three different combinations of resistors between 100Ω and $100k\Omega$ and confirm that resistors add in series and that KVL is valid.

Get a short BNC cable and a BNC to alligator clip adapter from the front. The red wire is connected to the central connector and the black to the outer cylinder in this adapter. Set up the left circuit of Figure 6 with the three 1k resistors again and measure the voltage **across** each

resistor by connecting the adapter with the BNC cable to the scope. Why is the voltage different from the value measured with the multimeter? (What was just said in the box above?)

TEST OF KCL:

Simulate the circuits shown in Figure 7 with Circuitlab; let the program calculate the voltage at all nodes and currents through all resistors. Do the results make sense?
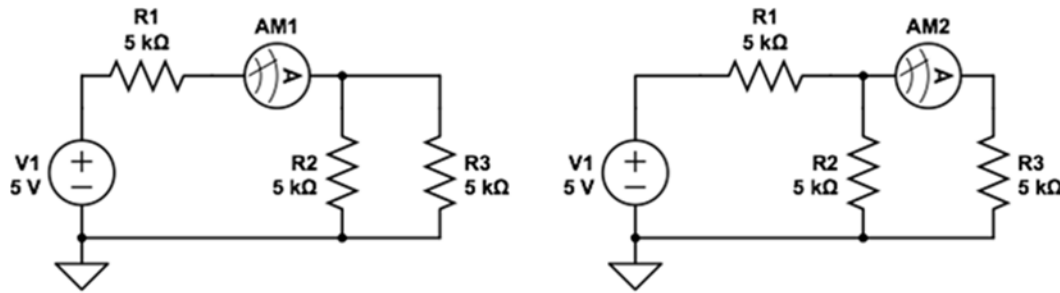

Figure 7: Two circuits which demonstrate KCL.

Set up the circuit on the breadboard and then

1. Measure the voltages across each resistor (Did you measure the true resistance of each resistor already?)

2. Measure the current through each resistor individually using the ammeter. Remember that the current has to go through the Ammeter and that the Ammeter has to be connected in series with the respective resistor. The figure on the left shows you how to measure the current through R1 (which then splits into two currents going through R2 and R3, respectively). The figure on the right shows you how to measure the current through R3 independent of the current through R2 (Don't forget to also measure the current through R2).

Replace the two 5k parallel resistors with one 10k and one 1k resistor and repeat the measurements. Use the measured voltages and resistances to calculate the currents through each resistor.

Do these values agree with the measured currents? Do the values confirm KCL? Does this all agree with the simulations?

TEST OF THEVENIN AND NORTON (SECTION 1.2.1.3):

Thevenin's theorem states that any combination of voltage and current sources and resistors (or generally impedances) that ends in two nodes can be replaced by an equivalent voltage source and series resistance (or internal resistance of the voltage source). These are called the Thevenin voltage and Thevenin resistance. We will use the circuit shown in Figure 8 to confirm this.

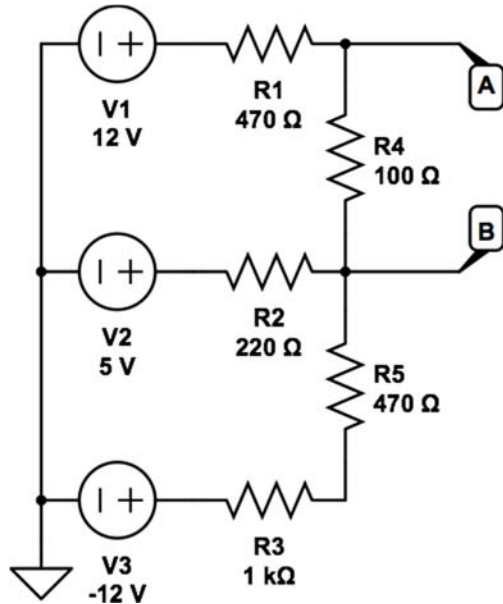**Be careful: Never connect the power lines directly to each other.**

Figure 8: Testing Thevenin and Norton.

First, simulate the circuit in Circuitlab. What will the potential difference between points A and B be?

If you directly connect points A and B *in the simulation* with an ammeter, you can measure the Norton current and calculate the Thevenin resistance from this.

Now, disconnect the power supply from the breadboard and set up the circuit. **Note that V3 is one way to show the -12V power line in a circuit diagram.** Check the circuit carefully before you reconnect the power supply to the breadboard. Then measure the voltage between points A and B with the multimeter. This is your measured Thevenin voltage.

Compare it with the calculated voltage. Do this first before you continue. If they don't agree within 10%, something might be wrong with your circuit and the next experiments could damage the ammeter!

The Thevenin resistance can be experimentally determined in two ways:

1. Measure the short circuit current:

   a. Disconnect the power supply first.

   b. Connect the ammeter between the two nodes, set it to high ampere first.

   c. Connect the power supply and use the scope and a probe to check that all three voltages are still there. If not, disconnect the power supply, check the fuses and get your instructor.

   d. If all this is OK, change the ammeter setting to 400mA and measure the Norton current.

   e. Calculate the Thevenin (or Norton) resistance from your measurement.

      f. Compare all your results with the results from the simulation.

2. Measure the resistance between the nodes:

      a. Disconnect the power supply and the cables from the power lines on the breadboard to your circuit.

      b. Replace each power supply with a wire (Connect the left sides of R1, R2, and R3 with wires).

      c. Measure the resistance between the two nodes A and B with the multimeter.

      d. Compare with the result above.

Test the effects of the output resistance of this voltage source.

Remove the short circuit wires and reconnect the power lines to your circuit. Place a 220Ω resistor as the load resistor between the nodes A and B and measure the voltage across this resistor.

- Is the result consistent with an ideal voltage source with an internal resistance of R_{TH}?

- Is the result consistent with an ideal current source with an internal resistance of R_{NOR}?

- What are the internal resistances of ideal voltage sources and of ideal current sources?

This is an example of input and output impedance. The self-made voltage source has an output impedance of R_{TH}. The input impedance of the load is in this case the 220Ω resistor. Input and output impedances are in general complex impedances which we will discuss in the next Lab.

APPLICATIONS:

Resistors are used in many electrical circuits for different purposes. One purpose is to limit a current that can go through a second device such as a light emitting diode (LED, we will discuss diodes in Lab 3). Another purpose is as a reference for resistive sensors. Many simple sensors change their electrical resistivity as a function of an external parameter like temperature, pressure or humidity. A photo resistor or photo cell is a device which changes its apparent resistivity as a function of the received light power. In the following experiment, we will use this to build a simple sensor.
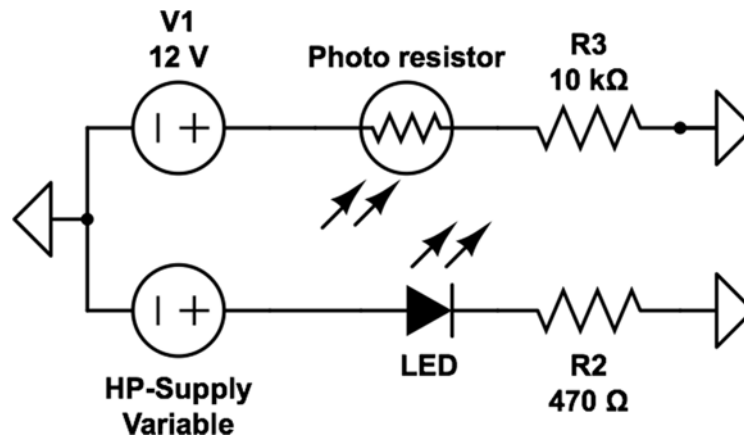
Figure 9: Resistors as voltage dividers and current limiters in a simple application.

Figure 9 shows a circuit which uses resistor R2 as a current limiter and resistor R3 as a reference in a voltage divider. Set up the circuit. Place the LED close to the photocell (NSL-4542 drawer) and use one of the toilet paper rolls with the polarizer sheet and your hand above both to block the room light on the photocell.

Note: Connect the negative pole (black socket) to GND (green socket) on the HP.

Task: Measure the photo current (current through the photo cell) as a function of the current through the LED.

1. Set the HP to 1V (voltage limited).

2. Measure the voltage across R2 with a multimeter and calculate the current through the LED from it.

    • Does the result agree with the current reported by the HP?

3. Measure the voltage across the reference resistor R3 using the scope.

Repeat the measurement for at least 6 different voltages ranging from 1V to 7V by changing the voltage setting on the HP.

LAB REPORTS:

You are now ready to write your first lab report.

In your lab report start by describing the purpose behind each experiment you did. Include a detailed schematic for each experiment, describe your experimental procedures with data on the component values and quantitative results, include simulation results, and finally give your conclusions based on the simulated and experimental results. Make the experiments the objects of the reports. I.e. while the purpose of the labs is for you to learn, the purpose you write should NOT be "The purpose of this experiment was to learn…" (that makes you the object). Rather you might write for one of the experiments above "The purpose of this experiment was to

confirm KVL for resistors wired in series…" Things that were done should be written about using the past tense. While it is not a hard rule, do try to avoid pronouns (I, we, …) i.e. rather than e.g. "We wired three 1kΩ resistors in series…" write "Three 1kΩ resistors were wired in series…" Finally, although the experiments should be a team effort with your lab partner, the lab write-ups are to be individual efforts (do however list your lab partner as co-worker).

Lab 2: R,C,L circuits and AC-signals

In Sections 2.2 and 2.3 of the textbook you read about capacitors and inductors and how they combine in parallel and series circuits to give an effective capacitance or inductance, respectively. To confirm these relations we could, for example, measure the amount of charge on systems of capacitors but our multimeter measures currents and voltages while the oscilloscope only measures voltages. Instead, we will measure the charging and discharging of capacitors and how they influence AC signals.

VOLTAGE DIVIDER:

Prior to working with capacitors, we will use our function generator and oscilloscope to explore a simple voltage divider made from two resistors as shown in Figure 10:
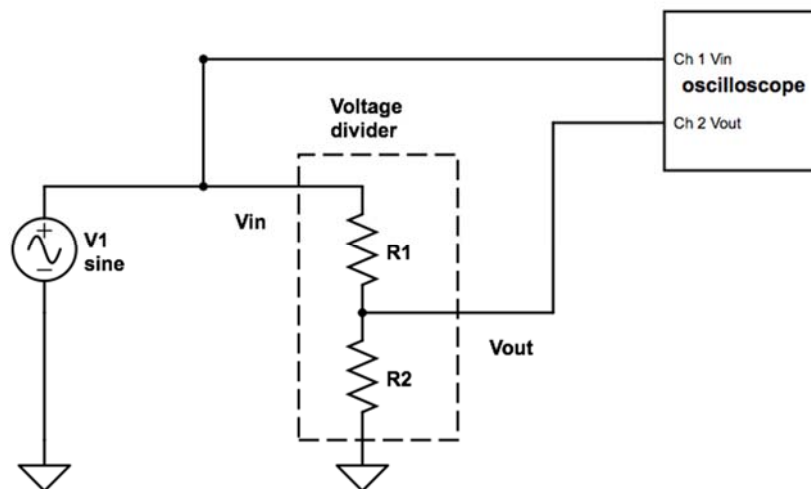
Figure 10: Two resistors in series form a voltage divider which divides the supply voltage independent of its frequency. R1 = R2 = 10kΩ.

Set up the circuit in Figure 10. Use a BNC-T to split the signal from the function generator and connect it to the circuit with a BNC-wire adapter and to channel one of the scope with a BNC cable simultaneously. Set the function generator to high-Z load. Use a probe to connect channel 2 of the scope to the node between the resistors.

The amplitude of an AC voltage is often given either as a straight amplitude as in A sin(2π ft) or as a peak to peak amplitude V_{pp} = 2A or, **in cases of sin waves**, as V_{rms} = A/√2. The last definition is useful for power calculations. The average electrical power which dissipates in a simple resistor is V^2_{rms}/R. Following this logic, what would be the relation between V_{rms} and A for square and for triangular waves?

Let the function generator produce a 10Hz sin-wave with a 1V amplitude (2Vpp). Connect the scope to the circuit as shown in the figure. Set the amplitude and time scale of the scope to values that a few periods of both signals are clearly visible on the scope. Measure the ratio of the two signals. Increase the frequency by factors of 10 and repeat the measurement until you reach 100 kHz.

Does the ratio of the two voltages depends on the frequency?

Replace R1 with a 100k resistor and repeat the measurement.

Switch R1 and R2 and repeat the measurement again. Do the results follow the voltage divider equation (eq. 1.21 in the text book)?

CAPACITORS:

Capacitors store and release charges. We will first study the charging and discharging processes (Section 2.4) in an RC circuit and then learn how capacitors combine to change the effective capacitance (discussed in Section 2.2).
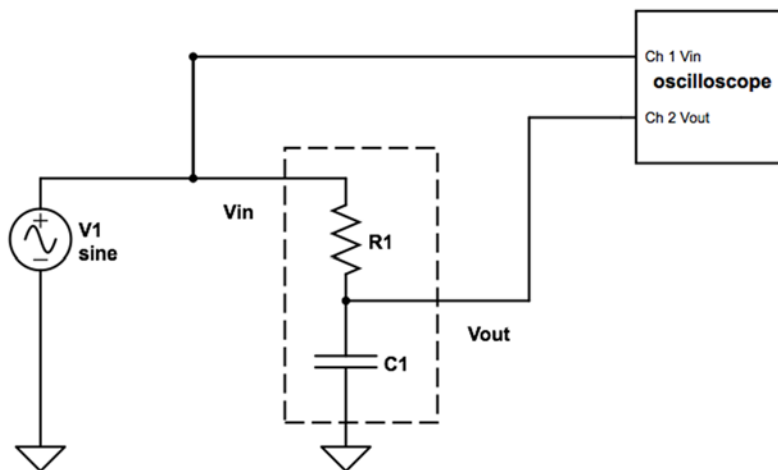

Figure 11: Charging and discharging of a capacitor. R1 = 10kΩ, C1 = 10nF.

Set up the RC circuit shown in the Figure 11 by replacing R2 of Figure 10 with C1. Let the function generator produce a square wave with an amplitude of 2Vpp and a period of 0.5s. Set up the time axis of the scope to 0.25s/div and the vertical axis to 0.5V/div and adjust the vertical offset such that 0V is in the center of the screen. Set the trigger to channel 1, AUTO, and adjust the trigger level such that the square wave forms a standing wave on the screen.

Observe the signal from channel 2 on the screen. The signal should look similar to the input signal as the time constant is small compared to the period of the input signal.

Now increase the signal frequency and decrease the time base on the scope (sec/div knob) to reproduce successively the three situations shown in Figure 2.7 in the book.

At what signal period T (inverse frequency) do you see a curve similar to the first output voltage (RC << T/2)?

Calculate the time constant for this RC circuit (textbook section 2.4.1). Measure the time constant for this circuit using the cursor measurement functions of the scope. For this measurement it might be best to decrease the time/div setting on the scope to look at just the charging or discharging part of the curve. Do not forget that you can change the trigger from triggering on either the rising or falling slope of a signal. How do the measured and calculated time constants compare? From this time constant you can determine the more precise capacitance of your nominally 10 nF capacitor. Hang onto and use this capacitor in the following sections.

At what signal period T (inverse frequency) do you see a curve similar to the second output voltage (RC ~ T/2) of textbook figure 2.7?

At what signal period T (inverse frequency) do you see a curve similar to the third output voltage (RC >> T/2) of textbook figure 2.7? Note: You might need to change the vertical axis as well to display the signal with sufficient amplitude on the screen. This circuit, in this regime is also called an integrator (see 2.5.3 in the book for details). Consider: what is the integral of a square wave? How does that compare to what you see on the scope here?

Add a second 10nF capacitor wired in parallel with the first capacitor. Measure the new time constant of this new circuit. Next move this second capacitor to wire it in series with the first capacitor. What is the time constant now? From these time constants, since you know the resistance you can deduce the equivalent capacitances of these capacitor combinations. Are these consistent with your expectations based on eq. 2.6 and 2.10 in the textbook?
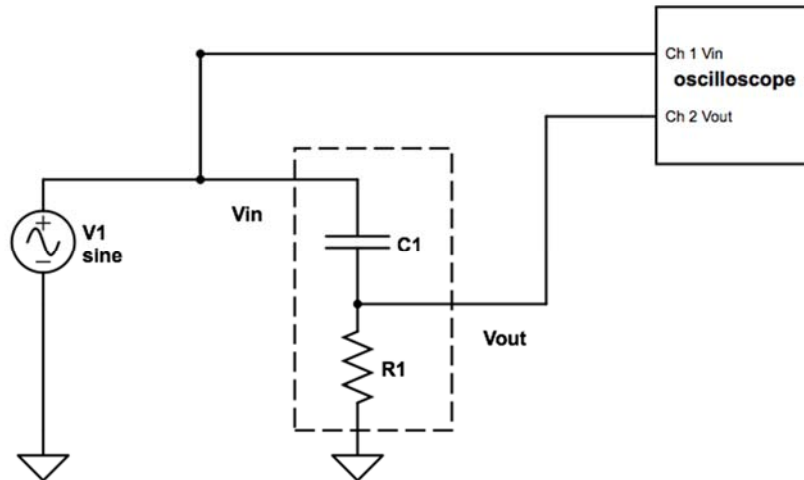
Figure 12: The voltage across R in an RC circuit. R1 = 10kΩ, C1 = 10nF

CR CIRCUIT:

Switch R1 and C1 (see Figure 12) and repeat the measurements (only the single capacitor experiments) to reproduce the waveforms shown in Fig. 2.9 in the textbook.

The voltage across the resistor should be the input voltage minus the voltage across the capacitor (KVL) measured earlier. Is this the case?

HIGH PASS FILTER:

This CR circuit is also called a high pass filter (HPF) which describes its response to sin-waves at different frequencies (Section 2.5 in the textbook). It attenuates the amplitude of low frequency input signals while allowing high frequencies to pass nearly unattenuated. To see this, set up the function generator to produce a sin wave with a 1V amplitude and a frequency of 1Hz. Measure the amplitude of the 1Hz oscillation across the resistor with the scope. Also measure the phase shift, φ, between this output and the input signal by measuring the time difference (Δt) between points of equal phase (e.g. adjacent zero crossings of the two sin waves). The phase shift is the fraction of a complete period, Δt/T, times 2π (to put it in radians) by which points of equal phase differ i.e.

$$\phi = 2\pi\frac{\Delta t}{T} = 2\pi f \Delta t$$

Now increase the frequency by factors of 10 and repeat the measurements (If the signal at 1Hz is too weak, start with 10Hz instead.).

Measure at a few additional frequencies in the region around the corner frequency $(1/2\pi RC)$ of the filter. Then plot the ratio of the amplitudes (Vout/Vin) versus the frequency in a log/log plot (Fig 2.13 shows the result in a linear/linear plot). Below this, on the same frequency scale plot the phase difference between the input and output signals. You should also simulate the circuit in CircuitLab and plot the outputs there.

These two plots together are called Bode plots and are used to analyze complex impedances, amplifier circuits, filters and many other devices or systems with complex transfer functions.

LOW PASS FILTER:

Rebuild the RC-circuit shown in Figure 11. This RC-circuit is called a low pass filter. Measure and simulate the frequency response (or transfer function) of the LPF the same way you did measure and simulate the frequency response of the HPF and also display the data in Bode plots.

LR-CIRCUITS

The voltage across an ideal resistor is proportional to the current. Its complex impedance is $Z_R = R$ which is frequency independent. The voltage across an ideal capacitor is proportional to the charge (or the integrated current) on it. Its complex impedance is $Z_C = 1/j2\pi fC$ (see page 47 in the text book). This is infinite at DC and becomes smaller and smaller at increasing frequencies.

The voltage across an ideal inductor depends on changes in currents through it. Its complex impedance (eq. 2.80) is $Z_L = j2\pi fL$. It is zero at DC and becomes larger and larger at high frequencies. Accordingly, LR circuits can also be used to build HPFs and LPFs (see Figures 2.20 and 2.21).

Measure the transfer function of the LR circuit shown in Figure 13. Is this is a high pass or a low
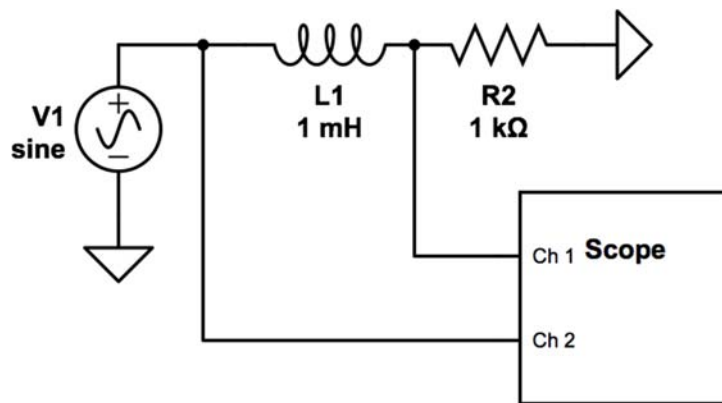
Figure 13: A filter using an inductor and a resistor in series.

pass filter?

LRC-CIRCUITS

We now consider resonant circuits as described in Section 2.6, figures 2.21 and 2.22.

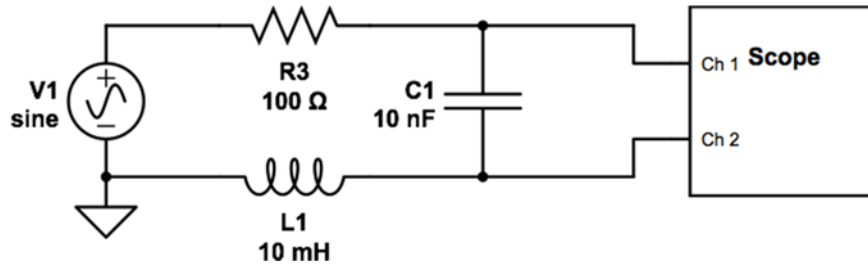Set up the circuit in Figure 14. (Simulation: Use the frequency domain tool.)

Figure 14: A resonant LRC circuit.

Connect the function generator to the LRC circuit and use probes to measure the voltages on both sides of the capacitor simultaneously. Do NOT ground any side of C1, rather use the math function of the scope display the voltage across the capacitor. Note that the inductor itself has a small resistance which you should measure with the multimeter and include in your analysis/simulation (CircuitLab lets you do this in the specification of the inductor's parameters). Measure the transfer function (amplitude and phase) of the circuit, find the resonance frequency, and the width of the resonance curve (similar to one shown in Fig. 2.22 in the book).

RINGING OF AN LRC CIRCUIT (SECTION 2.7 IN TEXT BOOK)

Figures 2.24-2.27 in the book show the responses of an LRC circuit when the voltage is turned on or off. We can use a square wave at low enough frequency (well below f_{Res} such as 10Hz) to emulate this. With R3 = 100Ω, the circuit should be a underdamped. See if you can observe the ringing shown in Figures 2.24 and 2.25.

Determine L1 from the measured resonance frequency and the accurate capacitance of C1 that you determined above; it should be within 15% of the expected 10mH. Calculate the critical resistance at which the circuit will be critically damped (sec 2.7.3) and replace R3 first with a resistor similar to the critical resistance and measure the nearly critically damped case (fig 2.27). Then replace this resistor with a resistor which is about a factor 3 larger and demonstrate the over damped case (fig 2.26).

FOURIER ANALYSIS (SECTION 2.8)

We can use the bandpass filter shown in figure 15 to measure the amplitude of the Fourier components of a square wave (eq. 2.125). The basic idea of this measurement is that the bandpass filter only lets a sin wave at a specific frequency through and blocks signals at all other

frequencies. So it can be used to extract the amplitude at a specific frequency component (with a certain frequency band) of an arbitrary signal (here the square wave).
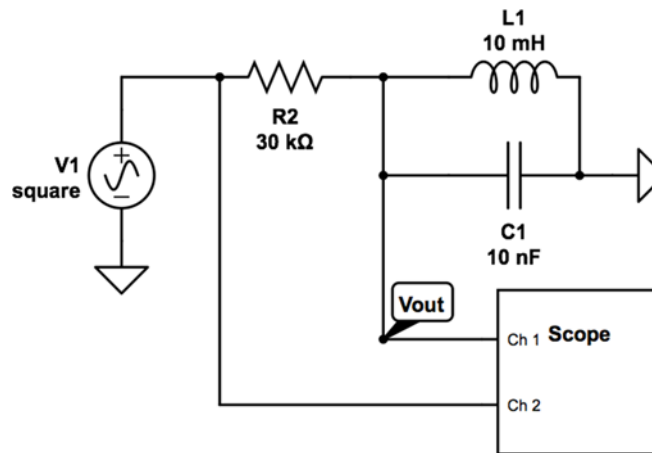

Figure 15: The bandpass filter.

Use the function generator to produce a square wave at the resonance frequency and measure the amplitude of the fundamental sin wave with the scope. Then decrease the frequency to $f_{Res}/3$, $f_{Res}/5$, $f_{Res}/7$, … and then measure the amplitude of the resonance frequency after the bandpass for each input frequency. These signals will not look like clean sin waves as the bandpass filter is fairly low Q[1] and also transmits a fraction of the other frequency components. Try to measure the amplitude at f_{Res} by 'eyeballing' what is coming from f_{Res} and what is coming from other frequency components.

This bandpass filter can also be used to determine π experimentally. Set the function generator to produce again a square wave at the resonance frequency. Measure the amplitude of the extracted sin wave at Vout. Then switch the function generator to produce a sin wave at the same frequency and same amplitude and measure that amplitude at Vout. Then calculate the value from the ratio between the amplitude of the injected square wave and the amplitude of the sin wave at f_{Res}. Compare the amplitudes to the expected amplitudes based on equation 2.125.

---

[1] See http://en.wikipedia.org/wiki/Q_factor for a discussion of Q-factors in resonant circuits.

Lab 3: Diodes

The world of electronics would be stuck in the 19th century if we only had access to simple linear impedances (R, L, C). Semiconductor materials allow the fabrication of additional electronic components which are used in most modern electronic devices. We already used a photocell and a LED in an earlier experiment (both relying on semiconductors to function) without describing their electronic characteristics. In this Lab will explore the behavior of the simple diode to lay the foundation for the even more important transistors and integrated circuits to come later.

A diode is characterized by its I-V (current-voltage) curve. It tells you how much current flows through a diode for a given voltage across it. But in contrast to linear components, the dependency is not linear.

DIODE I-V CURVE

The I-V curve of a diode can be measured with the setup shown in Figure 16:
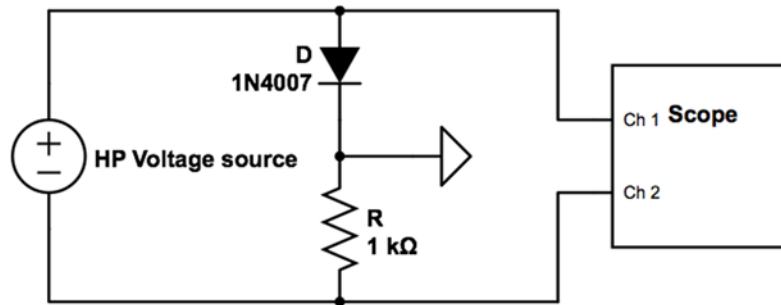
Figure 16: This circuit can be used to measure the I-V curve of the diode

The above setup works the following way. The HP supply provides a potential difference across the diode and the resistor. The ground connection between them does not influence this difference and is only there to provide a reference for the scope to measure the voltage across D and R. The voltage across R can be used to calculate the current which goes through R and through D. The voltage across D together with the current measurement allows you to measure the I-V curve.

Important: In the circuit of Figure 16 the HP voltage source should not be grounded on either side (neither the red nor the black output should be connected to the green output at the HP. Check that!)

Turn the voltage of the HP to zero and set its current limit to 100 mA. Increase the voltage of the HP source from 0 to 2V in steps of 50mV and measure the voltage across the diode and across the resistor with the scope. You can determine the current I through the circuit (and thus through the diode) from the voltage drop measured across the resistor and its resistance. Plot the results first as a function of the HP voltage and then in an I-V curve that plots the current versus the voltage that develops in this circuit across the diode only, V_d (similar to Fig. 3.12 in the text book). To obtain the latter use KVL. Determine the factor e/nkT (eq. 3.10) from the data. To do this use only the high forward bias part of the I-V_d curve from say 0.5 V to 2 V for which the 1 in the parenthesis of textbook equation 3.10 can be ignored to give that:

$$I = I_0 \exp\left(\frac{eV_d}{nkT}\right)$$

Plot $\ln(I)$ versus V_d and fit a straight line to it. Note the factor n in the exponent. This factor is a correction factor to the simplified theory presented in the textbook called the ideality factor (sometimes called the emission coefficient). Determine n from your data assuming T = 300K; the value is typically between 1 and 2.

Next change the polarity of the HP source and increase the voltage either until you see the breakdown voltage (Fig 3.14) or until the maximum voltage. Some diodes might show the breakdown, some might not.

A way to display the I-V curve on the scope directly is to replace the HP with a transformer.

First connect the transformer to a power outlet. The transformer is a center tapped transformer[2]. Measure the AC voltages across the full secondary coil first with the multimeter and then with the probes from the scope. Repeat the measurement across half of the coil (between the center connection and one of the others). If we assume that the input voltage is 120V_{AC} (US rms voltage) what is the ratio n_2/n_1 of the transformer? In the following experiments we will only use the full secondary coil and not the center tap.

Replace the HP in Figure 16 by the output terminals of the transformer. Set the scope to X-Y mode where the position of the signal on the horizontal axis is proportional to the diode voltage (Ch1) and the vertical position to the current (Ch2). Try it out and show it to the instructor.

> For many applications, a very simplified model is used to represent a diode (see textbook Figure 3.18) where the exponential increase of the current is replaced by a vertical line resembling a switch which requires a voltage drop of typically 0.6V (for silicon diodes). The current is then limited by the load resistance (textbook eq. 3.14).

---

[2] http://en.wikipedia.org/wiki/Center_tap

SIMPLE DIODE APPLICATIONS

\Diodes can be used to reduce or even limit the voltage at a given node inside a circuit. Figure 17 shows four different examples.

Use three silicon diodes (1N914) and a 1k resistor in series to set up the voltage dropper shown in Figure 17. Start with $V_0 = 2V$ and increase to 4V in steps of 0.2V. At each step, measure the potential above ground between each diode pair and across the resistor. Do the results agree with your expectation? Do the results justify the simple 0.6V voltage drop model?

Diodes can be used to protect other parts of a circuit from higher and potentially damaging voltages. Set up the Diode Clamp I circuit shown in Figure 17. Let the function generator generate a 1kHz triangular wave with an amplitude of 3V. Increase the HP voltage from 0 to 3V in steps of 0.2V and observe V_{out} on the scope. You should find the clamped voltage is not quite flat. Why is this? Simulate the circuit and zoom in on the flat region and compare with your experimental results. The disadvantage of this circuit is that we use a very expensive HP power supply to set the clamp voltage to the appropriate level.
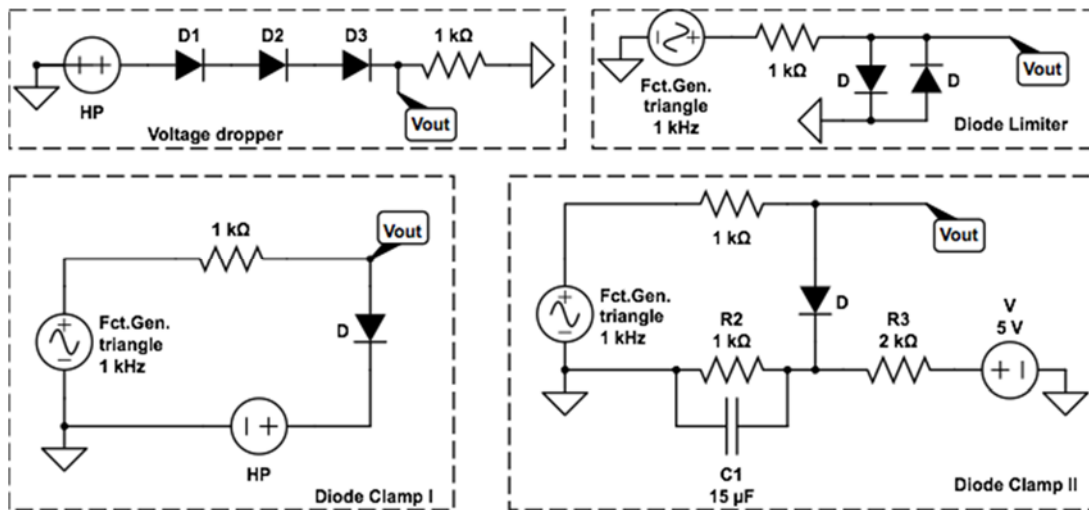
Figure 17: The voltage dropper reduces the voltage across the 1k resistor. The Diode clamp I limits Vout. Diode clamp II uses a voltage divider to limit Vout.

The Diode Clamp II circuit replaces the HP with a DC voltage source and a voltage divider. Set up this circuit first without the capacitor (C1) to provide the clamping voltage. Drive the circuit again with a 3V triangular wave, and examine the peak of the output. Why is it so rounded?

The problem with the voltage divider used in the Diode Clamp II circuit is that it is a high output impedance source. The current through the diode also has to flow either through R2 or R3. This increases the voltage between R2 and R3 and then the voltage V_{out}. This is one example why circuits which have to provide a fixed voltage at a given point have to have low impedance outputs. The cap provides a low impedance path to ground for the AC signal, making the output impedance of the voltage source low for the AC signal.

As a remedy, add the 15 µF capacitor in parallel with the 1k resistor. Why does this help?

Simulate the circuit with and w/o the cap. Notice the exponential charging of the cap in the simulation. What is the time constant of the charging.

Diode limiter:

The diode limiter in Figure 17 limits the absolute output voltage. Set up the circuit and drive the diode limiter with a triangular wave of varying amplitudes, from 1 V to 5 V, and measure Vout. Discuss the result.

RECTIFIER CIRCUITS (SECTION 3.2.3)

Diodes allow current flow in one direction only. This can be used to rectify AC signals, i.e. to turn AC signals into DC signals or AC power supplies into DC power supplies. For these experiments, we will use a transformer (see section 2.9 and figure 2.29 and 2.31).

Half-wave rectifier:

Construct the half-wave rectifier circuit shown in Figure 18. Use the two scope probes to
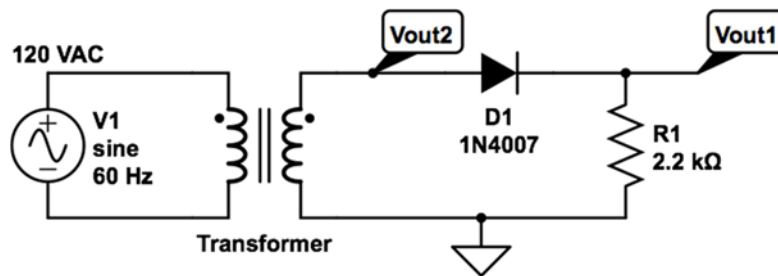
Figure 18: A half-wave rectifier.

measure V_{out1} and V_{out2}. Simulate the circuit using the time domain simulation tool.

What is the voltage difference between the two measured peak voltages Vout1 and Vout2? Can you explain this difference?

Obviously, the output of the half-wave rectifier is not very useful. Still, with the addition of a low pass filter, the voltage could be turned into something that resembles a reasonable DC voltage. But we always lose half of the sin wave. The center tapped full-wave rectifier (textbook Fig 3.29) improves the performance but still only uses half of the transformer at a time.
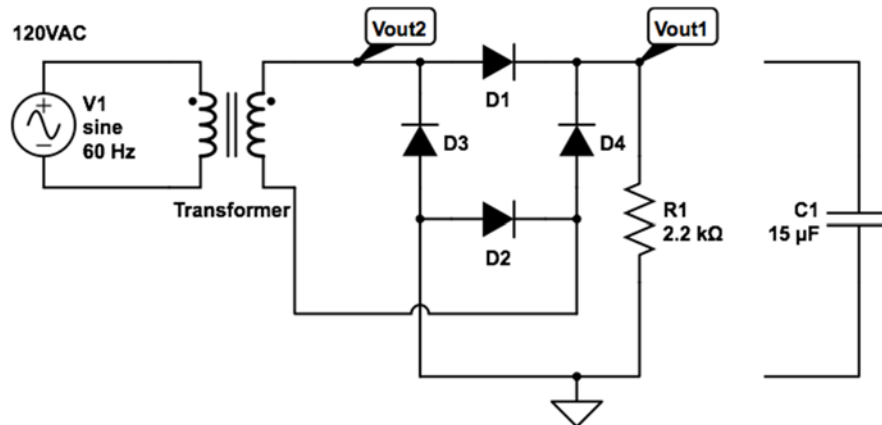
Full-wave rectifier:

Figure 19: A full-wave rectifier.

Use four 1N914 diodes (or similar) to set up the circuit in Figure 19. Be very careful with the polarity of the diodes (Why?).

For the initial measurements, do not connect C1 to the circuit and measure the output voltage across R1 and the voltage V_{out2} with the scope. Be careful how you connect ground to the circuit. The only ground here should be the scope's grounded return clip. Be certain it connects to the diode bridge as shown in Fig. 19 before plugging in the transformer or you are likely to blow diodes (probably in pairs).

Look at and save the output waveform. Look at the output waveform near zero volts. Why are there flat regions? Explain V_{out2} and V_{out1}.

Simulate the circuit using the time domain tool. Choose a fairly high time resolution to resolve the flat regions near zero. What causes these flat regions?

RIPPLE (SECTION 3.2.4)

While the above circuit generates a potential that is always positive from the input AC voltage, this is not yet a nice DC voltage to work with. To improve this, add the 15µF capacitor in parallel to the output resistance and measure the voltage now. Does the output make sense? Calculate what the ripple amplitude should be, and then measure it. Add the capacitor in your simulation. Do the experiment, the simulation, and the analytical values agree with each other?

Now replace the 15⌠F capacitor with a 500 ⌠F capacitor across the output (experiment and simulation) and see if the ripple is reduced to what you predict. This circuit could now act as a reasonable voltage source for low current loads. To make a power supply of higher current capacity, you'd need to use more robust diodes (1N4002) and a larger capacitor.

Section 3.2.4 describes different types of power filters and their limitations. The key features are the remaining *ripple* or *ripple factor* and the *regulation* which describes the ability of a voltage source to maintain its output independent of the supplied current. See Table 3.1 for a summary.

ZENER DIODES (SECTION 3.2.5)

So far, we only used the properties of diodes in the forward bias mode. The breakdown of a diode under reverse bias is characterized by an often even steeper increase in current as a function of the applied voltage. Zener diodes are special diodes which are designed to operate in reverse bias breakdown mode (something not generally healthy for normal diodes). This property provides another method to regulate the output voltage of a DC power supply. Figure 20 shows a Zener limiter which limits the voltage across R2. Set this circuit up, drive with $V_{PP} = 20V$ triangular wave and monitor V_{out}. Describe how this circuit works.
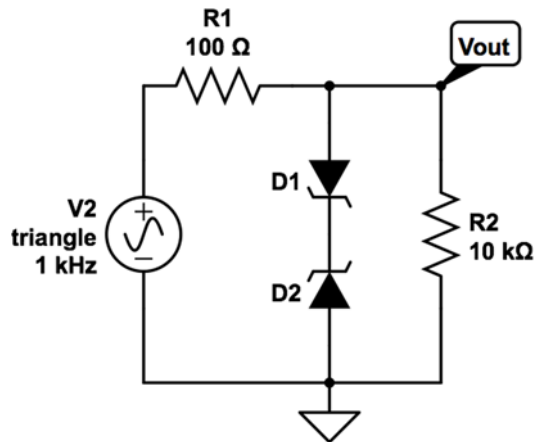
Figure 20: A Zener limiter.

Lab 4: Bipolar junction transistors

Bipolar junction transistors consist of three differently doped semiconductors in series: A npn transistor uses a heavily n-doped emitter, a lightly p-doped base, and a lightly n-doped collector on the other side of the base. A pnp transistor uses p-doped materials for emitter and collector and a n-doped base in the middle. We will mostly work with npn transistors but pnp transistors behave similarly.

BASIC PROPERTIES (SEE SECTION 4.2):

Base-emitter and base-collector I-V curves:

Set up the left circuit in Figure 21 and measure the I-V curve of the base-emitter pn-junction inside the transistor (see instructions around Figure 16 in Lab 3). Then measure the I-V curve of the base-collector pn-junction (right circuit in Figure 21). Compare the two results with each
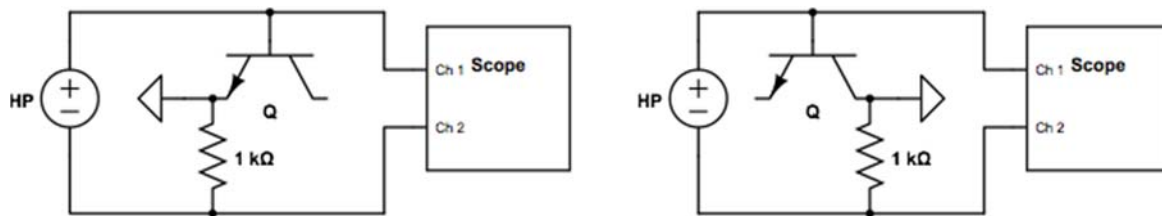
Figure 21: I-V Curve measurements of the pn-junctions in a npn 2N3904 transistor.

other and with the simple diode measurement (Fig. 16).

Basic operation

Several different voltages and currents determine the operating point of the transistor. The most commonly used ones are the base-emitter voltage V_{BE}, the base-emitter current I_{BE}, the collector

emitter current I_{CE}, and the collector-emitter voltage V_{CE}. The collector-base voltage V_{CB} and
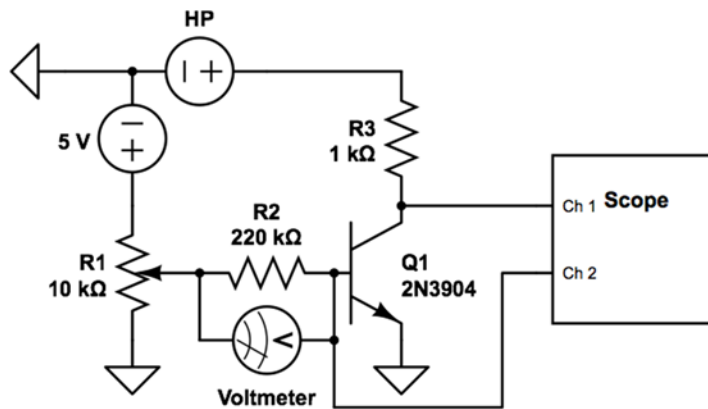
Figure 22: Measuring the I-V curve of a transistor

collector-base current I_{CB} can be calculated from the first four using KVL and KCL.

Typical transistor analyses treat the base-emitter junction as a simple forward biased diode with V_{BE} around 0.6V. This leaves three critical parameters, I_{BE}, V_{CE} and I_{CE} which define a family of I-V curves. These can be used to characterize the transistor (Textbook figure 4.4). You will measure these curves for one transistor using the setup shown in Figure 22.

The base voltage will be provided by the 5V line and the R1 potentiometer. Determine the base current from the measured voltage across R2. Measure the base emitter voltage V_{BE} with a scope probe. The HP power supply is used to supply V_{CE}. The current display will show the current going into the collector. However, we can also determine the current by measuring the voltage on the other side of R3 with a scope probe. Do this to verify the reading from the HP power supply.

Tune the potentiometer R1 to allow 1µA base current and then increase V_{CE} from 0 to 1V in steps of 0.1V and from 1V to 5V in steps of 0.5V and measure I_C and V_{BE} for each setting. Then increase the base current in 2µA steps up to 7µA and repeat the measurements. Plot the I-V curves (Textbook figure 4.4).

Also plot the collector current as a function of the base current for V_C = 1V, 3V, 5V.

What is the gain, ß, of the transistor (Equation 4.5 in textbook)?

APPLICATIONS

The textbook takes a very general but less intuitive approach to transistor applications. In the lab manual, we follow the approach in Chapter 2 of the *Student manual for the art of electronics by Hayes and Horowitz.* (One copy is in the e-lab room.)

We start with a simple model where

- $V_{BE} = 0.6V$

- $I_E = I_C + I_B = (\beta+1)I_B \sim \beta I_B = I_C = I_E$, as $\beta \gg 1$

This assumes that the circuit is set up such that these conditions can be met. This is not obvious; for example the supply voltages and resistors connected to the collector and the emitter have to allow a large enough current flow. The model you might want to keep in mind is that of a valve where I_B is a control signal which tells the valve how far it should open or what the maximal allowed current can be. But if the flow from the collector is limited by a large collector resistor, the amplification will be much lower. The circuit also has to be able to maintain $V_{BE} = 0.6V$ and $V_B-V_C < 0.6V$ to keep the np junction between the base and collector 'closed'.

Emitter follower:

Set up the circuit in the dashed box in Figure 23. This circuit shows one simple way to add a DC voltage to an AC signal. Seen from the HP, R2 and C1 act as a voltage divider with an infinite impedance (C1) as the second element. For the function generator, C1 and R2 are a high pass filter with a low enough corner frequency (Calculate it for your lab report) for the 10kHz signals we use in later experiments (This little circuit is necessary because the DC-offset in the function generator is limited to twice the AC amplitude. This is a typical limitation for digital function generators and has to do with the bit resolution). Tune the HP voltage to 1V output voltage (max current setting) and set the function generator to produce a 100Hz triangular wave with a 100mV amplitude (no DC offset) and measure the voltage between C1 and R2 with a scope probe. Then increase the frequency to 1kHz and 10kHz.
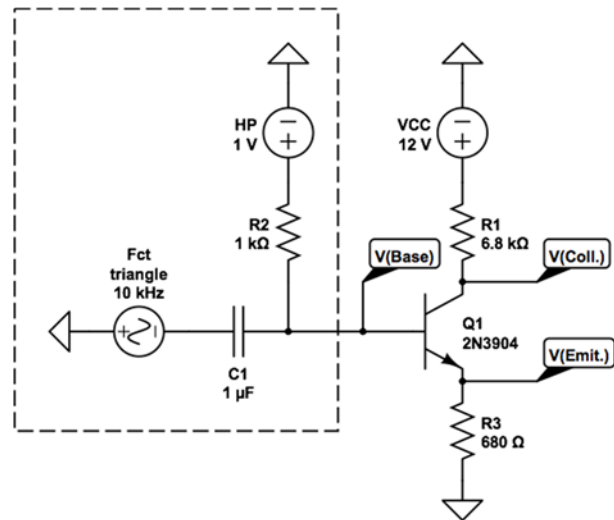
How and why does the shape of the signal change?

Figure 23: Generic Transistor circuit

Assume that the output resistances of the HP and of the function generator are negligible. Calculate the Thevenin resistances for the DC and for the 10kHz AC signal of the dashed box voltage source.

Now add the right part of figure 23 to the circuit. Turn the HP power supply to 0V and connect the node between R2 and C1 to the base of the transistor. Use scope probes to measure the base and the emitter voltages. AC-couple the scope and increase or decrease the sensitivity (reduce/increase Volts/div) of the scope as needed. Use the multimeter to measure the DC offset potentials at the base and at the emitter. Then increase the HP voltage in 0.2V steps and repeat the measurements until you reach 2V.

At what HP voltage level do you start to see parts of the AC voltage at the emitter?

At 0.8V and above, the emitter voltage should follow the base voltage with a fixed offset of ~0.6V. This emitter follower circuit forces the emitter voltage to follow the base voltage (DC and AC). The voltage gain of the circuit is one.

The usefulness of this circuit becomes clear when you think about the current amplification in the context of output and input impedances. The simplest way to see this experimentally is to remove C1 and the function generator, bypass R1 with a wire and replace R2 with a 6.8kΩ resistor. Measure the emitter voltage as a function of the HP voltage from 0 to 1V in steps of 0.2V and from 1 to 15V in 2V steps.

In this circuit R2 now acts as the Thevenin resistance or output resistance of our voltage source and R3 is the load resistance. Without the transistor only 1/11 of the output voltage would drop across R3. With the transistor nearly the entire voltage minus 0.6V drops over R3 except for very low and high base voltages. The transistor has changed the apparent impedances of the load seen by the source and of the source seen by the load. Seen from the load, the source acts as a voltage source with an output impedance which is a factor ß smaller than the output impedance without the transistor follower circuit (here 6.8kΩ -> 68Ω for ß=100). Seen from the source, the load looks like a factor ß larger output impedance (680Ω -> 68kΩ for ß=100).

Amplifier:

The current gain can also be used to amplify variations in the base voltage (= a signal added to the base voltage). The transistor will adjust the current flow into the collector to ensure that the emitter voltage follows the base voltage (with a 0.6V difference). This current has to flow

$$V_E = V_B - 0.6V \quad , \quad I_E = \beta I_B \quad , \quad V_{CC} - V_C = R_C I_C$$

$$\text{and } I_C = I_E \quad \Rightarrow \quad \Delta v_C = -\frac{R_C}{R_E}\Delta v_E = -\frac{R_C}{R_E}\Delta v_B$$
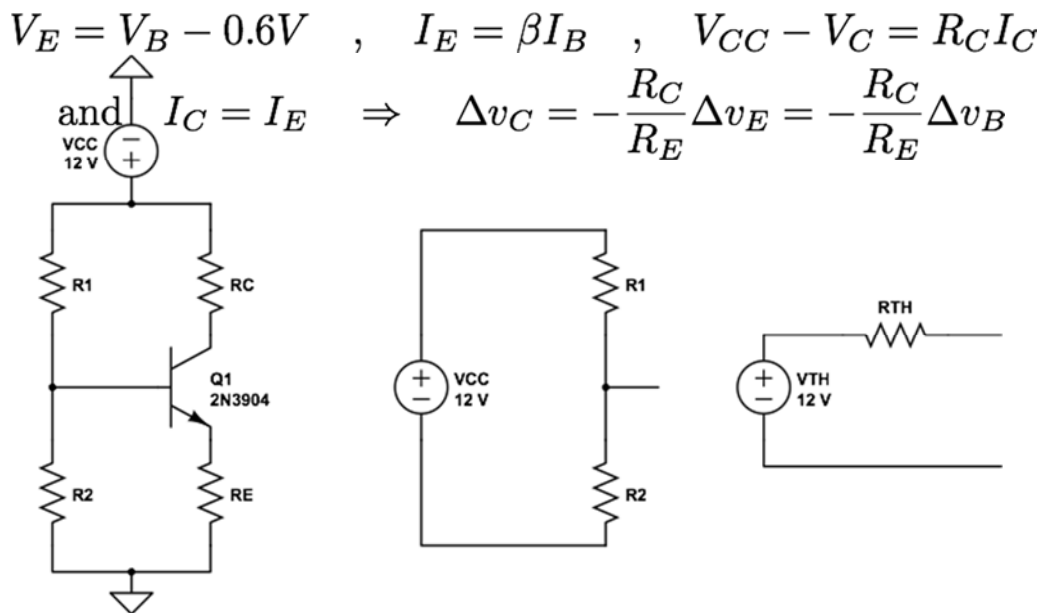
Figure 24: The general DC-bias circuit.

through the collector resistor R1 and the collector voltage will adjust (if possible) to enable this:

Thus any signal added to the base voltage will be inverted and amplified by the transistor with the output taken at the collector.

Set up the circuit in Figure 23 again, set the HP to 1V and let the function generator produce a sin wave with 0.1V amplitude at 10 kHz. Use a BNC-T to also send the signal to the trigger input of the scope to synchronize the scope and the function generator. You could also use the sync

output from the scope as well but the 5V square wave will cross talk into your small signals; try it!. Use the scope probes to measure the voltage at the base and at the collector. Switch to AC coupling and adjust the vertical resolution of each channel to measure the amplitude of the AC signals accurately. What is the voltage gain of the circuit?

Now increase the amplitude of the AC signal in 100mV steps until around 500mV where you should start to see distortions in the signal (One way to observe this is to adjust the vertical scales on the scope such that both curves ideally lay on top of each other. At what amplitude do you see these distortions and why? Adjust the HP voltage until the signal is again free of any obvious distortions and continue to increase the signal amplitude until you see distortions at high and low output voltages. You should now be able to use the HP voltage to either improve the quality of the signal at high or at low output voltages but not over the entire range.

Discuss what limits the linear range (no distortions) of this common emitter amplifier circuit.

High gain amplifier:

Transistors are often used to amplify weak AC signals by large factors. One way to try to increase the gain is to reduce R_E. So we could replace R_E with the smallest resistor we have, a wire, and use it to connect the emitter directly to ground. The real resistance is then limited by the internal resistance in the emitter portion of the transistor which is typically 25Ω [1mA/I_C] (it actually depends on the collector current and is approximately 25Ω at 1mA, 12.5Ω at 2mA, 50Ω at 0.5mA, see Hayes Horowitz page 100-104). On the collector side we could try to increase the collector resistor to larger and larger values. This has multiple limitations. First, the output impedance of this circuit is equal to the collector resistor and we know by now that large output impedances should be avoided. Second, the collector voltage V_C could fall below ~V_E+0.2V which is a necessary condition for the transistor to work properly as an amplifier.

A way to lower the impedance at the emitter without reducing R_E, is to lower it for for the signal frequency alone and not at DC. Place a 1µF capacitor parallel to R3 to produce a low impedance for our AC signal, here at 10 kHz. The gain will now be so high that we will need to reduce the amplitude of the applied signal (the function generator cannot produce signals < 100mV) by placing a 20kΩ resistor between the function generator and C1. This resistor forms a voltage divider with R2 and reduces the input voltage by a factor 21. Set the amplitude at the function generator to 100mV and set the HP initially to 1V. Connect one scope probe to the base and one to the collector of the transistor. AC-couple the signal on the scope to be able to see it (removing the DC on which the signal rides). Use the sync output of the function generator to trigger the scope and average the signal to reduce the noise as needed.

Now tune the HP signal to optimize (maximize) the signal output. Measure the DC voltages at the base, collector, and emitter with a voltmeter for this setting.

At what voltage do you see the strongest signal? Explain in your lab report.

Measure the gain of the circuit. Is this the expected gain?

DC-Bias (Section 4.4.1 in textbook):

The experiments above show that the transistor is a versatile device which allows amplification of voltages and currents. However, it always requires a DC base voltage and some tuning of the collector and emitter impedances to work properly for the applied signals and envisioned gains.

So far, we have used the HP voltage source to provide and adjust the DC base voltage. This is certainly not an option in electronic circuits. More generally, there is a fixed DC voltage Vcc already available which the DC-bias circuits use to provide the required base voltage.

The basic idea is a simple voltage divider formed by R1 and R2 connected to the 12V voltage source as shown on the left side of Figure 24. Looking backwards from the base this can be modeled as a Thevenin voltage with a Thevenin resistor which is then applied to the base of the transistor:

$$V_{TH} = \frac{R2}{R1 + R2} VCC \qquad R_{TH} = \frac{R1R2}{R1 + R2}$$

The real base voltage also depends on the effective load resistance which is formed by the base emitter junction and RE. Carefully selecting R1, R2, RE and RC ensures that $V_{BE} \sim 0.6V$ and $V_{CE} > 0.2V$. These conditions have to be fulfilled for the transistor to operate either in the saturation region or in the linear active region. Note that the emitter follower and the common emitter amplifier both work best in the linear active region where V_{CE} exceeds about 1V (see Figure 4.4 in text book).

The idea of the bias circuit and the amplifier is that the DC parameters determine the operating point (also called quiescent point) of the transistor in the I-V diagram.

Experiments with the common emitter amplifier:

Task: Design a common emitter amplifier DC-bias circuit using only our 12V source.

See figure 25 for details. Note that this is not exact science, the currents depend in a complicated way on the voltages and the equations can only be solved numerically. So we need to rely on simplifications and common sense.
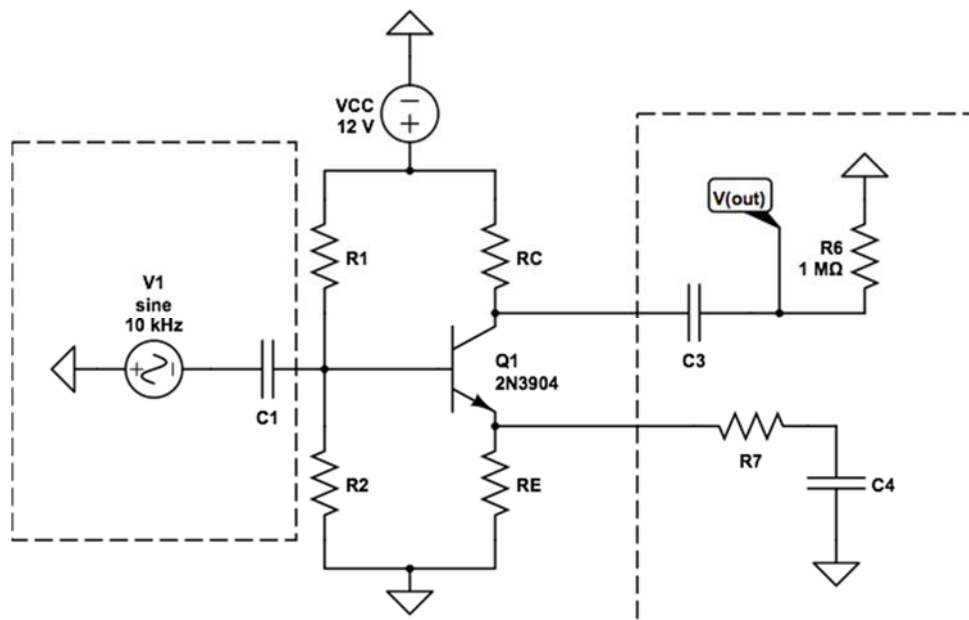
Figure 25: A common emitter amplifier with DC-bias and AC coupled in and outputs.

The first step is to define the quiescent point in the I-V-diagram. As an example, we choose I_C = 4mA. The output voltage for I_C should be $V_C = V_{CC}/2 = 6V$ to ensure a large dynamic range. This sets R_C = 6V/4mA = 1.5kΩ. The emitter resistor sets the emitter voltage and the DC gain (which we don't care about for our AC signals). A good rule of thumb is to ensure that $V_E = 1V$. So we will use $R_E > 1V/4mA = 250Ω$.

Now we can calculate the voltage divider. $V_B \sim 1.7V$ for all expected base currents. This requires that $R_{TH} << ßR_E = 25kΩ$ (we use ß=100 in this example). A factor 10 between them is typically OK. So select R1 and R2 such that $V_{TH} \sim 1.7V$ and $R_{TH} \sim 2.5kΩ$. R2 = 3kΩ and R1 = 18k will do. This would complete the circuit. Use circuit lab and select a 2N2222 transistor to model the circuit (Run DC Solver) to verify the voltages and currents around the transistor. The values will not be exact but close enough for our circuit.

Use a 2N3904 and a quiescent current of $I_C = 0.5mA$ to design your own common emitter amplifier DC-bias circuit. Model the circuit to verify your design and then set it up, measure the base, collector, and emitter voltages and compare with the expected values.

Use this circuit and add the parts in the dashed areas in figure 25 to build an amplifier with a gain of 100 for a 10kHz input signal with a 10mV amplitude. Use a voltage divider (not shown) with a 10k and 1.2k resistor to divide the 100mV signal by about a factor 10 prior to C1. Then choose C1, C3, C4, and R7 to achieve a gain of 100 at 10kHz. Note that R6 (1MΩ) should only be required for the simulation.

Once this works, measure the transfer function of the circuit: Measure the ratio |V(out)|/|V(in)| at 1Hz, 3Hz, 10Hz, … up to 30MHz. If the signal is to small, increase the amplitude of the input signal as needed. Which components define the bandwidth? Do the simulated and experimental results agree?

Emitter follower and bootstrapping:

Figure 26 shows a DC-biased emitter follower. A well designed bias circuit which requires a large dynamic range will set V(emit) to V2/2 ~ 6V. If the selected quiescent current is $I_C=I_E=0.5mA$, RE has to be 12kΩ. The base voltage has to be around $V_B=6.7V$. A 'stiff' Thevenin voltage source requires $R_{TH} < ßRE/10$. This allows us to determine R1 = 215k and R2 = 270k.

Your task is to design a similar DC-biased emitter follower with a quiescent current of 1mA and a quiescent emitter voltage of 2V. Simulate the circuit, set it up, measure the DC base and emitter voltages and compare the experimental and simulated results (see central part of Figure 26 for the layout).
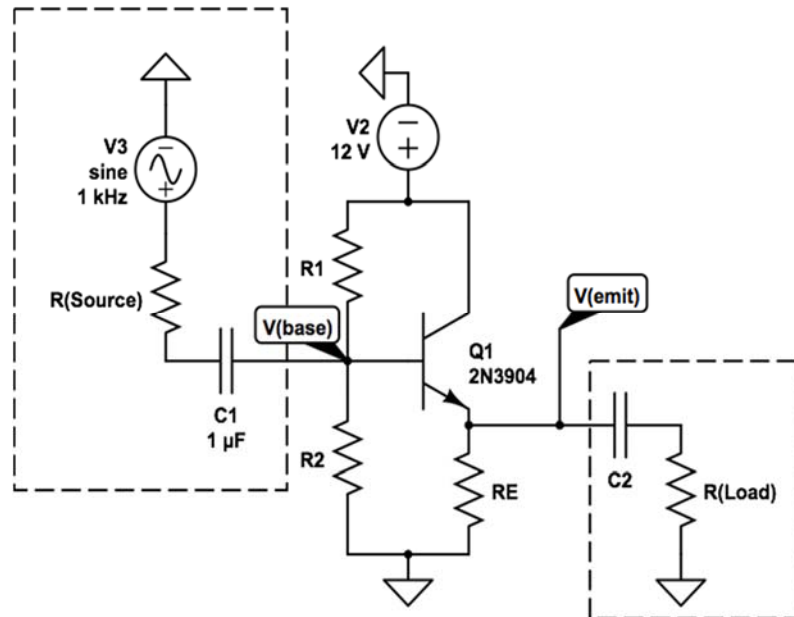
Figure 26: An emitter follower with DC-bias and AC coupled inputs and outputs.

Now add the AC voltage source on the left first without the source resistor and also add the AC-coupled load (C2=1μF and R(Load) = 1MΩ) to the circuit. Use a BNC-T to measure the input signal with the scope. Use a scope probe to measure the voltage across R(Load). Set the function generator to produce 100mV sin wave with a 1kHz frequent. The voltage across the load should be identical to the input voltage.

Add a 20kΩ source resistor to the input to represent a high impedance signal source. The signal across the 1MΩ resistor should now be significantly reduced. The reason is that the source resistor 'sees' not only the transistor and the impedance at the emitter as a load but also the two resistors R1 and R2 which produce the DC-bias voltage as an additional load. In this particular setup, this additional load will be very close to R2. The circuit in Figure 27 solves this problem.
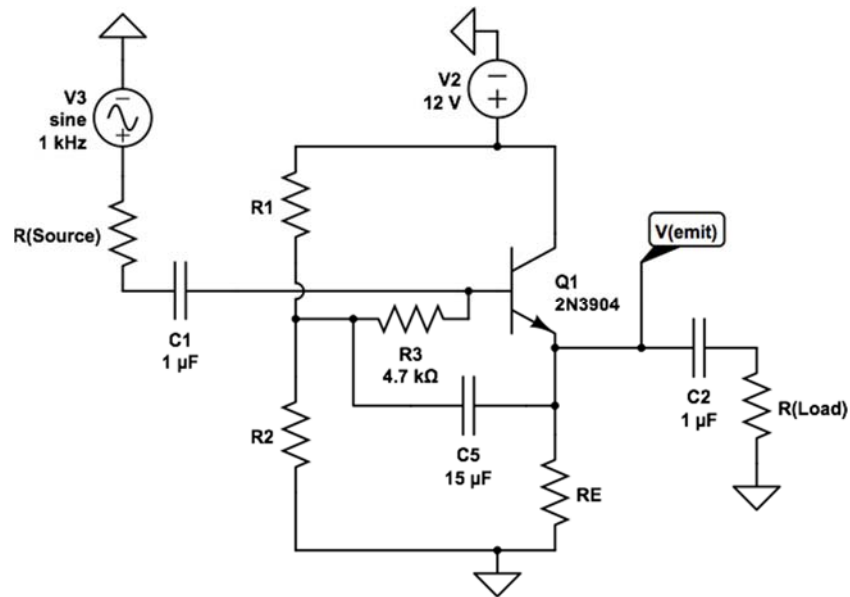
Figure 27: An emitter follower with DC-bias, AC coupled in and outputs, and bootstrapping

Add C5 and R3 to your circuit and verify that the signal across R(Load) is again nearly identical to the input signal even with a 20kΩ source load. Explanation: The impedance of C5 is negligible for the signal frequency and is only there to block the DC signal. As a result of the connection R3 is parallel to the base emitter junction of the transistor. As the emitter follows the base, the voltage across R3 will not change and this prevents the AC current from 'seeing' R1 and R2. The only impedance visible for the source is the impedance at the emitter times ß.

A 1MΩ load resistor is not useful in this circuit; we only used it to show the effects of bootstrapping. The current through such a load would be so small that the 20kΩ source could drive the load directly. A more realistic load is a low impedance load which needs to see a large current instead of a voltage. This can be represented for example by a 100Ω load. Driving such a load directly with a 20kΩ source would reduce the voltage by a factor 200. The emitter follower reduces the voltage only by a small factor. Replace the 1MΩ load with a 100Ω load resistance and measure the amplitude of the voltage across it.

Lab 5: Transistors II

In Lab 4, you characterized transistors and set up the first amplifier circuits. We assumed to some degree that transistors are ideal current amplifiers which exist in imperfect circuits with for example limited supply voltages (which lead to saturation). In this Lab, we want to go a little bit beyond the idealized behavior and also look at field effect transistors or FETs.

DARLINGTON

We start with the Darlington circuit, an often used high gain amplifier. See the HH 6.4 for details.

Set up and simulate the circuit shown in Figure 28.
Use a resistor substitution box for R.

- Set R to be a few kΩ. Record your value of R. At this point the transistors provide very limited resistance between the collector of Q1 and ground and operate in the saturation region (V_{CE} small), and $I_C \leq 12$ mA. Measure V_C and V_B of Q1. How do they compare with single transistor values which you could measure by grounding the emitter of Q1. Explain.
- Increase R to 500k-50M, to force I_C down to a few mA. Measure ® at collector currents around 1mA and 10 mA. Also monitor V1 and V2. V2 can be used to calculate the input current. If needed, use a voltage divider to reduce V1 by a factor of 10 to around 500mV.
- What is the overall current gain of the circuit?
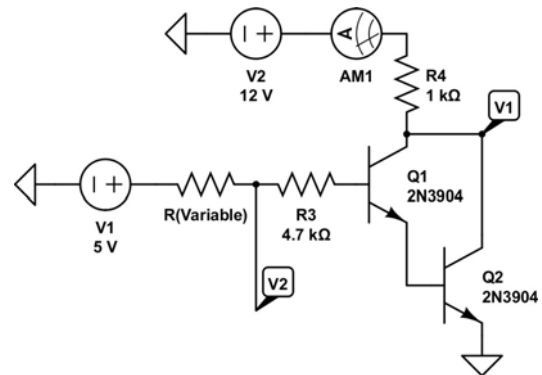
Figure 28: Darlington circuit.

Transistor issues: Temperature and Early effect

Current Mirrors

Source: HH 5.3 (page 120 in Student Manual)

- The circuit in Figure 29 uses two pnp transistors (the circuit would also work with npn transistors) to build a current mirror. The assumption is that in the linear regime the collector current is controlled by the base current which in turn is controlled by the base emitter voltage. This assumes that the I-V curve is flat in the linear regime.
- Build the classic current mirror shown on the right. How closely does the output current equal the programming current (I_C) through the 15kΩ resistor? The match should be pretty bad, due to the fact that the I-V curve is not really flat (this is called the Early Effect) and also due to temperature
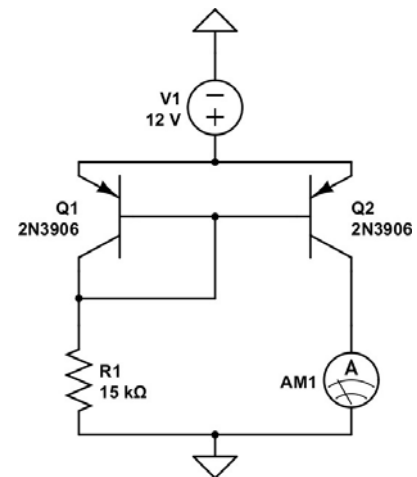

Figure 29: Current mirror I

effects. Both tend to make I_{out} larger than $I_{program}$. Squeeze each of the transistors in turn and see how much I_{out} is affected by differences in temperature between Q_1 and Q_2. Does this pattern of response make sense?

• Build the same circuit using a transistor array, the CA3096. The figure on page 120 of HH Lab book shows you the pinout. Measure I_{out} using the matched PNP pair of the CA3096. How closely does I_{out} match $I_{program}$? This version should perform better; the proximity of the two transistors will remove temperature differences and effects. However, you are still likely to see I_{out} larger than $I_{program}$. Why? Add a 10k pot in series with the current meter and vary its setting. Does the output current vary? Why or why not?

• The Wilson Mirror in Figure 30 beats the variations in output current due to the Early Effect. Watch the variation in output current as you vary the load. How constant is the current now? What property of this circuit accounts for the improved performance?

• Simulate the classic current mirror and the Wilson mirror and compare the results with your experimental results.

Figure 30: Current mirror II

Field Effect Transistors

Field effect transistors or FETs control the current between drain and source with the gate-source voltage.

FET Characteristics

Source: HH 7.1

Set up the circuit shown in Figure 31.

• Measure (and simulate) I_{DDS} and V_P for a few samples of 2N5485 (use J310 for the simulation).

• Verify the relation between I_D and V_{GS}, in a semi-log plot.

• Check that your values fall within the quoted maximum range:

4 mA $< I_{DSS} <$ 10 mA

-4 V $< V_P <$ -0.5 V
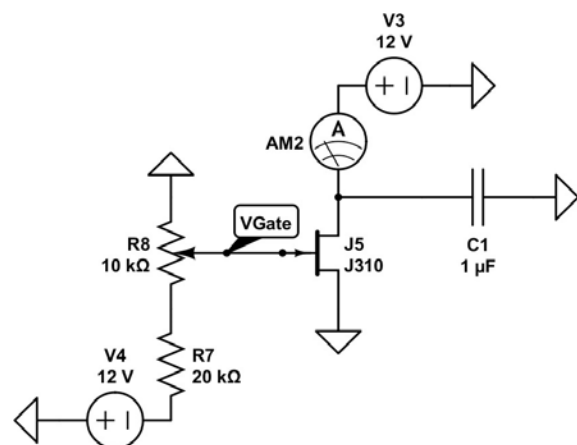
Figure 31: FET Characteristics

- Pick out one of the transistors to use for the following sections. Note which one you're using.

FET Current Sources

Source: HH 7.2

Set up the circuit shown in Figure 32 and vary the resistance of the load R1 between 20k and 100K, and watch V_{DS} with a multimeter as you monitor I_{out}. What is V_{DS} when the constant current behavior starts to break down? Make a table containing 6 to 7 points that covers the region when you switch from constant current to the linear range. It should occur when V_{DS} is near $V_{GS} - V_P$. Does your FET's linear region begin around this value of V_{DS}?

Simulate the circuit using the DC Sweep function. Plot Vdrain and Vsource and compare with your experimental results.

Discuss what is going on in the lab report.
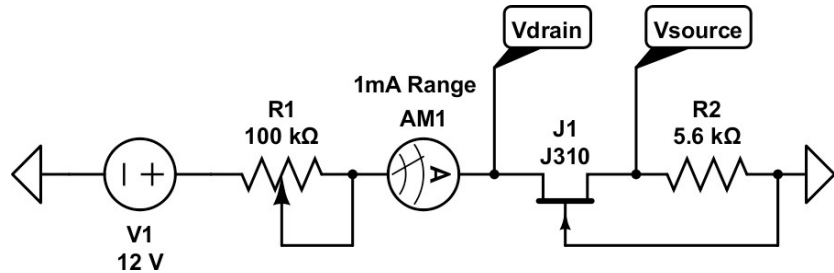
Figure 32: FET as a Current Source

FET as a Variable Resistor

Source: HH 7.4

When you tested the FET as a current source you found that the circuit failed when V_{DS} shrank so far that the device fell into the linear region. This circuit shown in Figure 33 is intended to always operate in the linear region. It will fail if V_{DS} gets too large. Drive the circuit with a ~0.2V sine wave at 1 kHz. Vary the potentiometer and record the results in attenuation and amounts of distortion. If you're not seeing the distortion clearly, try driving the circuit with a triangle wave. How would you predict this distortion? How does the circuit treat a larger input waveform?
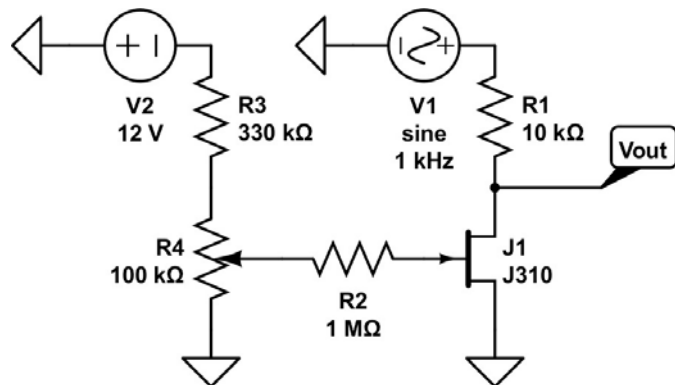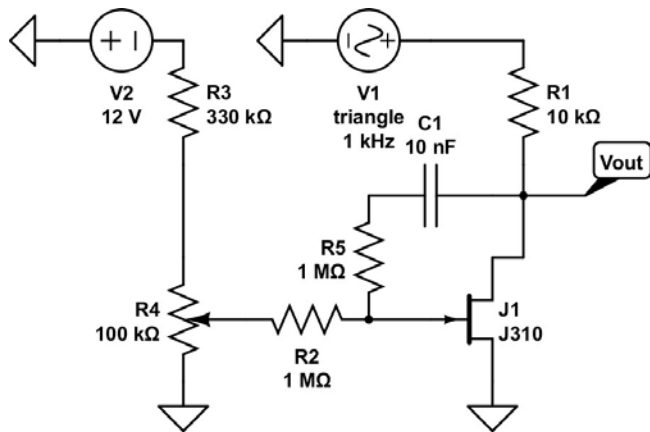
Figure 33: FET as a variable resistor

Figure 34: FET with improved linear range

It might be best to first simulate the circuit with a triangular wave input and find a good set of parameters to see the distortion in the simulation. Then choose similar parameters in the experiment. They won't be identical because the type of FETs are different.

Adding ½ v_{DS} to the gate signal straightens out the curves (see Figure 34 for how to do this). Examine the effect on the shape of V_{out} as you drive the circuit with a ~0.2 V triangle waveform. Simulate, build and test the circuit. Your lab report should include a discussion how and why this works.

The previous two FET circuits used a potentiometer to vary the attenuation of a signal. You didn't need a FET for that purpose; the potentiometer alone would do the trick. The following circuit shown in Figure 35 uses the ability of the FET to control it's own resistance via a changing voltage. This is a much more useful application.

Use a second function generator to drive the point that you drove with the pot. This lets the second signal modulate, or vary, the attenuation periodically. The frequency of modulation should be much lower than the signal frequency. Try a modulation frequency around 50 Hz, with a small amplitude around 0.2V. Trigger the scope on the modulating signal, not the composite output.

You can turn this circuit into an AM radio transmitter! Use a sine wave around 1 MHz for V1. Attach a few pieces of wire to the point called out, and then tune an AM radio to a quiet place on the dial. Adjust the sine wave frequency until you can hear the modulating signal (single tone). You'll need to adjust the frequency in very small steps, so that you don't miss the modulation signal.

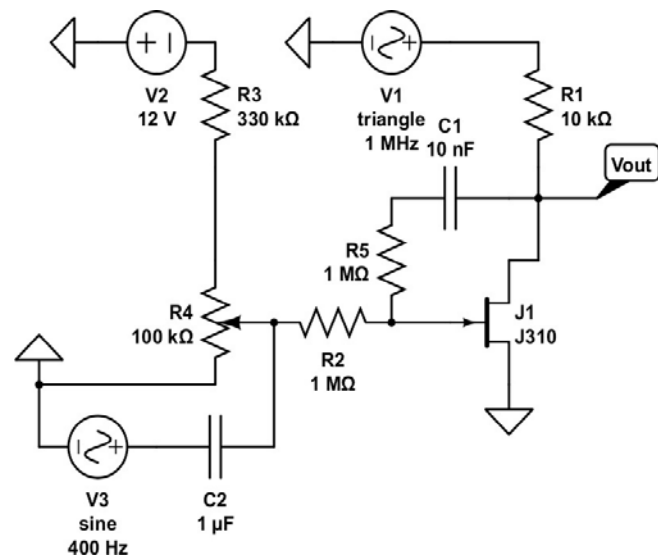**Demonstrate that this circuit works.**

Figure 35: Building a radio

Lab 6: Operational Amplifiers

We will be using one of the best-known op-amps, the 741. It is in an 8-pin device dual-inline package (DIP). A dot on one corner of the package identifies pin 1. The internal connection to each pin is shown here together with a schematic diagram of the input and output of the op amp.

The op amp requires external power. Connect the +12V to pin 7, the −12V to pin 4. These connections are omitted in most of diagrams but must be provided in the circuit for the op amp to work.. In simulations where railing or saturation could be a problem use the op amp with explicit rail inputs (coming out from the sides) and power these in the simulation.
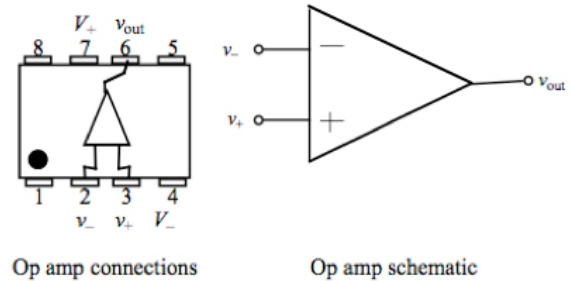
Figure 36: Operational Amplifier

Op-Amps have very high input impedances and very low output resistances; V_{out} is a voltage source with a low Thevenin impedance.

The ideal Op-Amp follows two simple rules which are often called the Golden Rules of Op-Amps:

- Op-Amps adjust V_{out} to ensure $V_- = V_+$

  - provides the current to ensure $V_- = V_+$

- The Op-Amp inputs draw no current.

These two rules allow you to analyze the response of an Op-Amp under ideal conditions.
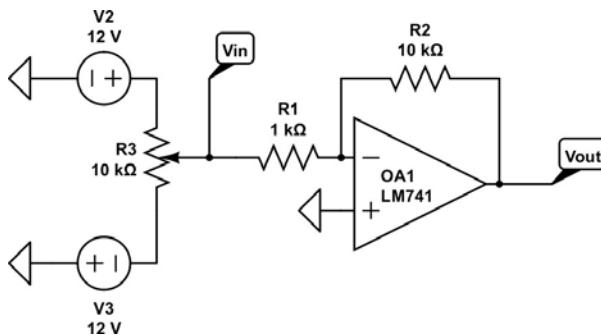
Basic **operational amplifier circuits**

Figure 37: Inverting amplifier

Inverting and non-inverting amplifier at low frequencies

The circuit on the right shows an inverting amplifier circuit. The +/-12V sources and the 10k pot are there to tune Vin.

- Simulate and build the circuit.

In simulation and in the experiment:

• Measure Vin and Vout for various settings of the potentiometer R3.

• What are the max and min values for Vout?

• What is the amplifier gain (Vout/Vin) when the Op-Amp is not saturating or railing? Find this for three distinct Vin settings when your output is not at either rail.

• Does this agree with your simulation?

• Find the gain of the amplifier for low-frequency (10-100Hz) AC input at several frequencies with distinct amplitudes (at least 3): Connect the output of the function generator to Vin of the inverting amplifier circuit. Is the DC gain, found above, and the AC gain the same?

• To test for the voltage offset leave the non-inverting input connected to ground and replace the AC input connected to R1 with a wire to ground, so that the inverting input is now also grounded (but through R1). The op amps voltage offset is the output voltage (measure it) divided by the gain of this resistor combination.

A second way of connecting an op amp as an amplifier is shown below.

Figure 38: Non-inverting amplifier
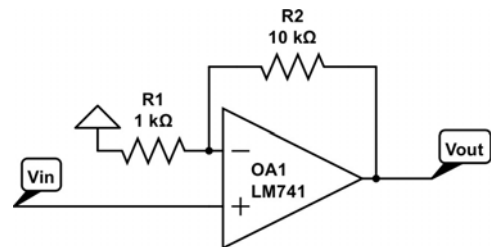
• Repeat your 3 AC and 3 DC pot gain measurements with the same set of original resistors. Repeat these measurements at the same frequencies for AC and pot values for DC. Compare the results to the values found for the previous circuit.

• Why is one circuit called "inverting" and the other "non-inverting"?

• Do your results agree with your simulations?

The lab report should include the golden rules and how to apply them to these two cases.

Inverting and non-inverting amplifier at high frequencies

• Assemble an inverting amplifier with a gain of −10 and measure its gain from 100 Hz to 1MHz. Make measurements in decade steps (100 Hz, 1 kHz, 10 kHz, etc.). Record the values of the resistors used.

• Assemble a gain −1000 amplifier and repeat the measurements. Hint: With a gain this large, you will need to send your input AC through a voltage divider to reduce its voltage dramatically, so that your output is not hitting the rails.

• Make plots of the log of the gain versus the log of the frequency, putting both plots on the same graph. Report how the two graphs are similar and how they differ.

• Simulate the frequency dependence of these two amplifiers using the frequency domain tool. Be aware that in the frequency domain CircuitLab always makes the input voltage 1V.

o   An important parameter for op amps is the gain-bandwidth product (GBW). Op amps are designed to have extremely high open loop gain, that is, even a few ten microvolts difference between the two inputs can give >1V at the output (making the open loop gain G of order 1V/10μV=100,000 or more). A drawback of this enormous gain is a limited frequency range (the bandwidth) over which the op amp can deliver this high gain. Defining the bandwidth as the range from DC to the frequency where the gain has dropped by 3dB ($0.707G_{DC}$) we would find that the LM741 op amp has an open loop gain bandwidth of less than 10Hz.

This would seem to severely limit op amps to very low speed circuits but in fact op amps are designed with such excess high gain to exploit negative feedback. That is, by picking off a portion of the output and feeding it back to the inverting terminal (through an impedance) we give up some of the high gain, with the benefit of getting a greater bandwidth. In fact op amps are designed to have a constant gain-bandwidth product (GBW) so that knowing the GBW allows you to calculate the frequency at which your op amps gain (with feedback) will begin suffer. For example the LM741 has a GBW of 1.5 MHz. Which means that given a feedback that makes its voltage gain 1, it will have this gain from DC to 1.5 MHz (actually, at the latter frequency its gain will have fallen off to $0.707G_{DC}=0.707$). A voltage gain of 1 does not seem too useful (though it might be, as a follower, giving a current gain) but suppose you make the voltage gain 10. Since the GBW = 1.5 MHz is constant this means that $(10)(f_{3dB})$=1.5 MHz so the new bandwidth at this gain will go out to f_{3dB}=1.5 MHz/10=150 kHz.

o   Increase the GBW Parameter of the Op-Amp in the simulation by a factor 10. How does this change the frequency response in the simulation?

o   Pick an Op-Amp with a larger GBW compared to the LM741 and redo the simulation.

•   Repeat for a non-inverting amplifier.

Discuss the frequency responses in your lab report.

What did you expect?

What did you see?

Input impedance of Op-Amp circuits

As discussed before, low input and high output impedances of an electronic device can dramatically alter the behavior of a circuit. We will now measure the input impedance of the inverting and non-inverting amplifier circuits (Note: This is **not** a measurement of the input impedance of the operational amplifier alone). The simplest way to do this is to add the resistor

box (R4 in the figure on the right) between the function generator (generating a low frequency ~10Hz or DC signal) at the input of the circuit (Vin in both figures above) and increase the resistance (R4) until the output voltage drops by a factor 2. Do this for different R1 resistors (1k and 10k for example) for both cases.
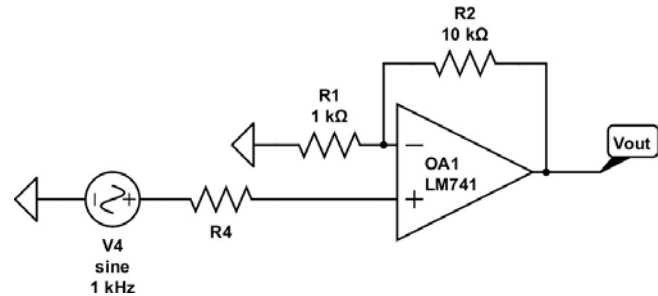
In your write-up describe what you did and what (and why) you conclude the input impedance to be what it is. The reason you can measure this on the LM741 op amp is because it has conventional BJT (npn) transistor inputs. If you tried this with say an LF411 op amp which uses JFET inputs you would have little luck in measuring this. Look up the data sheet for an LF411 and report what it says about its input resistance.

Figure 40: Current-to-voltage converter

Other standard applications of operational amplifier

The current-to-voltage converter

How do the two golden rules permit us to analyze the operation of a current-to-voltage converter? Consider the photometer circuit shown here (use the OP801WSL phototransistor, the emitter is the wire closest to the tab on the case and the collector the furthest from that tab). The second golden rule means that the current through the phototransistor, I, is equal to the current through the resistor. The first golden rule says that the summing point, the - input, is at virtual ground. That means that the current through the resistor is given by $I = -v_{out}/R$. Thus $v_{out} = -IR$. Also because of the first golden rule the voltage across the phototransistor is 12V, no matter what the current through it is. This ensures a wide linear range for the photo transistor.
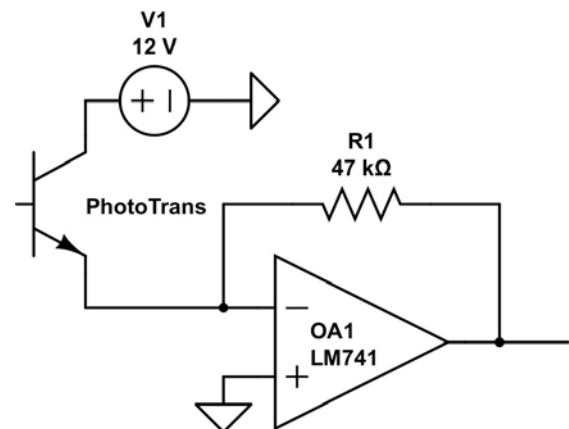
Set the circuit up and use the polarizer sheets and a toilet paper roll to measure the voltage response as a function of the angle between the sheets. Change R1 such that the output voltage is at least a few volts for aligned polarizer sheets (maximum signal). Does the response confirm the law of Malus (see for example: http://www.physicshandbook.com/laws/maluslaw.htm)?

Figure 41: The summing amplifier

Figure 39: Input Impedance measurement

Figure 42: The difference amplifier

Mathematical operations

The summing amplifier shown on the right is one example of a circuit that is capable of a mathematical operation. It is often used to add a DC offset to an AC signal. Set up the summing amplifier with resistors in the 1k-10k range to generate a 2V 1 kHz signal riding on a 1V DC offset starting with a 200mV AC signal from the function generator. Simulate the circuit first so you know what to expect.
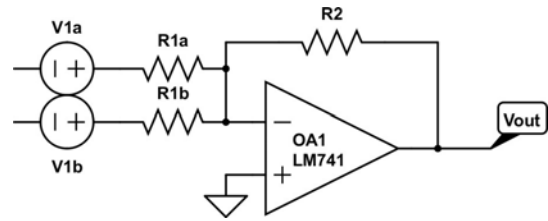
The difference amplifier shown on the right produces the weighted difference between the two voltages V1a and V1b. Set it up with resistors in the 1k-10k range to subtract a DC offset from an AC signal. Simulate the circuit first and start by making all resistances 1k. Then vary each resistor (one at a time until you understand what each one does to the output.
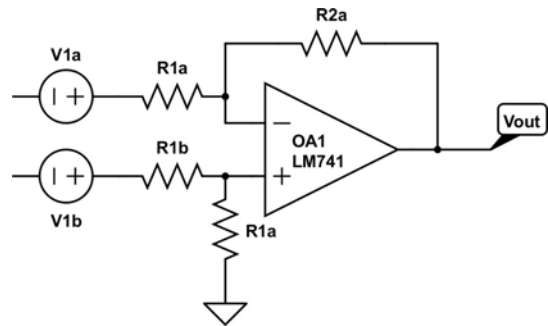
For the lab report: Derive the output voltages for the summing and the difference amplifier using the golden rules. Calculate the expected signals and compare with the experiment and simulation.

The integrator

Construct the integrator shown here, using a 100K resistor and a 0.01 μF capacitor. Drive it with a 1 kHz square wave and save the output image. This circuit is sensitive to small DC offsets on the input waveform. Predict the peak-to-peak triangle wave amplitude at the output that would result from a 2Vpp, 500 Hz square wave input, and then try it. How well did you predict the output? What would happen if you were to put a 10M resistor in parallel with the capacitor in the feedback circuit? What is the function of the 10M resistor? Try it!
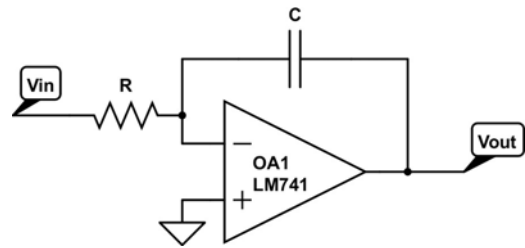
The Differentiator

Try driving this active differentiator with a 10 Hz triangle wave, and then a 10 Hz sine wave. Does the signal look like the one you expect? Use the Golden Rules to derive the output expected, and compare to what you've observed.
Change the frequency and see when the signal starts to look no longer like the derivative of the input signal. Simulate the circuit.
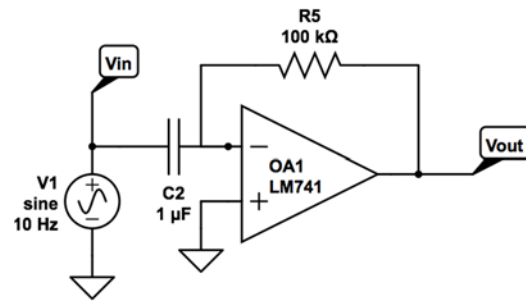Discuss what happens here.

Figure 44: The differentiator

Figure 43: The integrator

Lab 7: Extending the use of Op-Amps: Diodes and Transistors

Current drivers controlled by operational amplifiers

The output current of an LM741 is limited to 20 mA. How can you connect an operational amplifier to a high-current device such as one that can produce light, sound, magnetic fields, or heat? Rather than using special-purpose high-current op-amps, we can use diodes and transistors for the high current part but continue to use the Op-Amp to control everything.

LED Driver:

The circuit in Figure 45 allows one to change the current through the LED by changing the input voltage into the Op-Amp.

Use the HP to generate Vin. The inverting Op-Amp circuit will invert and amplify this signal. V(Op-Amp) has to be positive to forward bias the base-emitter junction. **So Vin has to be negative**!
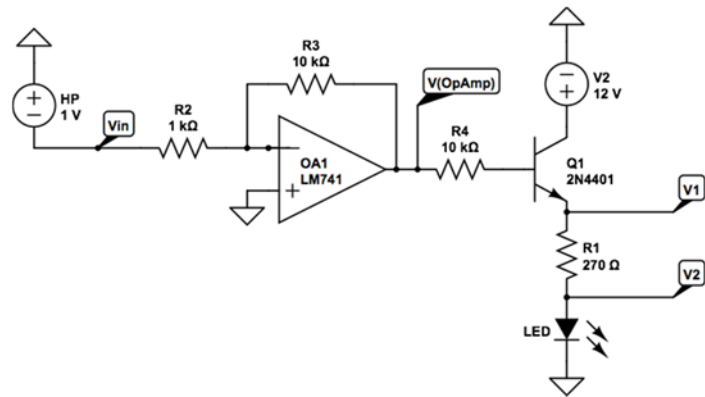
R4 limits the base current going into the

Figure 45: The LED Driver

base to ~1mA. The base current sets the collector current which is then going through R1 and the LED.

Adjust the voltage at Vin and measure (and simulate) V1 and V2 observing the intensity of the light emitted by the LED. Obviously, the difference between the voltages gives you the current going through the LED. What is the base current? What is the ß of the transistor?

Replace the HP by the function generator. Set it to ~1V and change the frequency from initial 0.3Hz to a frequency at which you don't see the LED blinking anymore.

In the lab report you should plot and discuss the results. Simulate the circuit.

Constant-current source

Setup the circuit in Figure 46 on your breadboard and in the simulator. The current through R3 will be (nearly) identical to the current through R2. The Op-Amp will adjust its output voltage and with it the base current to ensure that V- = V+ . (12V-V-) sets the current through R2.

Tune R1 to limit the current to 0.1mA. The change R3 and confirm that the current is independent of its value. This circuit is a load independent current source (up to a certain level).

Figure 46: The constant current source

To see why the current through R3 is independent of   its resistance,

analyze the op amp circuit using the two golden rules.

Push-Pull current driver

Producing high-current, positive and negative voltages with an op amp

The single transistor driver has an important limitation: there is no way to have an output that can produce either positive or negative currents. A combination of two transistors, one *npn*, the other *pnp* called a push-pull pair, avoids this problem. It is used in many circuits to drive a variety of devices. For now, we'll just explore how it works.

The TIP series of transistors are power transistors (30 W) capable of 1A current (hence the heat sunk package). Please note that the pinout for the TIPs are different from the transistor pinouts (search out the data sheet to learn how to connect them).

Use a function generator to supply a 1-kHz sine-wave signal to the input of a 741 op amp. View the output on the oscilloscope as you change the input amplitude. **Save the output images. Why is the output not a faithful reproduction of the input?** Also measure the voltage at the base of the transistors. Is this a faithful reproduction of the input?

Simulate the circuit as well using TIP31 and TIP 32 transistors.

A simple change can improve this: Instead of connecting the feedback resistor R2 to the output of the Op-Amp, connect it to V1 and then repeat the measurement and the simulation. Is this now a more faithful reproduction of the input signal? How does the output voltage of the Op-Amp change now? Discuss this in your lab report.
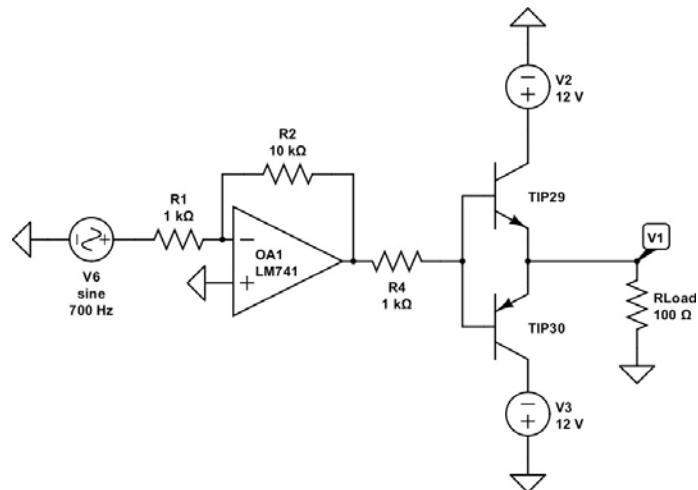

Figure 47: The Push-Pull current driver

Other **applications of operational amplifier**

Active Rectifier and Active Clamps

Simulate and build the two circuits below. In the simulation, you have to include the supply lines for the Op-Amps; otherwise the simulator assumes a constant voltage gain independent of the input level.
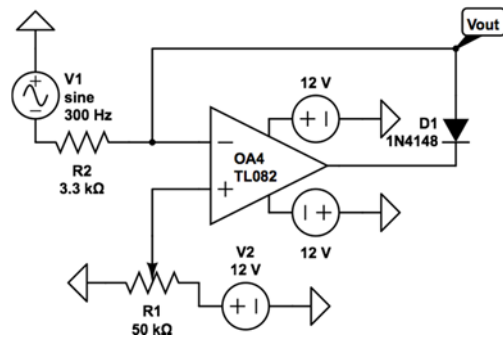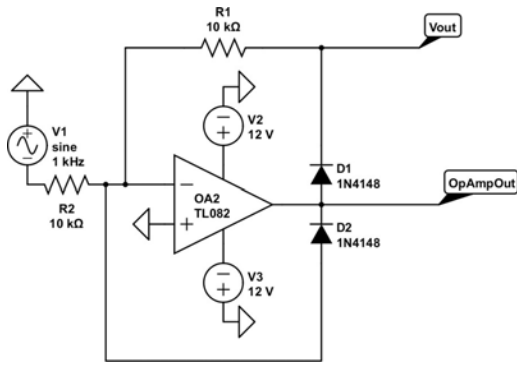
Figure 48: Active rectifier and active clamp

The active rectifier will produce a signal which is similar to the half wave rectifier. For a positive V1 voltage the Op-Amp will generate a negative Op-AmpOut voltage which has to be high enough to drop 0.6V across D2 and still achieve 0V at V-. Vout will now be 0V as well. For a negative V1 voltage the Op-Amp will generate a positive voltage which opens D1 (voltage drop of 0.6V) and creates Vout such that V-=0. As R1 = R2, Vout = - V1 in this case. Look how the voltages at all nodes change with respect to the input signal. This will help you to understand the circuit.

The output voltage Vout in the active clamp circuit will be limited to the positive voltage applied to V+. How does this work? For a voltage V1 < V+, the Op-Amp will generate a positive output voltage which will be blocked by the diode from reaching Vout. As the negative input of the Op-Amp is high impedance, Vout = V1. For a voltage V1 > V+, the Op-Amp will generate a negative output. This will open the diode and allow the Op-Amp to set V-=V+ and clamp the voltage at Vout to V+.

Common mode rejection ratio

In principle a differential amplifier should not amplify a signal that is common (the same) to both inputs. In practice that is difficult to achieve. The common mode rejection ratio is a measure of how good an amplifier does in amplifying the difference signal, while rejecting common mode signal. If you measure an amplifier's differential gain and divide it by its common mode gain you get the common mode rejection ratio (CMRR). This is usually expressed in dB as:

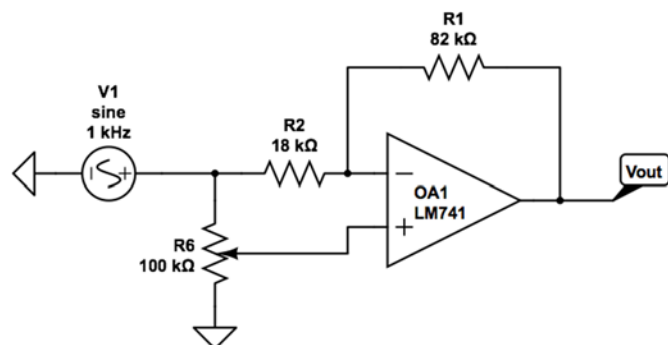
Figure 49: CMRR

$$CMRR = 20Log\frac{G_{diff}}{G_{comm}}$$

Since the common mode gain appears in the denominator and the ideal amplifier should ignore common signals (i.e. in principle $G_{comm} = 0$) the ideal amplifier has a CMRR that is infinite.

Set up the circuit shown in Figure 49 and measure G_{comm} by tuning R1 to minimize $|V_{out}|$. Calculate $G_{comm} = |Vout/Vin|$ from the data. Measure the ratio of the two resistor parts in the pot (you need to take the pot our of the circuit for this). Then measure the differential gain by applying the same signal to only the inverting input port (ground the other). Calculate $G_{diff} = |Vout/Vin|$. Calculate CMRR from the results. The 741 has a CMRR of at least 70dB.

What is the ratio of the two resistor parts inside the potentiometer when the output voltage was minimized? Does this agree with your expectation?

What you have measured here is the best CMRR you can get using a potentiostat to provide the balanced resistors you need for a differential amplifier. If we had precision resistor pairs that were equal to better than 0.1% we could measure the intrinsic CMRR of the LM741 op amp, which should be even higher than what you got from the relatively crude measurement here.

Lab 8: Oscillators and Timers

We have seen in Labs 6 and 7 how negative feedback can be used to perform mathematical operations, build filters, or control for example current sources. Lab 8 will show how to use positive feedback to build comparators and oscillators. It will also introduce a specific integrated circuit, the '555', which is often used in timing circuits. Several of the circuits are also discussed in Chapter 7 of the text book.

Positive feedback in Op-Amps
The comparator and Schmitt trigger

The output voltage of a comparator without feedback is either low (near 0V) or high (near 5V). It depends on which of the two input signals is larger. When both voltages are nearly identical, any small change can cause the output to change. This is an unstable situation which can lead to oscillations. A Schmitt trigger uses positive feedback to create a hysteresis which avoids this issue.

Note that the comparator is essentially a switch. Its output terminal is the collector terminal of a transistor (within the device) that has its emitter connected to ground (usually via pin 1). Referring to Figure 50, if $V^+ > V^-$ that transistor is off so that Vout rises to V2 through R2 (the pull-up resistor). Alternatively, if $V^+ < V^-$ that transistor is on, and its grounded emitter takes the output to ground.

Set up the circuit shown in Figure 50. Use the 311 as a comparator. First set the circuit up without positive feedback (Do not install R5 yet). Connect Vout to the scope and tune the potentiometer such that the output is high. Next adjust the potentiometer such that Vout goes low. Finally, tune the potentiometer so that you are close to the switch point. If you are close enough you should see an oscillation at the output. Save an image of the oscillations.

Now we want to measure the slew rate of the comparator. Remove R3 and R4 and connect the function generator to the positive input of CMP1. Tune R1 such that the voltage at the negative input is 1V. Set the function generator to produce a 1kHz square wave with a 1V (2Vpp) amplitude and a 1V DC offset. Display Vout on the scope and measure the slew rate of the comparator. Increase the frequency to 10 kHz and then to 100 kHz. Does the slew rate depend on the input frequency?

Figure 50: Comparator and Schmitt trigger

Restore the original circuit and add the positive feedback resistor R5. This circuit is now an example of a Schmitt trigger. Measure the hysteresis. Simulate the circuit; circuit lab has a comparator block which works reasonably well. Compare the simulated and measured hysteresis.
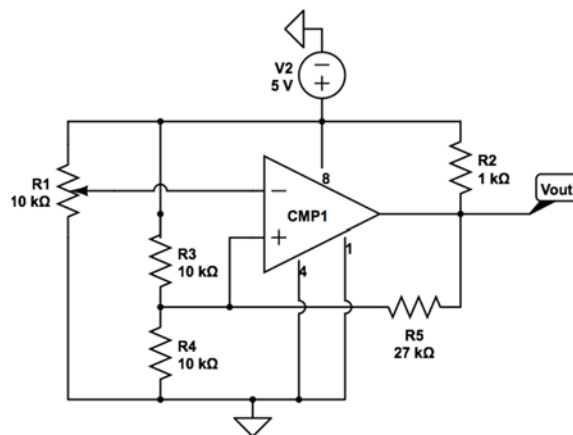
The RC relaxation oscillator

The RC relaxation oscillator shown in Figure 51 is another example of a useful positive feedback circuit. Set it up on your breadboard and measure the voltages at the inveting and non-inverting terminals and Vout pairwise with the scope.

Figure 51: RC relaxation oscillator

The resulting square wave at Vout allows you to measure a lower limit for the slew rate of the 741. Calculate the frequency of the oscillation and compare with your experimental and simulated data.

Explain what is happening in your lab report: V⁻ shows the charging and discharging of C1. Vout sets two different voltage levels for V⁺ (which ones?). Once V⁻ reaches V⁺, Vout changes. This changes V⁺ and also either ends the charging of C1 and starts the discharging or ends the discharging and starts the charging until V⁻ is again equal to V⁺ (or better: until V⁻-V⁺ changes sign again).

Although the time constant of the charging and discharging process is in both cases given by R1 and C1, the square wave is not symmetric. Why? Think about where you are in the exponential charging/discharging curve with respect to the final voltage (final voltage if the Op-Amp wouldn't switch again).
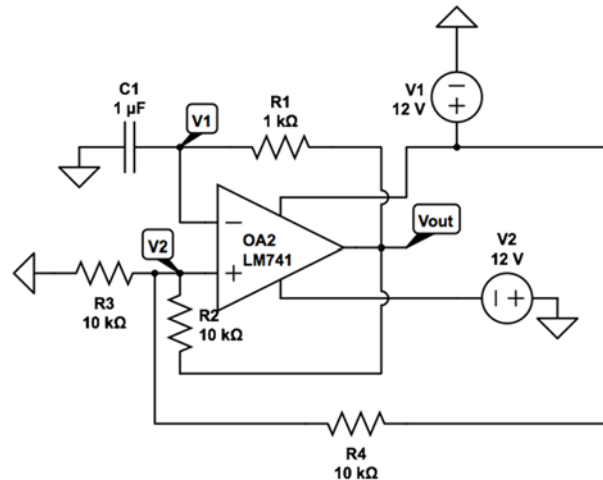
The monostable multivibrator

The 555 (pronounced "five-fifty-five") is one of the most versatile integrated circuits ever made. The 555 contains a three-resistor voltage divider, establishing reference voltages of 2/3 *V*+ and 1/3 *V*+. The left-hand comparator compares the threshold input to 2/3 *V*+. The right-hand comparator compares the trigger input to 1/3 *V*+.

These two comparators go to a Flip Flop (F/F). The Trigger comparator turns the Flip Flop on, the threshold turns it off. When the Flip Flop is on (off), Q (not used) is high, Qbar is low (high). Qbar sets the base voltage of the transistor to 0V (5V). This blocks (allows) current flow between pin 7 and ground through the transistor. The output signal (pin 3) monitors the state of the transistor.
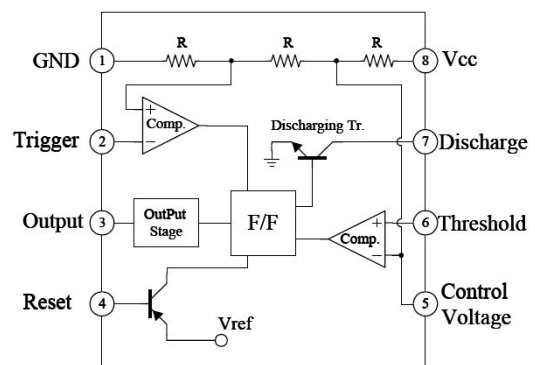

Figure 52: 555 equivalent circuit

555 test circuit

You will first explore the output signal and discharge connection as a function of the three inputs: trigger, threshold and reset. The experimental setup for this test is shown in Figure 53. First connect the reset connection to 5V ('high'). Let us define the trigger voltage to be 'high' ('low') if it is above (below) V1/3 and the threshold voltage to be 'high' ('low') is it is above (below) 2V1/3. Now measure the output voltage for all four possible states and produce two tables similar to the one below. Start with low/low and then go first

Figure 53: 555 test circuit

clockwise through the table and then counter clockwise (the results will not be the same!). Are V_{out} and $V_{discharge}$ correlated?
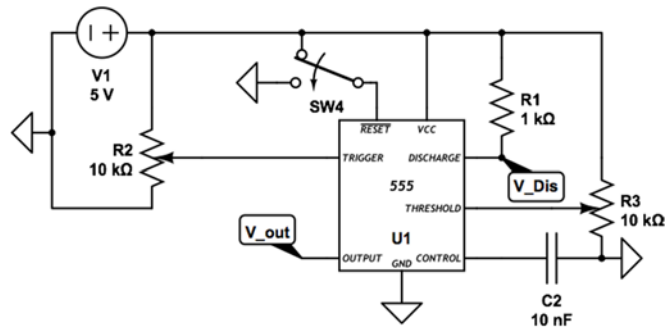
V_Thres \ V_Trigger	Low	High
Low		
High		

One Shot Operation

When the switch is **briefly** closed, pin 2, the Trigger input, is pulled to 0V. The Flip Flop is turned on. As a result the inverted output Qbar is set to 0V. This turns off the transistor connected to it, allowing the voltage on the Discharge (pin 7) to rise. Meanwhile, the Output (pin 3) is raised to 5 V.

The current through *R1* charges *C1*. When the voltage across *C1* reaches 2/3 V+ (~3.3 V) the threshold is reached. The Flip Flop is turned off. This raises Qbar to 5V. The transistor turns on, making a low-resistance connection between Discharge (pin 7) and ground, shorting the capacitor and discharging it. At same time the Output (pin 3) is turned off. The time required for the capacitor to reach the threshold voltage is $t = 1.1\ RC$.
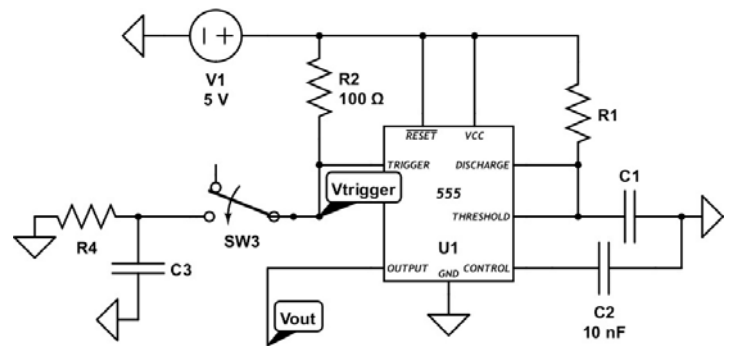
Figure 54: One shot operation

It is important that the voltage at the Trigger input return to 5V before the output goes back to 0V. This can be achieved by choosing R4 and C3 carefully (See below).

Set up the one-shot multivibrator. Choose R1 and C1 to generate a pulse which is about 1.1ms long. The fast switch can be realized using C3 and R4 attached to a slower push button switch. Before the switch SW3 is closed, C3 will be discharged through R4 and the potential at the open switch contact will be 0V. When the switch is closed, the trigger will be connected to C3 and will be (initially) set to 0V. However, the current through R2 will charge C3 fairly fast and raise the voltage at the trigger back to 5V. The maximum voltage it reaches will depend on R4, and the time it takes to charge C3 up via the R2-C3 time constant.

Measure the trigger voltage and the Vout. In the lab report, discuss how the R1-C1 combination sets the width of the pulse and how you picked R4 and C3.

Simulate the circuit and leave it intact; you will need it later.

Astable Oscillator

The 555 has many other uses. The principal use is as an "astable" multivibrator (see Figure 55). In this mode its output oscillates, producing an asymmetric square wave (on and off times are not equal). The time constants are given by the resistors R1 and R2 and the capacitor C1.

Set up the astable oscillator with a second 555. Choose values which generate a low of ~10ms and high of ~15ms. Measure the output voltage and also the voltage Vcap. Simulate this circuit as well.
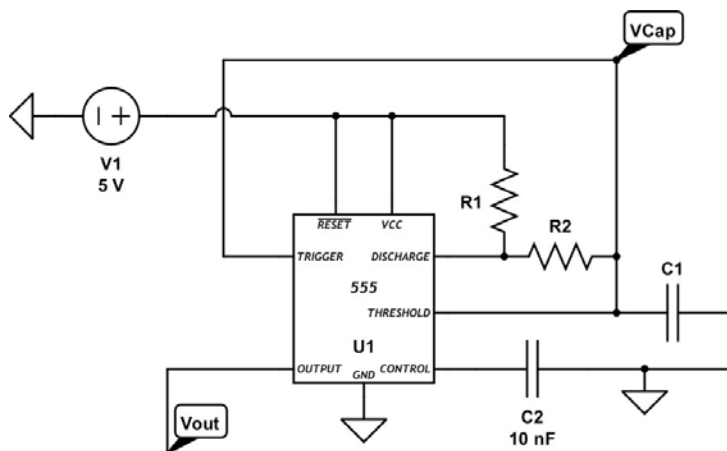

Figure 55: Astable Oscillator

Connect the output to a speaker via a 100Ω series resistor (to limit the current) and a 1k pot in series to adjust the volume of the speaker; speakers are low impedance devices. Once the speaker is buzzing, move the reset connection to ground. This should turn the speaker off. Add a switch which switches the reset connection between 5V and GND to turn the speaker on and off.

Visual and Audio Alarm system

We can use the one shot timer and the astable oscillator to build an alarm system which, when triggered, sets off a speaker and an LED. Change the time constant of the one shot timer to ~5s and connect its output to the reset connection of the astable oscillator. Also add to the output of the one shot an 820Ω resistor in series with a red LED to ground. When you close the switch of the one shot timer, the alarm should start and the LED should turn on for ~5s. Show this to the instructor during class time. Also simulate this.

Sawtooth Oscillator

A sawtooth wave can be created using the circuit. of Figure 56. Build the circuit and look at the waveform on the capacitor, using an oscilloscope. Can you predict the frequency? What do you observe? What should the output from pin 3 look like?

To understand what is going on, add a 100Ω resistor between the collector and the cap to monitor the voltage across it (don't ground any end of the resistor. Calculate the current that is charging the cap from that data. From this data you can then calculate the frequency of the sawtooth oscillator.
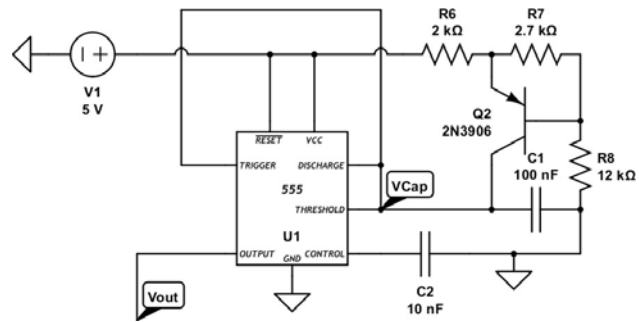
Simulate the circuit as well.

Figure 56: Sawtooth Oscillator