Chapter 1: DC circuit basics

Overview

Electrical circuit design depends first and foremost on understanding the basic quantities used for describing electricity: Voltage, current, and power. In the simplest circuits these are related by Ohm's law. After defining and understanding these quantities, we will begin a discussion of network analysis and look at a few examples.

Voltage

Voltage (measured in volts, V) is measured between two points in a circuit. It is the amount of potential energy delivered per unit charge as a charge flows along the circuit. You measure a voltage by connecting the two terminals of a voltmeter to the two points in your circuit. You must measure the voltage while your circuit is operating. If you have a good *voltmeter*, your measured voltage will be the same as the voltage was before you connected the voltmeter. A voltage decrease in the direction of current flow represents energy flowing out of the circuit (usually into heat). Voltage sources, such as batteries and power supplies, produce voltage increases along the direction of current flow.

Current

Current (measured in amperes or amps, A, which is equivalent to Coulombs/second) is measured at a single point in a circuit. It is the rate at which charge flows along the circuit. To measure a current with a *current meter* (or *ammeter*) you must break the circuit at that point and connect each end to one of the terminals of the ammeter. Current can only flow in complete circuits. That's why a switch stops an electrical circuit – it breaks the circuit and interrupts the current flow.

Power

Power (measured in Watts, W) is the rate at which electrical energy is converted into heat or some other form of energy. Since we all know that energy is conserved, a circuit needs a *power supply* or battery, which converts another form of energy into electrical energy, in order for it to operate. The power consumed is the product of the current flowing through an element times the voltage drop across the element

$$P = IV \tag{1.1}$$

Where *P* is the power, *I* is the current, and *V* is the voltage.

IV Characteristic

We can completely characterize any element that has two terminals by its "IV" characteristic (i.e. how much current flows through it when a given voltage is put across it?). The IV characteristic is given by the functions I(V) or V(I) and is frequently defined graphically.

Ground

Ground is the name given to the $V=0$ reference point. This makes it easier to refer to voltages, since you can generally assume that the 2nd point is ground if it is not explicitly stated. We will also use it to represent the implied completion of a circuit, even when we do not explicitly show a wire connection between different places in a circuit, since all ground connections are connected together.

Resistance

A *resistor* is a two-terminal device that converts a voltage into a current or converts a current into a voltage. The current through a resistor is always related to the voltage drop across the resistor by Ohm's Law

$$V = IR \tag{1.2}$$

where R is the resistance (measured in Ohms, Ω) This also means that a resistor generates heat when a current flows given by:

$$P = IV = I^2R = V^2/R. \tag{1.3}$$

Each of these expressions give the same answer but one or the other will be easier to use, depending on what you know about the circuit (I or V, or both).

It may seem silly to have a device that just turns electrical energy into heat, but resistors actually perform many important roles:

1. They turn electrical energy into heat. This can be useful if you want to make an electrical heater, or (more likely) a light bulb. Light is just a form of heat given off by a very hot source. Sometimes you must dissipate power somewhere. For example, if you want to deliver 100 mA current at 12 Volts to part of your circuit, but your power supply only gives you 15 Volts, then the extra power

    $$300\text{mW} = (100\text{mA}) (15\text{V} - 12\text{V})$$

    must go somewhere. You dissipate it in a resistor rather than by heating your sensitive transistors.

2. If you have a voltage source and you want a specific amount of current then a resistor does the job. It converts a voltage difference between two points into a current flowing through the resistor. Sometimes, you will add a resistor in series in a circuit to prevent the power supply from delivering too much current in case you short the output lines together. This is called a *current limiting resistor*.

3. If you have a current flowing and you want to convert it into a voltage then a resistor is again the solution. This might seem a little more far-fetched, because you may not be familiar with constant *current sources*. When we begin studying transistors we will find that they will behave as current sources. So we will use a resistor on the output to convert the current into a voltage.

When we consider capacitors, in week three, we will generalize resistance for AC circuits by allowing for phase shifts. We will call this generalization to the complex plane *impedance*. For this week we will assume resistance and impedance to mean the same thing.

Network Analysis

If you connect one power supply and lots of resistors together in a complicated network, then currents will flow through all the various elements so as to insure that charge is conserved, energy is conserved, and Ohm's Law is satisfied. Simultaneously satisfying all these conditions will give you exactly one solution. We will start with two resistors and then proceed to more complicated systems. Specifically, two resistors in series will draw the same current as an equivalent resistor with resistance. This is stated mathematically as:

$$R_S = R_1 + R_2 \tag{1.4}$$

Alternatively, two resistors in parallel will draw current equivalent a resistor with resistance

$$\frac{1}{R_{//}} = \frac{1}{R_1} + \frac{1}{R_2} \text{ or } R_{//} = \frac{R_1 R_2}{R_1 + R_2} \tag{1.5}$$

Note that the resistance of 2 resistors in parallel is always less than the smaller, but not less than half of that smaller resistance. If the two resistors differ by a large factor, then you can ignore the larger resistor. For example, a 1 kΩ in parallel with 1 MΩ is within 999 Ω or very close to 1 kΩ.

Standard resistors come in a few dozen difference ratings (more on this in lab). So in general one can get any particular value one wants. One can make up an equivalent resistor using a few resistors in series and parallel combinations to get a much larger range of options. In Design Exercise 1-1, we have you design a network to have a specific net resistance given a very limited set of resistances.

Voltage Dividers

By far, the most useful thing you can do with two resistors is to make a *voltage divider*, which is just two resistors in series. Voltage dividers are shown schematically in figure 1 on the right in two slightly different but equivalent representations. It "divides" a total voltage into two parts with each part proportional to the resistance in that leg. So, if you know the starting voltage and the target voltage, then you can calculate the required ratio of the two resistors.

Figure 1.1: Two alternative schematics for a same voltage divider circuit.

$$V_{OUT} = V_{IN} \left(\frac{R_2}{R_1 + R_2} \right) \tag{1.6}$$

In Design Exercise 1-2 you get to derive this relationship.

A Loaded Voltage Divider
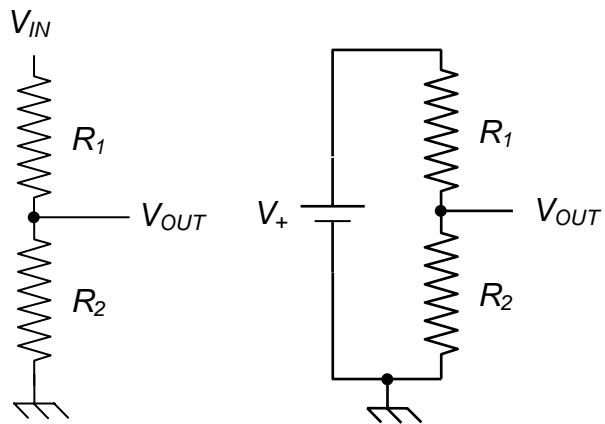
Of course, the way you will use V_{OUT} from a voltage divider is to connect it to something, usually another resistor, as shown in figure 2 on the right. This then changes the equivalent resistance in the lower leg, and therefore changes the voltage across that leg!

In order to choose the actual resistor values, you have two competing concerns: (1) low resistance makes the divider less sensitive to loading when you use the target voltage (i.e. "stiffer"); and (2) high resistance draws less current, and therefore uses less power. We'll spend more time on this and formalize the considerations in the coming weeks.

For now, let's just consider what happens to our voltage divider when we connect a load resistance (R_L) from its output to ground. In this case we can replace the R_2 from the expression for the unloaded divider with an equivalent resistance of R_2 and R_L in parallel. This gives us a voltage across R_L of:
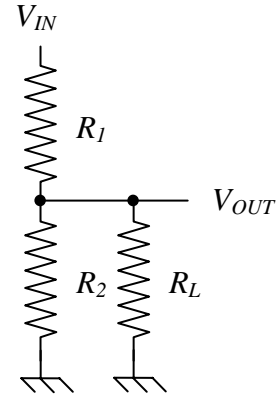
V_{IN}

R_1

V_{OUT}

R_2 R_L

Figure 1.2: A loaded voltage divider.

$$
\begin{aligned}
V_{OUT,LOADED} &= V_{IN} \frac{R_{2//L}}{R_1 + R_{2//L}} \\[2mm]
&= V_{IN} \left(\frac{\dfrac{R_L R_2}{R_L + R_2}}{R_1 + \dfrac{R_L R_2}{R_L + R_2}} \right) \\[2mm]
&= V_{IN} \frac{R_L R_2}{R_1 R_2 + R_L R_1 + R_L R_2} \\[2mm]
&= V_{IN} \frac{R_2}{R_1 + R_2} \frac{R_L}{R_{1//2} + R_L} \\[2mm]
&= V_{UNLOADED} \frac{R_L}{R_{1//2} + R_L}
\end{aligned}
\tag{1.7}
$$

where $R_{2//L}$ is the equivalent resistance of R_2 and R_L in parallel, $R_{1//2}$ is the equivalent resistance of R_1 and R_2 in parallel, and $V_{UNLOADED}$ is the output voltage of the unloaded voltage divider. From the last expression it is clear that the loaded voltage is always less that the loaded voltage. The smaller the load the larger the difference between loaded and unloaded, while large load resisters do not affect the output significantly.

So, if you want to ensure that you do not affect the output of a device, you would like to have the input of your device to look like a large resistor and the output look (in this case $R_{P:1,2}$) look like a small resistance. Next week we'll show that this is a general conclusion and more formally define these concepts.

Design Exercises

Design Exercise 1-1: Use only 1 kΩ resistors to create a network with an equivalent resistance of 667Ω. What is the minimum number of required resistors?

Design Exercise 1-2: Use network analysis and Ohm's Law to derive a formula for V_{OUT} for an unloaded voltage divider.

Design Exercise 1-3: Assuming that R_1, R_2 and R_L are 1 kΩ resistors and V_{IN} is 10V, compute V_{OUT} for both a loaded voltage divider and an unloaded voltage divider. How much does the output voltage change when it is loaded?

Chapter 1: DC circuit basics

Lab 1: Voltage, Current, Resistance and Power

Your notebooks must be complete, understandable, and address all activities, design exercises, observations, and questions noted in the laboratory's procedures. Remember to use your notebook as a laboratory journal and record your data, design calculations, notes and scratch work. ***Make sure to write a conclusion for each exercise and each week.***

1. Measure the resistances of a few resistors with an ohmmeter and see how well they match their specified values. Are they within specifications, based on their tolerance bands?

2. Construct the resistor network you designed in exercise 1-1. Check to see that the total resistance agrees with your calculation. Put 10V into your device and use an ammeter to see if is draws the expected current. How much power is being consumed, and how would one measure it?

3. Make a voltage divider from 1 -100 kΩ resistors and apply input voltages in the range of 2 V – 15 V. Measure the output voltages and the current of this device. Be sure to sketch your experimental setup in your lab notebook! Plot your measurements with the voltage on the vertical axis and the current on the horizontal axis. The *equivalent* or *input resistance* of your voltage divider is the slope of the curve. Relate this to the resistors you used to construct the voltage divider.

4. Make your voltage divider from Exercise 1-3. Measure the output voltage and output current for a variety of load resistances. Note how the output voltage changes and compare your results to your calculations. Do the results make sense?

5. Learn to boot your oscilloscope and perform some basic tests. Feel free to consult the scope's manual to help you with the details as needed.

    a. Select default configuration.

    b. Connect a 10X probe to channel 1, connect the ground to the scopes test ground, and connect the probe tip to the calibration test point on the front of the scope. You should see a square wave appear with a 1 kHz frequency and 5V amplitude. Verify the operation with a 1X probe setting as well.

    c. Adjust the trim screw on the probe until the edges of the square wave are true (instead of rolling over or overshooting).

    d. Test the probe's calibration (i.e. do you get 1 kHz and a 5 V amplitude?).

    e. Zoom in and out on the signal and play with some of the measurements available to you on the scope, (including the FFT feature).

Chapter 1: DC circuit basics

Chapter 2: Kirchhoff's Laws and Thévenin's Theorem

In this course we will be using a variety of mathematical and conceptual models to describe the electrical components and circuits that we will encounter. Ohm's Law was the first such model. In this chapter we will consolidate some of the concepts from last week into a generic model of a linear device known as a **Thévenin equivalent device (input impedance, output impedance, and internal voltage)**.

The Thévenin equivalents will also be generalized and used to describe properties of signals. A **signal** will be described not only by its **voltage** but also by the **output impedance** of the device that produced or transported it. The concept of the **impedance of a signal** will be used extensively through this course as we discover various methods to transform the properties of a signal.

This week we will also examine network analysis in a complete framework known as Kirchhoff's Laws. These rules will allow us to analyze the properties of any combination of resistors and power supplies.

I. Kirchhoff's Laws

If you connect lots of power supplies and resistors together in a complicated network, then currents will flow through all the various elements so as to insure that charge is conserved, energy is conserved, and Ohm's Law is satisfied for each resistor. Simultaneously satisfying all these conditions will give you exactly one solution. The method for writing down equations to represent these conservation laws is called Kirchhoff's Laws. To explore these laws, we will consider the sample circuit shown in figure 2.1 on the right.
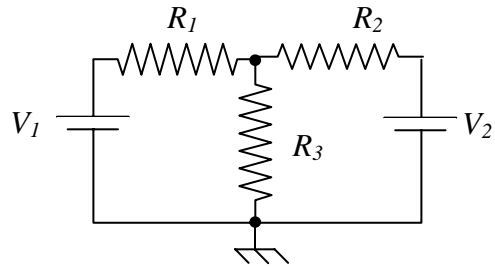
Figure 2.1: A circuit with two power supplies and three resistors.

Kirchhoff's Point Law

Kirchhoff's Point Law says that all of the current that flows into a junction must come out of the junction. This means that charge is conserved – none falls out of the circuit or pools in any location. This concept is illustrated for a junction in the example circuit (see figure 2.2). In this case, we have
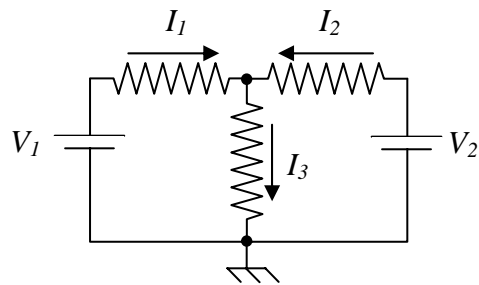
$$I_1 + I_2 - I_3 = 0. \qquad (2.1)$$

Figure 2.2: A choice of current directions for the sample circuit.

Note the sign conventions. We say that a current is positive if we define it to flow into the junction and negative if we define it to flow out the junction.

Kirchhoff's Loop Law

Kirchhoff's Loop Law says that if you sum the voltage drops around any closed loop in a circuit the total must be zero. Since the voltage drops represent the energy per unit charge, this means that all the energy which comes out of the circuit (in heating resistors, for example) must come from some power sources (such as power supplies). This is just conservation of energy.

Figure 2.3: A choice of three Kirchhoff loops.

The sign conventions are that the voltage increases across a power supply and decreases across a resistor in the direction of current flow. The direction of current flow should be the same as the one defined for the Point Law. In our example circuit there are three loops that could be considered, as shown in figure 2.3, above. From these three loops, we obtain the three following relationships:
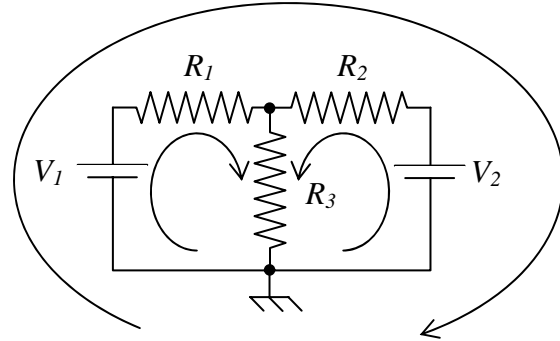
$$0 = V_1 - I_1 R_1 + I_2 R_2 - V_2$$
$$0 = V_1 - I_1 R_1 - I_3 R_3 \quad\quad\quad\quad\quad\quad\quad\quad (2.2)$$
$$0 = V_2 - I_2 R_2 - I_3 R_3$$

Any two of these relationships (since all three are not independent) and the point relationship can be used to solve for the current in each resistor and hence the power in each resistor.

It can be a real nightmare to solve more complicated networks and we will develop models and methods to avoid solving complicated networks wherever we can. We will usually do this by making one part of a circuit relatively independent of another part. Our most common use of Kirchhoff's Laws will usually be to say that *V=IR* and apply the simple relationships for resistors connected in series or parallel. In the next section we will begin to explore how this is possible.

II. Thévenin Equivalents

The Thévenin equivalents are models to describe the input and output properties of a device. We will assume the input of a device looks like a resistor with the other end connected to ground. We will assume that an output looks like an ideal voltage source followed by a resistor. Both of these are shown in figure 2.4, below. Note that for this week's discussion the term resistance and impedance are treated as synonymous.

One can determine the input impedance (Z_{IN}) by simply applying a voltage to the input, measuring the current flowing into the device, and applying Ohm's Law.
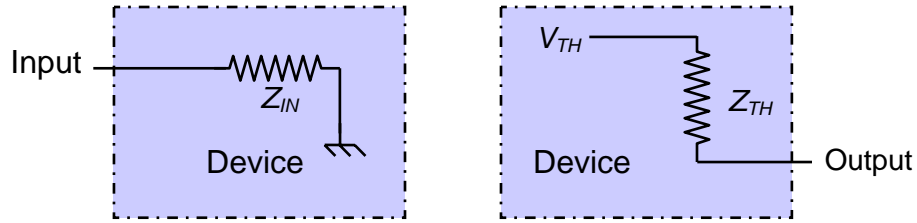
Figure 2.4: (left) The Thévenin equivalent circuit for an input device, and (right) the Thévenin equivalent circuit for a voltage source.

Given any electrical device with two output terminals (we will assume one is called ground) there are a number of you might do to try to determine its properties:

1. Measure the voltage across the terminals.

2. Attach a resistor across the terminals and then measure the voltage.

3. Change the resistor and measure the voltage again, repeat.

Clearly, these are all variations on one simple measurement. What is the minimum number of measurements you must do to characterize this black box completely? Thévenin's answer is just two!

Thévenin's Theorem

Any linear (i.e. Ohmic) system with two terminals can be completely characterized as an ideal voltage source (V_{TH} or Thévenin voltage) in series with one resistor (Z_{TH} or Thévenin resistance, and also called internal resistance sometimes). No matter what resistance (R_{LOAD}) you connect across the two terminals, it just forms a voltage divider, as shown in fiure 2.5 on the right, and so the voltage across that load resistor is given by:

Figure 2.5: A voltage source connected to a load, can be treated as voltage divider

$$V_{LOAD} = V_{TH}\left(\frac{R_{LOAD}}{Z_{TH} + R_{LOAD}}\right). \qquad (2.3)$$

A special case is when there isn't any load applied (i.e. option 1 above, where $R_{LOAD} = \infty$) then the output is simply V_{TH}. From the results of these two measurements we can solve for R_{TH}.
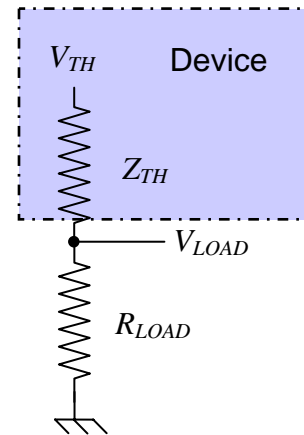
Voltage Divider as a Thévenin Device

Let's consider what happens to our voltage divider from last week when we connect a load resistance to its output. From last week we computed the output of a loaded voltage divider was given by:

$$V_{LOAD} = V_{IN} \frac{R_2}{R_1 + R_2} \frac{R_{LOAD}}{R_{1//2} + R_{LOAD}}. \tag{2.4}$$

If we equate factors from in this relationship with the ones in the previous section, we can see that for a loaded voltage divider produces Thévenin equivalents of

$$V_{TH} = V_{IN} \frac{R_2}{R_1 + R_2} \text{ and } Z_{TH} = R_{1//2} = \frac{R_1 R_2}{R_1 + R_2}. \tag{2.5, 2.6}$$

The relationship for V_{TH} makes sense. It is simply the unloaded output voltage as describe in the previous section. The output impedance is simply the parallel equivalent of the two resistors on the voltage divider.

Designing a Stiff Voltage Divider

A *stiff* output implies that the output voltage stays relatively close to its Thévenin, or unloaded, voltage when a load is implied. A rule of thumb for designing a stiff voltage source is that the output should not deviate by more than about 10% when loaded. From the relationships in the previous section, we can see that this implies that the load resistance must be more than a factor of 10 larger than the output impedance (in the voltage divider case $R_{TH} = R_{1//2}$). We can think of this at a *Rule of 10.* We will usually design it so that R_1 itself is smaller than the expected load by a factor of 10.

Generalization and Application of the Rule of 10 and Notable Exceptions

Since we are going to design and construct some very complicated circuits this year, we need to be able to focus on parts of circuits, which we will call *sub-circuits*. For example, we will often build amplifiers in three stages. The basic idea is to always insure that the downstream elements do not load the upstream elements. (1) The first stage will be a "buffer" amplifier that takes a signal and amplifies the current but not the voltage. This will allow us to connect it to the next part of the amplifier without changing (*i.e.* loading) the characteristics of whatever generated the signal. The second stage will be a voltage amplifier that amplifies a signal's voltage to whatever level we need. The final stage will be another buffer amplifier that again amplifies the current so that we will not load our voltage amplifier when we use its output.

This gains us two major advantages: We will never have a large number of interdependent simultaneous equations to solve, and our circuits will work under a wide range of differing operational conditions.

Here is the general procedure:

- Any sub-circuit can be modeled, using Thévenin's Theorem, as an ideal voltage source (V_{TH}) in series with a resistance (R_{TH}).

- When we connect this to the next sub-circuit we can represent that by a load resistance (R_{LOAD}) thus making a voltage divider.

- We can keep our equations simple, so that we need not employ the full Kirchhoff's Laws formalism, simply by following a "factor of 10" rule.

In summary, anything we design to produce a reliable voltage should have R_{TH} smaller R_{LOAD} by least a factor of 10. That is, we do not want the voltage to change significantly when the sub-circuit is connected to subsequent parts of the circuit.

Constant Voltage Sources

The most common sub-circuit we will use will supply a voltage to a later part of a circuit. It may be a power supply to power all the other sub-circuits, or it may be a biasing network to keep a transistor in its normal operating region. For these type sub-circuits, we want V_{OUT} to be relatively independent of R_L, so this means $R_{LOAD} > 10R_{TH}$.

Constant Current Sources

We will later see that transistors are current amplifiers. This means that we will often want to drive an amplifier with a current source. A current source can also be modeled as ideal voltage source in series with an R_{TH}. Again, the complete circuit will look like an ideal voltage source driving a voltage divider, but we want the current through R_L to be relatively independent of R_L. This means we want the same current as when we remove R_L and just connect (or "short") the two leads together. By applying the same logic above this implies that we want $R_{LOAD} < 10R_{TH}$ for a stiff *current source*.

Impedance Matching

There are two cases when we require $R_{LOAD} = R_{TH}$, which we will call "impedance matching". If you want the most efficient transfer of power to your load, then you must choose $R_{LOAD} = R_{TH}$. By the most efficient transfer, we mean that

$$I^2 R_L = \left(\frac{V_{IN}}{R_L + R_{Th}} \right)^2 R_L$$

(2.7)

will have its largest value when $R_{LOAD} = R_{TH}$. Of course this will severely load the source, dropping the output voltage by exactly a factor of two. This may, however, be fine in simple circuits – like lighting a light-bulb or driving an audio speaker.

The second case is when you have very fast signals traveling along transmission cables such as the BNC cables that we often use to connect laboratory equipment. Then, we will want to prevent reflections at our connections, since these could give rise to phony signals. Then we will want to choose $R_{LOAD} = R_{TH}$, even though it will decrease the amplitude of the signal. We often accomplish this matching by attaching a 50Ω *terminator* to the end of our BNC cable to ensue that the impedance is matched.

Design Exercises

Design Exercise 2-1: Use Kirchhoff's laws to determine the resistance of the Wheatstone bridge depicted in the circuit in figure 2.6 on the right.
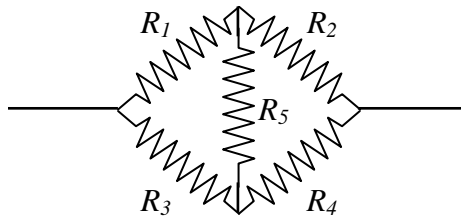

Figure 2.6: A Wheatstone bridge circuit.

Design Exercise 2-2: Find a relationship between V_{out}, I_{out}, and Z_{TH} that does not depend on R_{LOAD} in the Thévenin equivalent circuit of figure 2.7 on the right.

Design Exercise 2-3: How would you measure the Thévenin resistance and voltage of a power supply?

Design Exercise 2-4: Prove that the power transfer from a power supply with Thévenin resistance, R_{Th}, to a load resistor, R_{Load}, is maximum when $R_{Th}=R_{Load}$.
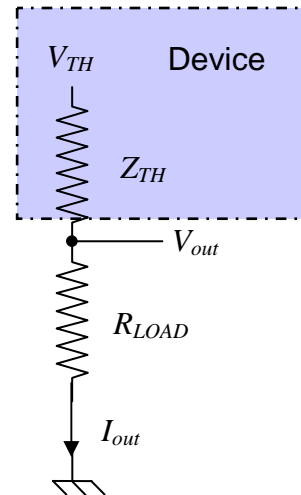
Figure 2.7: A voltage source with load resistor.

Lab 2: Kirchhoff, Thévenin, and Impedance Matching

1. Construct a Wheatstone bridge from 5 resistors in the 1–10 kΩ range. Measure its resistance at a few voltages (i.e. measure current and voltage). What is its resistance? Does it agree with your calculation from design exercise 2-1?

2. Construct a voltage divider similar to the one you made last week for lab exercises 1-3 and 1-4. Set V_{in} to 10 V and measure V_{TH} and Z_{TH} seen by a load resistor – you will not be able to measure Z_{TH} directly, so you will have to come up with a method for doing so (hint: the result of design exercise 2-2 can be usefully adapted to make a measurement). Do the measured V_{TH} and Z_{TH} agree with what you expect from your calculations?

3. Use your voltage divider setup from part 2. Determine experimentally the load resistance which results in the maximum output power out of the voltage divider.

4. Set the breadboard power supply to 3 V and measure V_{TH} and Z_{TH} of the breadboard power supply for this setting. Before doing your measurements, you should list the potential difficulties of such an experiment. You should also consult with the instructors before attempting it -- please do not let I_{out} exceed 1.5 A. Comment on the engineering of the power supply.

Chapter 3: Capacitors, Inductors, and Complex Impedance

In this chapter we introduce the concept of complex resistance, or impedance, by studying two reactive circuit elements, the capacitor and the inductor. We will study capacitors and inductors using differential equations and Fourier analysis and from these derive their impedance. Capacitors and inductors are used primarily in circuits involving time-dependent voltages and currents, such as AC circuits.

I. AC Voltages and circuits

Most electronic circuits involve time-dependent voltages and currents. An important class of time-dependent signal is the sinusoidal voltage (or current), also known as an AC signal (Alternating Current). **Kirchhoff's laws and Ohm's law still apply (they always apply), but one must be careful to differentiate between time-averaged and instantaneous quantities.**

An *AC voltage* (or signal) is of the form:

$$V(t) = V_p \, cos(\omega t) \tag{3.1}$$

where ω is the angular frequency, V_p is the amplitude of the waveform or the *peak voltage* and t is the time. The angular frequency is related to the frequency (*f*) by $\omega = 2\pi f$ and the period (*T*) is related to the frequency by *T=1/f*. Other useful voltages are also commonly defined. They include the *peak-to-peak voltage* (V_{pp}) which is twice the amplitude and the *RMS voltage* (V_{RMS}) which is $V_{RMS} = V_p / \sqrt{2}$. Average power in a resistive AC device is computed using RMS quantities:

$$P = I_{RMS}V_{RMS} = I_p V_p/2. \tag{3.2}$$

This is important enough that voltmeters and ammeters in AC mode actually return the RMS values for current and voltage.

While most real world signals are not sinusoidal, AC signals are still used extensively to characterize circuits through the technique of Fourier analysis.

Fourier Analysis

One convenient way to characterize the rate of change of a function is to write the true function as a linear combination of a set of functions that have particularly easy characteristics to deal with analytically. In this case we can consider the trigonometric functions. It turns out that we can write any function as an integral of the form

$$V(t) = \int \widetilde{V} \cos(\omega t + \phi) d\omega \tag{3.3}$$

where $\widetilde{V}$ and ϕ are functions of the frequency ω. This process is called *Fourier analysis*, and it means that any function can be written as an integral of simple sinusoidal

functions. In the case of a periodic waveform this integral becomes a sum over all the harmonics of the period (i.e. all the integer multiplicative frequencies of the period).

$$V(t) = \sum_n A_n \cos(n\omega t + \varphi_n) \tag{3.4}$$

An implication of this mathematical fact is that **if we can figure out what happens when we put pure sinusoidal voltages into a linear circuit, then we will know everything about its operation even for arbitrary input voltages.**

Complex Notation

In complex notation we replace our sinusoidal functions by exponentials to make the calculus and bookkeeping easier still. Then we can include both phase and magnitude information. We'll define

$$e^{i\varphi} \equiv \cos\varphi + i\sin\varphi, \tag{3.5}$$

where $i^2 \equiv -1$.

The general procedure for using this notation is:

1. Change your problem into complex algebra (i.e. replace $\cos\omega t$ with $e^{i\omega t}$)
2. Solve the problem.
3. Take the real part of the solution as your answer at the end.

II. Capacitors

One of the most basic rules of electronics is that circuits must be complete for currents to flow. This week, we will introduce an exception to that rule.

The capacitor is actually a small break in a circuit. Try measuring the resistance of a capacitor, you will find that it is an open circuit. However, at the inside ends of the capacitor's lead, it has little plates that act as charge reservoirs where it can store charge. For short times, you do not notice that the break is there. Negative charge initially flows in to one side and out from out the other side just as if the two leads were connected. For fast signals, the capacitor "looks" like a short-circuit. But after a while the capacitor's reservoirs fill, the current stops, and we notice that there really is a break in the circuit.

For slow signals, a capacitor "looks" like an open circuit. What is fast, and what is slow? It depends on the capacitor and the rest of the circuit. This week, you will learn how to determine fast and slow for yourselves.

Capacitors serve three major roles in electrical circuits (although all three are just variations of one basic idea):

*   Charge integrators;

*   High or low frequency filters;

*   DC isolators.

In order to perform these functions analytically, we will need to introduce a number of new concepts and some significant mathematical formalism. In this process we will also develop a number of new concepts in analyzing electronic circuits.

Capacitance

A capacitor is a device for storing charge and electrical energy. It consists of two parallel conducting plates and some non-conducting material between the plates, as shown in figure 3.1 on the right. When voltage is applied positive charge collects on one plate and negative charge collects on the other plane. Since they are attracted to each other this is a stable state until the voltage is changed again.

A capacitor's charge capacity or *capacitance* (*C*) is defined as:

$$Q=CV \qquad (3.6)$$

which relates the charge stored in the capacitor (*Q*) to the voltage across its leads (*V*). Capacitance is measured in Farads (F). A Farad is a very large unit and most applications use μF, nF, or pF sized devices. Many electronics components have small parasitic capacitances due to their leads and design.

Figure 3.1: A capacitor consist of two parallel plates which store equal and opposite amounts of charge

The capacitor also stores energy in the electric field generated by the charges on its two plates. The potential energy stored in a capacitor, with voltage V on it, is

$$E = \frac{1}{2}CV^2 \qquad (3.7)$$

We usually speak in terms of current when we analyze a circuit. By noting that the current is the rate of change of charge, we can rewrite the definition of capacitance in terms of the current as:

$$V = \frac{1}{C}Q = \frac{1}{C}\int I dt \qquad (3.8)$$

or

$$I = C\frac{dV}{dt} = C\dot{V} \qquad (3.9)$$

This shows that we can integrate a function *I(t)* just by monitoring the voltage as the current charges up a capacitor, or we can differentiate a function *V(t)* by putting it across a capacitor, and monitoring the current flow when the voltage changes.
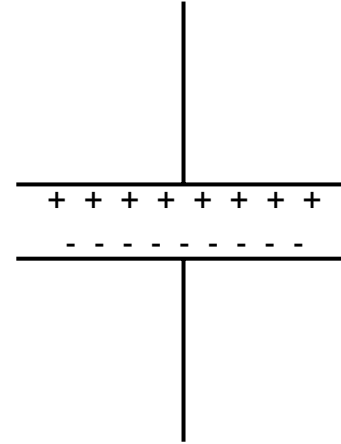
A Simple RC Circuit

We will start by looking in detail at the simplest capacitive circuit, which is shown in figure 3.2 on the right. An *RC* circuit is made by simply putting a resistor and a capacitor together as a voltage divider. We will put the resistor in first, so we can connect the capacitor to ground.

By applying Kirchhoff's Laws to this circuit, we can see that:

1. The same current flows through both the resistor and the capacitor, and

2. The sum of the voltage drops across the two elements equal the input voltage.

Figure 3.2: A simple RC circuit which integrates current.

This can be put into a formula in the following equation:

$$V_{IN} = IR + \frac{1}{C}\int I dt .$$

(3.10)

which can also be written as

$$\frac{V_{IN}}{R} = I + \frac{1}{RC}\int I dt .$$

(3.11)

We can also put this into the form of a differential equation in the following way:

$$\frac{dV_{IN}}{dt} = R\frac{dI}{dt} + \frac{I}{C}$$

(3.12a)

or

$$C\dot{V}_{IN} = RC\dot{I} + I .$$

(3.12b)

These equations show that times are measured in units of *RC*, and that what you see depends on how quickly things change during one *RC* time interval.

If the current changes quickly, then most of the voltage will show up across the resistor, while the voltage across the capacitor slowly charges up as it integrates the current. If the voltage changes slowly, then most of the voltage shows up across the capacitor as it charges. Since this usually requires a small current, the voltage across the resistor stays small.

But, what happens at intermediate times? To determine this quantitatively we will have to develop some more sophisticated mathematical techniques.

Solutions to RC Circuit

Rather than produce the general solution, we will concentrate on two special cases that are particularly useful. The first will be for a constant voltage and the second will be a sinusoidal input.
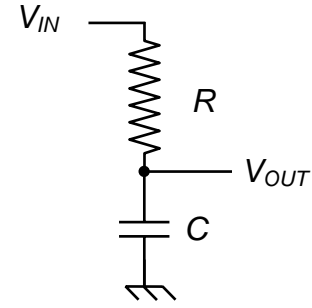
To study a constant supply voltage on an RC circuit, we set the left side of equation 3.12 equal to a constant voltage. Then we have a simple homogeneous differential equation with the simple solution for the current of a decaying exponential,

$$I = I_0 e^{-t/(RC)}, \tag{3.13}$$

which will account for any initial conditions. After a time of a few RC time periods, this solution will have decayed away to the supply voltage.

And now let us consider the other solution. In the prior section, we argued that if we can understand the *RC* circuit's behavior for sinusoidal input we can deal with any arbitrary input. Therefore, this is the important one.

Let's look at our simple *RC* circuit and suppose that we apply (or drive) a simple sine wave into the input:

$$V_{IN} = V_0 \cos(\omega t). \tag{3.14}$$

In complex notation, this means that we will set the drive voltage to

$$V_{IN} = V_0 \exp(i\omega t), \tag{3.15}$$

and we just have to remember to take the Real part at the end of our calculation.

If we put this drive voltage into the differential equation (equation 3.12), then it becomes a relatively simple inhomogeneous differential equation:

$$C \frac{dV_{IN}}{dt} = CV_0 i\omega \exp(i\omega t) = RC \frac{dI}{dt} + I . \tag{3.16}$$

This is relatively simple because it shows up so often in physics that you might as well memorize the solution or at least the way to get the solution. Note that mathematically it looks just like a driven harmonic oscillator.

We can obtain the solution by using the standard recipe for first order linear differential equations. We start by rewriting equation as

$$\frac{dI}{dt} + \frac{1}{RC} I = \frac{i\omega V_0}{R} \exp(i\omega t) , \tag{3.17}$$

which we then multiply by $\exp(t/RC)$ to obtain

$$\exp(t/RC) \frac{dI}{dt} + \frac{\exp(t/RC)}{RC} I = \frac{i\omega V_0}{R} \exp[(i\omega + \frac{1}{RC})t] . \tag{3.18}$$

The left hand-side of this equality can be rewritten under the form of a total derivative (multiplication rule) so that we now have

$$\frac{d}{dt}\left[I(t)\exp(\frac{t}{RC}) \right] = \frac{i\omega V_0}{R} \exp[(i\omega + \frac{1}{RC})t] . \tag{3.19}$$

This equation is easily integrable and can be rewritten as

$$I(t)\exp(\frac{t}{RC}) = \frac{i\omega V_0}{R} \int \exp[(i\omega + \frac{1}{RC})t]dt . \tag{3.20}$$

The integral is straightforward and yields the following expression:

$$I(t) = \frac{i\omega V_0}{R} \frac{1}{\frac{1}{RC} + i\omega} \exp(i\omega t) + Cst \cdot \exp(-\frac{t}{RC}) \tag{3.21}$$

The first term represent the "steady state" oscillatory behavior of the driven circuit, while the second term describes the transient behavior of the current after switching on the driving voltage. Since we are only interested in the long-term behavior of the circuit, we neglect the second term and concentrate on the first. After a little bit of algebra, we can rewrite the steady-state current as

$$I(t) = \frac{i\omega V_0}{R} \frac{1}{\frac{1}{RC} + i\omega} \exp(i\omega t) = \frac{\omega V_0 C}{\sqrt{1 + (\omega RC)^2}} \frac{\omega RC + i}{\sqrt{1 + (\omega RC)^2}} \exp(i\omega t) \tag{3.22}$$

The second fraction can be interpreted as a phase term with $\tan\phi = \dfrac{1}{\omega RC}$, so that the expression for the current becomes

$$I(t) = I_0 \exp(i\omega t + \phi) \tag{3.23}$$

with

$$I_0 = \frac{\omega V_0 C}{\sqrt{1 + (\omega RC)^2}} = \frac{V_0}{R} \cos(\phi) \tag{3.24}$$

The real solution of this simple RC circuit can be obtained by taking the real part of equation 23, and is left as an exercise to the reader.

The solution of the simple RC circuit appears to be rather complicated and involved, however it simplifies considerably when we plug equation 23 back in to the original integral equation from Kirchhoff's loop law (equation 10). After integrating the exponential and a little bit of algebra, we obtain

$$V_{in}(t; \omega) = I(t)R + I(t)\frac{1}{i\omega C} \tag{3.25}$$

This remarkably simple expression looks a lot like the standard Kirchhoff's loop law for resistors, except that the capacitor term behaves with a frequency dependent "imaginary" resistance.

RC Impedance

We will obtain the same solution as the one we obtained for the original voltage divider, as long as we assign an imaginary, frequency dependent, resistance to the capacitor. The "imaginary" part just means that it will produce a $\pi/2$ phase shift between the voltage and the current for a sinusoidal input. We will call this impedance

$$Z_C = \frac{1}{i\omega C}. \tag{3.26}$$

Now, the solution for an *RC* divider becomes somewhat simplified. We can compute the total current flowing through the circuit as

$$I = \frac{V_{in}}{Z_{tot}} = \frac{V_{in}}{R + Z_C} = \frac{V_0 e^{i\omega t}}{R + 1/(i\omega C)} = \frac{i\omega C V_0 e^{i\omega t}}{1 + i\omega RC} = \frac{V_0}{R}\frac{i\omega RC_0 e^{i\omega t}}{(1 + i\omega RC)} = \frac{V_0}{R}\cos(\phi)e^{i(\omega t+\phi)} \quad (3.27)$$

The voltage across an element is just this current times the element's impedance. For the voltage drop across the resistor it is largely the same as before:

$$V_R = IR = V_0 \cos(\phi)e^{i(\omega t+\phi)}. \tag{3.28}$$

For the capacitor, we get the following voltage drop:

$$V_C = IZ_C = I\frac{1}{i\omega C} = \frac{V_0 \cos(\phi)e^{i(\omega t+\phi)}}{i\omega RC} = -iV_0 \sin(\phi)e^{i(\omega t+\phi)}$$
$$= V_0 \frac{\cos(\phi)}{\omega RC}e^{i(\omega t+\phi-\pi/2)} = V_0 \sin(\phi)e^{i(\omega t+\phi-\pi/2)} \tag{3.29}$$

If everything is correctly calculated then the sum of the voltage drops across the two elements should be equal to the input voltage. Let's try it:

$$V_R + V_C = V_0 \left(\cos(\phi) - i\sin(\phi)\right)e^{i(\omega t+\phi)} = V_0 e^{-i\phi}e^{i(\omega t+\phi)} = V_0 e^{i\omega t} \tag{3.30}$$

Remember, you get the actual waveforms by taking the real parts of these complex solutions. Therefore

$$V_R = V_0 \cos\phi \cos(\omega t+\phi) \text{ and} \tag{3.31}$$

$$V_C = V_0 \sin\phi \cos(\omega t+\phi-\pi/2) = -V_0 \sin\phi \sin(\omega t+\phi) \tag{3.32}$$

This looks complicated, but the limits of high frequency and low frequency are easy to remember. At high frequencies ($\phi \to 0$), the capacitor is like a short, and all the voltage shows up across the resistor. At low frequencies ($\phi \to \pi/2$), the capacitor is like an open circuit, and all the voltage shows up across the capacitor.

If you consider the leading terms for the elements with the small voltages, you find that

$$V_C = V_0 \frac{(1 - i\omega RC)}{1 + (\omega RC)^2} \to -\frac{i}{\omega}\left(\frac{V_0}{RC}\right) \text{ as } \omega \to \infty$$

$$V_R = V_0 \frac{\omega RC(i + \omega RC)}{1 + (\omega RC)^2} \to i\omega(RCV_0) \text{ as } \omega \to 0 \tag{3.33}$$

Thus, at high frequency, the voltage across the capacitor is the integral of the input voltage, while at low frequency the voltage across the resistor is the derivative of the input voltage.

This says that as long as all the important frequencies are high, the capacitor will integrate the input voltage. If all the important frequencies are small, the resistor will differentiate the voltage. If there are intermediate frequencies, or a mixture of some high

and some low frequencies, the result will not be so simple but it can be determined from the voltage divider algebra using complex notation.

We finish by noting that the voltage on the capacitor is always **-π/2** out of phase with the voltage on the resistor.

III. Inductors

An inductors is a coil of wire, or solenoid, which can be used to store energy in the magnetic field that it generates (see figure 3.3 on the right). It is mathematically similar to a capacitor, but has exactly the opposite behavior: it behaves as a short circuit for low frequencies and as an open circuit for high frequencies (i.e. it passes low frequency signals and blocks high frequency signals).

The energy stored in the field of an inductor with inductance L is given by the following formula:

$$E = \frac{1}{2}LI^2 \qquad (3.34)$$

The SI unit of inductance is the Henry (H). Commercially available inductors have inductances that range from nH to mH. Small millimeter-size and centimeter size solenoids typically have inductances in the range of μH, while magnetic field coils can have a inductances in the mH range, and can sometimes have inductances of up to several H. Most electronics components have small parasitic inductances due to their leads and design (for example, wire-wound power resistors).

Figure 3.3: An inductor consists of a coiled wire, also called a solenoid. The dashed arrow "B", represent the magnetic field generated by the current in the inductor.

In an electric circuit, a voltage, or electromotive potential, is generated across the terminals of the inductor when the current changes due to Faraday's law. The voltage drop is given by the following simple expression:

$$V = L\frac{dI}{dt} \qquad (3.35)$$

From this equation, we see that the inductor operates exactly opposite to a capacitor: an inductor differentiates the current and integrates the voltage.

The LR circuit

We can analyze the LR circuit in much the same way that we derived the operation of the RC circuit. We start by applying Kirchhoff's loop law to the LR circuit in figure 3.4 below, and we find that

$$V_{IN} = IR + L\frac{dI}{dt} . \qquad (3.36)$$

If we apply a constant voltage the solution can be calculated using the techniques developed for the RC circuit and we calculate that

$$I(t) = I_0\left(1 - \exp(-\frac{R}{L}t)\right). \qquad (3.37)$$

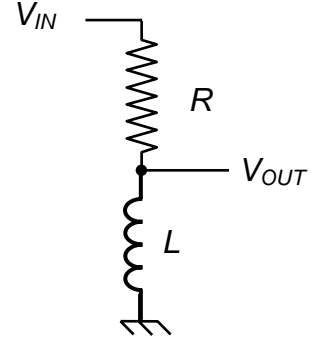The circuit approaches the steady state current $I_0 = V_{IN}/R$ with a time constant of L/R.

Figure 3.4: A simple LR circuit.

LR impedance

Instead of solving the differential equation for the LR circuit with a sinusoidal applied input voltage such as that given by equations 14 and 15, as we did with the RC circuit, we will just assume that the current has the form

$$I(t) = I_0 \exp(i\omega t + \phi) \qquad (3.38)$$

We plug this ansatz solution back into the differential equation of equation 31 and find that

$$V_{IN} = I(t)R + i\omega L I , \qquad (3.39)$$

from which we deduce that the inductor behaves as a resistor with frequency dependent "imaginary" resistance. The impedance of an inductor is therefore

$$Z_L = i\omega L \qquad (3.40)$$

Just as with the RC circuit, we can apply Ohm's law to the circuit to calculate the total current. Since R and L are in series, we obtain

$$I(t) = \frac{V_{in}}{Z_{total}} = \frac{V_0 e^{i\omega t}}{R + Z_L} = \frac{V_0 e^{i\omega t}}{R + i\omega L} = \frac{V_0}{R}\frac{1 - i\omega\frac{L}{R}}{1 + \left(\omega\frac{L}{R}\right)^2}e^{i\omega t} = \frac{V_0}{R}\cos(\phi)e^{i(\omega t - \phi)} \qquad (3.41)$$

where the phase is given by $\tan(\phi) = \omega\frac{L}{R}$. We calculate the voltage drop across the resistor using the expression for the current and find that

$$V_R = I(t)R = V_0 \cos(\phi)e^{i(\omega t - \phi)} \qquad (3.42)$$

The voltage drop across the inductor is calculated the same way, and we find

$$V_L = i\omega L I(t) = i\omega L\frac{V_0}{R}\cos(\phi)e^{i(\omega t - \phi)} = iV_0 \sin(\phi)e^{i(\omega t - \phi)} = V_0 \cos(\phi)e^{i(\omega t - \phi + \pi/2)} \qquad (3.43)$$

If everything is correctly calculated then the sum of the voltage drops across the two elements should be equal to the input voltage. Let's try it:

$$V_R + V_L = V_0\left(\cos(\phi) + i\sin(\phi)\right)e^{i(\omega t - \phi)} = V_0 e^{i\phi} e^{i(\omega t - \phi)} = V_0 e^{i\omega t} \qquad (3.44)$$

You get the actual waveforms by taking the real parts of these complex solutions. Therefore

$$V_R = V_0\cos\phi\,\cos(\omega t\text{-}\phi) \text{ and} \qquad (3.45)$$

$$V_L = V_0\sin\phi\,\cos(\omega t\text{-}\phi\text{+}\pi/2) = V_0\sin\phi\,\sin(\omega t\text{-}\phi) \qquad (3.46)$$

This looks complicated, but the limits of high frequency and low frequency are easy to remember. At high frequencies ($\phi \to \pi/2$), the inductor is like an open circuit, and all the voltage shows up across the inductor. At low frequencies ($\phi \to 0$), the inductor is like a short circuit or just a plain wire, and all the voltage shows up across the resistor.

It should also be pointed out that the voltage on the inductor is always **+π/2** out of phase with the voltage on the resistor.

IV. Transformers

Transformers are an ingenious combination of two inductors. They are used to transfer power between two circuits by magnetic coupling. The transformer changes an input voltage, without affecting the signal shape, similar to the voltage divider of last week. However it has several important differences:

- It can increase as well as decrease a signal's amplitude (i.e. AC voltage).

- It requires a time-varying (AC) input to work.

- It is much harder to fabricate.

- It usually does not work well for very fast signals (since inductors block high frequencies).
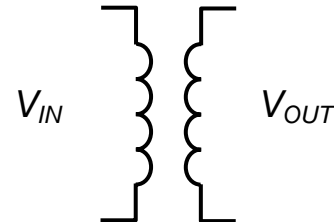
Figure 3.5: The schematic symbol for a transformer.

Transformers are commonly used as a major component in a DC power supplies since they can convert a 120 V AC wall voltage into a smaller voltage that is closer to the desired DC voltage (e.g. 5 V or ±15 V). The schematic symbol for a transformer is shown in figure 3.5, above.

Transformers are passive devices that simultaneously change the voltage and current of a circuit. They have (at least) four terminals: two inputs (called the primary) and two outputs (called the secondary). There is no real difference between the input and output for a transformer, you could simply flip it around and use the secondary as the input and the primary as the output. However, for the sake of clarity, we will always assume that you use the primary for input and the secondary for output.

The coupling between the input and output is done magnetically. This allows transformers to have a number of interesting benefits including:

- There is no DC connection between input and output, so transformers are often used to isolate one circuit from another.

- Transformers only work for time varying signals, when the inductive coupling between the coils is greater than the resistive losses.

Since they have no external power the output power can not be greater than the input power

$$P = V_P I_P \geq V_S I_S.$$ (3.47)

Usually, we will assume equality but there are small resistances (and hence resistive losses) in the coils and a poorly or cheaply designed transformer many not have the input and output sufficiently strongly coupled to each other. Depending on the device and the signal the output power may well be less than the input power.

Transformers are most commonly used to change line voltage (120 V RMS at 60 Hz) into a more convenient voltage. High power transmission lines use transformers to increase the voltage and decrease the current. This reduces $I^2 R$ power losses in the transmission wires. For our circuits we will use a transformer that reduces the voltage and increases the current.

Transformers are characterized by the ratio of the number of turns on the input and output windings. The magnetic coupling in an ideal transformer will insure that the number of turns times the current flowing is the same for the input and output:

$$N_P I_P = N_S I_S \Rightarrow \frac{I_S}{I_P} = \frac{N_P}{N_S}$$ (3.48)

Since the voltage must change in the opposite manner to keep the input and output power, the ratio of the voltages is the same as the ratio of the turns:

$$\frac{V_S}{V_P} = \frac{N_S}{N_P}$$ (3.49)

Transformers are usually called step-up or step-down according to whether the output voltage increases or decreases.

A transformer also transforms the impedance of a circuit, since it changes the ratio of V/I. Using our rules above, the ratio of output impedance to input impedance is the square of the ratio of turns:

$$\frac{Z_S}{Z_P} = \frac{V_S}{I_S} \frac{I_P}{V_P} = \left(\frac{N_S}{N_P} \right)^2$$ (3.50)

So, if you use a transformer as a step-up transformer, it increases the voltage and the impedance at its output relative to its input. If you use a transformer as a step-down transformer, it decreases the voltage and the impedance at its output.

Design Exercises

Design Exercise 3-1: Using Kirchhoff's laws, derive a formula for the total capacitance of two capacitors in parallel and a formula for the total capacitance of two capacitors in series.

Design Exercise 3-2: Using Kirchhoff's laws, derive a formula for the total inductance of two inductors in parallel and a formula for the total inductance of two inductors in series.

Design Exercise 3-3: Calculate V_{out} as a function of V_{in} in the RLC circuit of figure 3.6 on the right, using the formulas for Z_R, Z_C, and Z_L (do not use Maple / Mathematica / MATLAB / MathCad for these calculations and show all steps). V_{in} is a perfect AC voltage signal with a frequency of ω.

Plot the magnitude and phase of V_{out} as a function of ω for R = 1 kΩ, C = 1 µF, and L = 10 µH. What happens to the magnitude and the phase of V_{out} at $\omega = 1/\sqrt{LC}$?

(Maple / Mathematica / MATLAB / MathCad are permitted for the plots.)
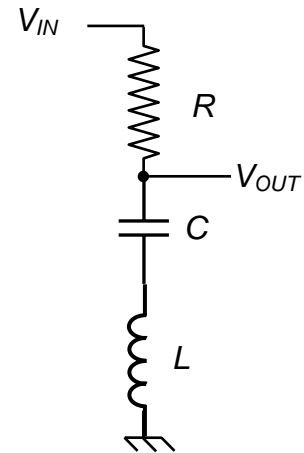
Figure 3.6: An RLC filter circuit.

Design Exercise 3-4: Calculate V_{out} as a function of V_{in} in the RLC circuit depicted on the right, using the formulas for Z_R, Z_C, and Z_L (do not use Maple / Mathematica / MATLAB / MathCad for these calculations and show all steps). V_{in} is a perfect AC voltage signal with a frequency of ω.

Plot the magnitude and phase of V_{out} as a function of ω for R = 1 kΩ, C = 1 µF, and L = 10 µH. What happens to the magnitude and the phase of V_{out} at $\omega = 1/\sqrt{LC}$?

(Maple / Mathematica / MATLAB / MathCad are permitted for the plots.)
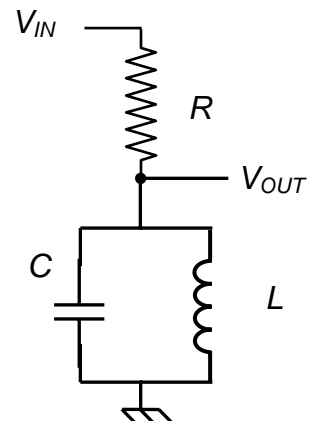
Figure 3.7: Another RLC filter circuit.

Lab 3: AC signals, Complex Impedance, and Phase

Section 1: Introduction to transformers

In this section, we use a transformer to change the impedance of an AC signal.

1a. Measure the **output impedance** of a signal generator with a 0.5V amplitude sinusoid output of 1 kHz and the input impedance of a speaker. Remember you are using AC signals. What does an AC current reading from a DVM mean in terms of the waveform? How do you measure current with an oscilloscope? Check this with the oscilloscope.

1b. Connect the signal generator to a speaker and measure the signal amplitude with, and without connecting to the speaker. The voltage drops so much because of the impedance mismatch. Measure the power into the speaker.

1c. Use a transformer to decrease the output voltage, while increasing the output current into the speaker. Measure V_{out}, V_{in}, I_{in}, and I_{out}. How well does the transformer transmit power? Does $V_{out}/V_{in} = I_{in}/I_{out}$? Estimate the ratio of primary turns to secondary turns?

1.d Measure the **output impedance** of the signal generator plus transformer circuit. Does the measured value agree with what you expect theoretically?

Section 2: The RC circuit

In this section, we take a first look at the classic RC circuit and the concept of phase.

2a. Get two capacitors and measure their individual capacitances. Measure the total capacitance with a capacitance meter when they are in series, and when they are in parallel. Do you get good agreement with what you expect?

2b. Construct the RC circuit to the right, with component ranges R=1-10 kΩ and C=0.001-0.01 µF. Set the function generator at approximately ω=0.1/RC with a square wave and describe what you see. Measure the time constant of the exponential and use it to determine the capacitance of C (R should be determined with a multimeter).

2c. (Same set-up) Set the function generator to sinusoidal output at ω=1/RC and measure the magnitude of V_{in} and V_{out}. Do you get what you expect ? Measure the phase of V_{out} with respect to V_{in} and make a Lissajou plot of V_{out} and V_{in}.
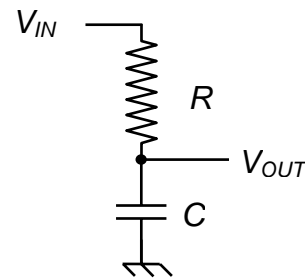
Figure 3.8: An RC filter circuit.

Chapter 4: Passive Analog Signal Processing

In this chapter we introduce filters and signal transmission theory. Filters are essential components of most analog circuits and are used to remove unwanted signals (i.e. noise) from the actual signal. Transmission lines are essential for sending signals from one device to another, such as from a detector to a data acquisition module.

I. Filters

Filters are ubiquitous in analog electronic circuitry. In fact, if you see a capacitor or an inductor in a circuit there is a good chance it is part of a filter. Filters are frequently used to clean up (i.e. remove high frequency noise) power supplies and remove spurious frequencies from a signal (frequently 60 Hz, switching power supply noise in computers, display screen noise, ground loop noise, and Radio-Frequency (RF) pick-up).

A. RC Filters:

RC filters are by far the most common filters around. They are simple to make (i.e. just a resistor and a capacitor), reliable, and involve relatively simple design calculations.

1. The Low-Pass RC Filter

The low-pass RC filter, or integrator, is used to remove high frequencies from a signal. Applications include the removal of RF pick-up noise and reducing ripple voltages on power supplies.

A generic RC low pass filter circuit is shown in figure 4.1 on the right. We have already calculated its performance in the previous chapter using Fourier analysis and complex impedances. We recall the results (equation 28):

$$V_{out} = V_C = V_0 \sin(\phi) e^{i(\omega t + \phi - \pi/2)} \tag{4.1}$$

Figure 4.1: the classic low-pass RC filter circuit.

Where $\sin(\phi) = 1 \big/ \sqrt{1 + (\omega RC)^2}$ and $V_{in} = V_0 e^{i\omega t}$. From these quantities we can compute the gain and phase performance of the filter. The gain is defined as $Gain = \left| V_{out}/V_{in} \right|$ and the phase as $\varphi = \phi - \pi/2$ (this is just the part after the ωt in the exponent).

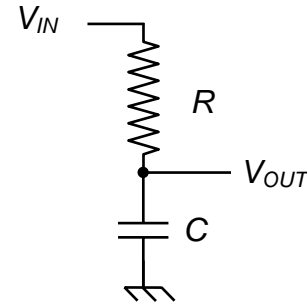The RC filter is just a voltage divider with complex impedances, so we can calculate the gain easily:

$$\left|\frac{V_{out}}{V_{in}}\right| = \left|\frac{1/i\omega C}{R + 1/i\omega C}\right| = \left|\frac{1}{1 + i\omega RC}\right| = \frac{1}{\sqrt{1 + (\omega RC)^2}} = \sin(\phi) \qquad (4.2)$$

The phase of the output voltage with respect to the input is easily computed and is given by

$$\tan\varphi = \tan(\phi - \pi/2) = \cot(\phi) = \omega RC \qquad (4.3)$$

At $\omega = 1/RC$, the output voltage drops to $1/\sqrt{2}$ of the input voltage, and consequently the power transmission drops to 50% or -3dB[1]. At this frequency, the voltage across the resistor and the voltage across the capacitor are equal in amplitude, but $\pm\pi/2$ out of phase with the drive voltage. The average value of V^2 across either the resistor or the capacitor is down by a factor of 2 from the drive voltage. Consequently, this frequency characterizes the *RC* circuit completely and is called the 3dB frequency. We can rewrite the gain and phase equations in terms of the 3dB frequency:

$$gain = \frac{1}{\sqrt{1 + (f/f_{3dB})^2}} \qquad (4.4), \qquad \tan\varphi = f/f_{3dB} \qquad (4.5)$$

with $f_{3dB} = 1/2\pi RC$.

The Bode plots (log-log or semi-log plots) for the gain and phase of the low pass filter are shown in figure 4.2, below.
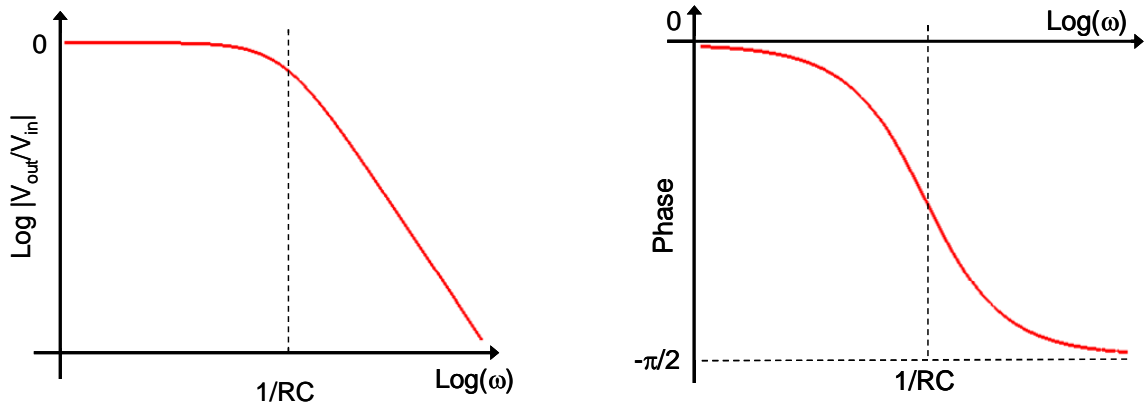
Figure 4.2: Bode plots of the relative magnitude (left) and phase (right) of the output of an RC low-pass filter.

---

[1] A dB or decibel is a notation for quantifying a ratio of two numbers. For power, a dB is defined as
$$dB = 10\log_{10}(P/P_0)$$
From this definition we can see that a ratio of 0.5 is roughly -3. Hence -3db is the same as halving a signal. For voltage or current, a dB is defined as
$$dB = 20\log_{10}(V/V_0)$$
So at -3dB, the voltage (or current) has dropped to $1/\sqrt{2}$ of its input value.

Past the -3 dB point, the log-log "slope" for the gain is -20 dB/Decade or -6 dB/Octave. The low-pass RC filter is also called an integrator because it integrates currents with frequencies above f_{3dB}. In other words $V_{out} = \frac{1}{C}\int I(t)dt$ (see chapter 3, equation 8) for currents with frequency components above f_{3dB}.

2) The High-Pass RC Filter

The high-pass RC filter, or differentiator, is used to remove low frequencies from a signal. Applications include the removal of DC bias voltages and 60 Hz pick-up voltages.

A generic RC high-pass filter circuit is shown in figure 4.3 on the right. Mathematically, high-pass filter can be treated the same way a their low pass cousins: they are almost identical except that Vout measures the voltage drop across the resistor instead of the capacitor. Qualitatively, the capacitor blocks DC and low frequency signals.

Figure 4.3: the classic high-pass RC filter circuit.
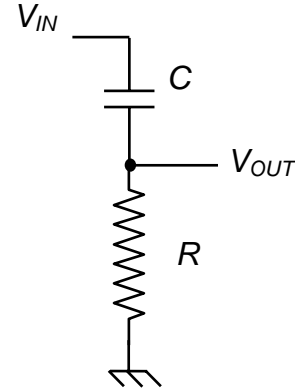
If we treat the high-pass filter as a complex impedance voltage divider, then we obtain immediately

$$\left|\frac{V_{out}}{V_{in}}\right| = \left|\frac{R}{R+1/i\omega C}\right| = \left|\frac{i\omega RC}{1+i\omega RC}\right| = \frac{\omega RC}{\sqrt{1+(\omega RC)^2}} = \cos(\phi) \qquad (4.6)$$

$$\tan\varphi = 1/\omega RC \qquad (4.7)$$

We can rewrite these using only f_{3dB} to produce more compact design equations:

$$gain = \frac{(f/f_{3dB})}{\sqrt{1+(f/f_{3dB})^2}} \qquad (4.8), \quad \tan\varphi = f_{3dB}/f \qquad (4.9)$$

The Bode plots (log-log or semi-log plots) for the gain and phase of the high-pass filter are shown in figure 4.4, below.

3) RC Filter Design

When designing an RC filter, you need to think about two things:

1. Choose an appropriate f_{3dB}.

2. Make sure the impedance is appropriate for your desired output load.

Step 1 is straightforward and depends on the frequencies you want to pass and block.

In Step 2 you pick R so as to satisfy the impedance requirements of your signal source and your signal destination. Typically, the signals we want to pass are around f_{3dB}, so the impedance of the capacitor and the impedance of the resistor will be about the same.
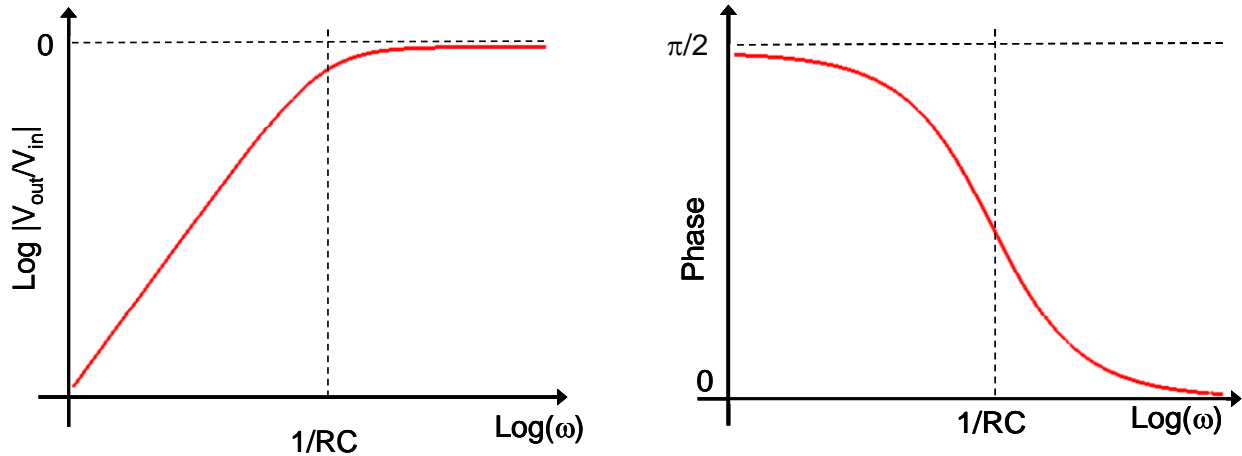
Figure 4.4: Bode plots of the relative magnitude (left) and phase (right) of the output of an RC high-pass filter.

When filtering an input, choose the resistance to be about 10 times smaller than the input impedance of the next stage. This will prevent the next stage from loading the filter. You would also like to choose the resistance to be at least 10 times larger than the previous stage of your circuit's output impedance.

Choose the capacitor for the appropriate $f_{3dB} = 1/(2\pi RC)$.

- If the signal you want to pass is low frequency, choose $f_{3dB} \approx 2f_{Pass}$, and hence $C \approx 1/(4\pi R f_{Pass})$.

- If the signal you want to pass is high frequency, choose $f_{3dB} \approx f_{Pass}/2$, and hence $C \approx 1/(\pi R f_{Pass})$.

3) Combination Filters

If you put several RC filters in a row you can make a more sophisticated signal filter. For example, several similar low-pass filters placed on after the other will produce a steeper fall off of the gain. One can also use a low-pass followed by a high pass to produce a bandpass filter ($f_{3dB,lowpass} > f_{3dB,highpass}$).

Design tip: When you are designing a bandpass filter, you should make sure that the resistor in the second stage is larger than the first stage by about a factor of 10. Otherwise, the second stage will load the first stage, and shift the effective f_{3dB} frequency of the first stage.

4) Applications

Here are some specific applications of RC filters:

Blocking Capacitor

This is a high pass filter that is used to eliminate DC. Suppose that you want to measure small time-dependent signals that happen to "float" on a high voltage. If you use a blocking capacitor, then the high voltage DC will not get through to your detection electronics, but the signal will get through. Choose f_{3dB} low to insure that your entire signal gets through.

Ripple Eliminator

This is a low pass filter used to build power supplies. Since most of our power is 60 Hz AC, our DC power supplies will convert AC to DC, but there will always be some residual 60 Hz "ripple". A low pass filter with f_{3dB} set well below 60 Hz will work. You do not use a resistance in this case, but let the combination of the loading resistance R_L and the Thevenin resistance of the previous components serve as your R. This usually requires a large capacitor since R_L might be quite small when you use the power supply. If the capacitor is not big enough, then f_{3dB} will then shift to a higher frequency, and the 60 Hz ripple will reappear.

Chip Supply Clean Up

Frequently the voltage which you supply to a chip component, such as an op-amp (we will study these later in the semester), may be "clean" when it comes out of the power supply, but will pick up noise by the time it reaches the component. In this case a 10 – 100 nF capacitor placed at the supply leads of the component will remove the high frequency pick-up noise.

Noise Eliminator

Any signal line is susceptible to picking up high frequency transients; especially if there are motors or switching power supplies (or FM radio stations!) nearby. A noise eliminator is a low pass filter with a high value of f_{3dB}.

Integrator

If you build an RC filter, but set the value of f_{3dB} much higher than the highest frequency in your signal, the filter integrates your signal. From our earlier analysis, when $f << f_{3dB}$, we can see that each (low) frequency voltage component will see a $\pi/2$ same phase shift and its amplitude will be proportional to $1/f$. This is exactly the prescription for integration. So, a high pass filter sends high frequencies out on the resistor, and the integral of very low frequencies on the capacitor.

Differentiator

If you build an RC filter with f_{3dB} lower than the lowest frequency in your signal, the filter differentiates your signal. From our earlier analysis, when $f \gg f_{3dB}$, each (high) frequency voltage component will see a $\pi/2$ same phase shift and its amplitude will be proportional to f. This is exactly the prescription for differentiation. So, a low pass filter sends low frequencies out on the capacitor, and the derivative of high frequencies on the resistor.

5. High frequency performance of capacitors

In principle, one could use an LR (inductor-resistor) circuit instead of an RC circuit to make low-pass and high-pass filters and obtain similar performance. However, in practice generally avoids inductors if possible. Inductors tend to be physically larger, more expensive, and deviate further from ideal performance than capacitors.

While capacitors generally offer superior performance to inductors, they also show significant deviations from the ideal $Z = 1/i\omega C$ impedance at high frequencies. Capacitors will generally have a little bit of spurious resistance (i.e. like a resistor) and inductance (i.e. like an inductor) at high frequencies. In fact, circuit designers will often model a real capacitor with the following simple circuit, though more complex circuits are sometimes necessary:
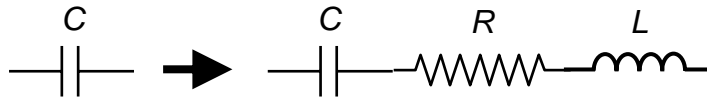
Figure 4.5: A real capacitor can be modeled as an ideal capacitor in series with some small resistor and inductor.

The resistance is due to the non-zero high frequency conductivity of the dielectric material separating the two conductor plates of the capacitor. The inductance is due to two effects: 1) the design of the capacitor, especially the leads, will contribute inductance., and 2) Maxwell's equations for electrodynamics require that a capacitor have an inductance at high frequencies[2]. Capacitor manufacturer will provide the specifications for the spurious resistance and inductance of their capacitors. The plot in figure 4.6, below, shows the frequency dependence of the impedance of a Cornell Dubilier acrylic surface mount film capacitor.

A common remedy for dealing with the inherent inductance of a capacitor at high frequencies is to place a small capacitor (10 – 100 pF) in parallel with the main capacitor in RC filter. The high frequency performance of the small capacitor will generally much better than that of the main capacitor: the small will pick-up the high frequency signal when the main, larger capacitor begins to have a significant inductance. The gain fall-off

---

[2] The Feynman Lectures on Physics, Vol. 2, by R. P. Feynman, R. B. Leighton, and M. Sands, p. 23-2.

of the RC filter will no longer be -20 dB/decade, but at least a low-pass filter will not start to behave like a high-pass filter (or vice-versa)!
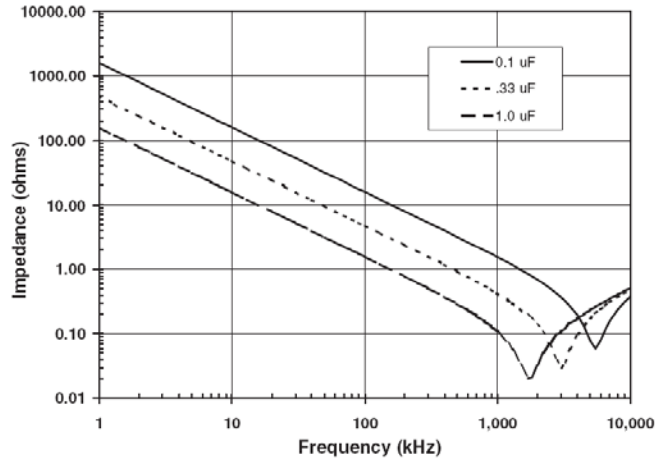

Figure 4.6: Impedance of real capacitors versus frequency. Note that the capacitors behave like inductors above a few MHz [source: Cornell Dubilier].

B. LC Filters

RC filters are by far the simplest and the most common type of filter found in analog circuits, however they suffer from a relatively slow roll off of the gain: while the gain or attenuation slope can be made steeper than -20 dB/decade, the transition region, or knee of the curve (the region where the gain changes from flat to a log-log slope), will always have the same shape and frequency width.

LC filters are more complex but can be engineered to produce much sharper features and steeper fall-off regions. The standard design for LC filters is an LC ladder with an un-interrupted ground line, such as in the 5th order filters shown in figure 4.7 and 4.8 below:
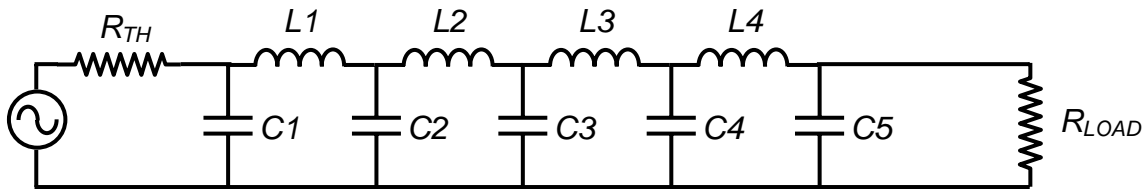
Figure 4.7: 5th order low-pass LC filter
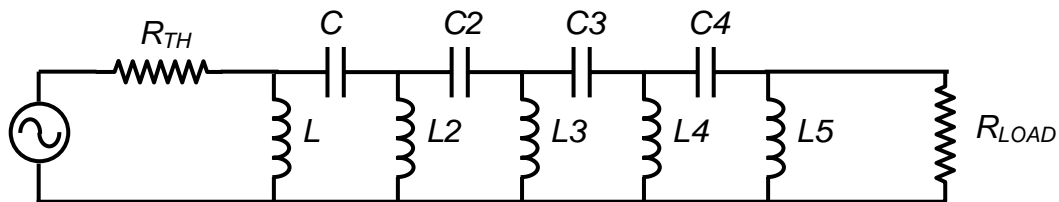
Figure 4.8: 5th order high-pass LC filter

The algebra required for computing the gain and phase Bode plots for these filters is generally quite cumbersome, and a computer program (i.e. Maple or Excel) is generally useful for helping with the design. A number of web-applets can also be found on the internet for determining all the inductor and capacitor values that will produce the required filter performance.

Example 1: 5th order Butterworth LC low-pass filter with f_{3dB}=10 kHz for 50 Ω input impedance and 50 Ω output impedance. The relative amplitude of the output is shown in figure 4.9, below.
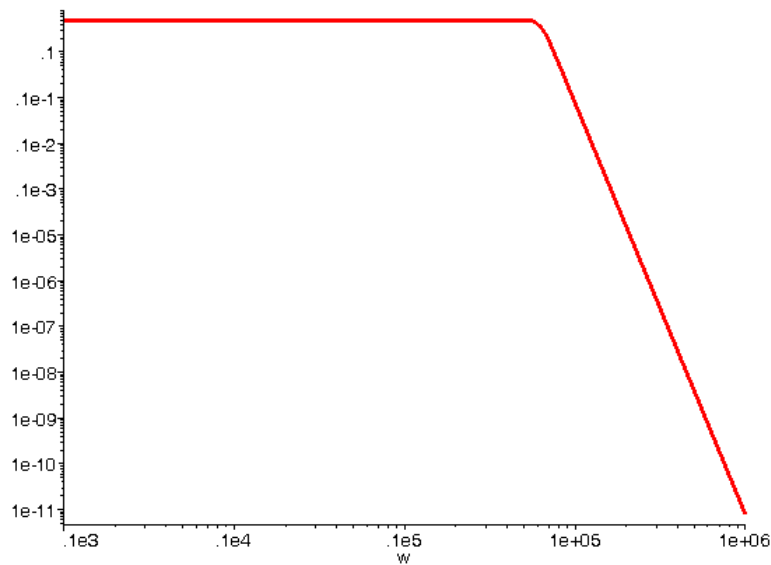
Figure 4.9: Bode plot of the gain of a 5th order Butterworth LC low-pass filter.

This Butterworth filter uses C1 = C5 = 0.6946 μF, C2 = C4 = 3.0642 μF, C3 = 4 μF, L1 = L4 = 5 mH, and L2 = L3 = 9.397 mH. Butterworth filters have a very flat pass-band, and a reasonably regular phase change across the knee of the curve.

Example 2: 5th order Chebyshev LC low-pass filter with f_{3dB}=10 kHz for 50 Ω input impedance and 50 Ω output impedance. The relative amplitude of the output is shown in figure 4.10, below.
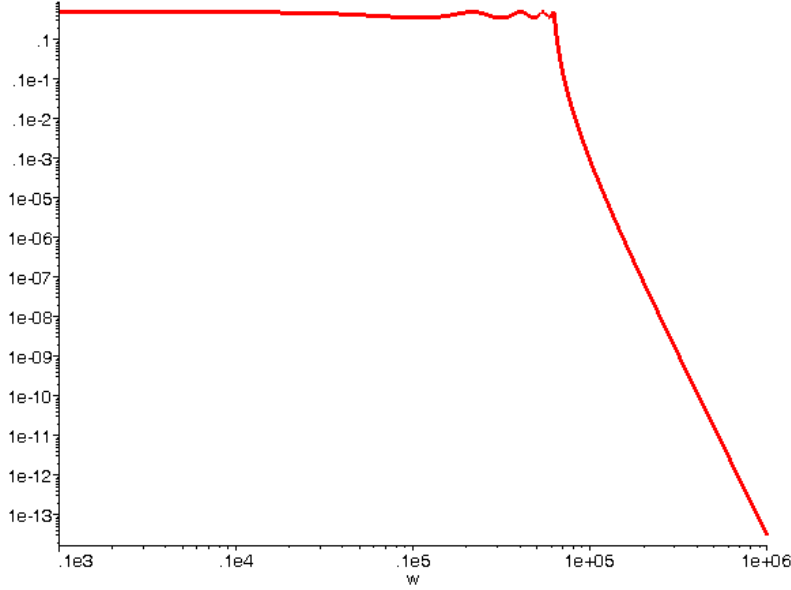
Figure 4.10: Bode plot of the gain of a 5th order Chebyshev LC low-pass filter.

This Chebyshev filter uses C1 = C5 = 1.125 µF, C2 = C4 = 1.486 µF, C3 = 1.505 µF, L1 = L4 = 0.617 mH, and L2 = L3 = 0.646 mH. Chebyshev filters have a very sharp knee and fast cut-off, but suffer from irregular transmission in the pass-band. They also have a highly irregular phase variation at the knee of the filter.

The above Chebyshev and Butterworth filters can be scaled to another frequency or load impedance with the following rules:

$$L_{new} = \left(\frac{R_{load,new}}{R_{load,old}} \right) \left(\frac{f_{3dB,old}}{f_{3dB,new}} \right) L_{old} \, , \; C_{new} = \left(\frac{R_{load,old}}{R_{load,new}} \right) \left(\frac{f_{3dB,new}}{f_{3dB,old}} \right) C_{old} \quad\quad\quad (4.10)$$

Some remarks on LC filters:

i. Generally, the higher the order of the LC filter, the sharper the cut-off will be, however this usually requires a trade-off in the regularity of the output phase.

ii. LC filters do not have resistive elements, and consequently they do not consume power, and do not load the source signal. However, the load of the input device into which the filter sends its output will load the LC filter and can shift the f_{3dB} point considerably – always remember to include the load resistor in your calculations.

iii. LC filters are widely used in RF circuits, where active filters do not have the bandwidth to respond to high frequencies.

iv. One cannot construct a filter with arbitrary gain and phase profiles. While we have treated filters in Fourier space, in the time domain filters must obey causality. In fact, one can derive Kramers-Kronig relations for filters.

II. Transmission Lines

Signals are sent from one device to another, or from one part of a circuit to another with transmission lines. The quality of your transmission determines the quality of your signal, whether you are connecting one device to another or a resistor to a capacitor.

LC Ladder Model of a Transmission Line

A transmission line consists of two parallel conductors separated by a fixed distance d, which gives rise to an effective capacitance C per unit length. The conductors will also produce a magnetic field, which is gives the transmission line an effective inductance L per unit length. We can model the transmission line as a repeated network of series inductors and parallel capacitors, or LC ladder, as depicted in the figure below.
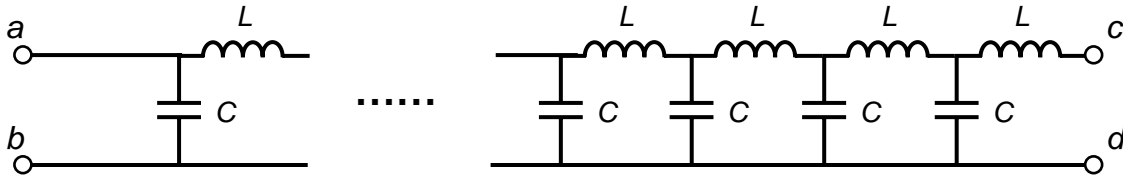
If the LC ladder is infinite and has an impedance Z_0, then if we add an extra LC ladder "rung", the total impedance should not change, and we obtain the following relation

$$Z_0 = (L + Z_0) // C \Leftrightarrow Z_0 = \frac{(i\omega L + Z_0)(1/i\omega C)}{(i\omega L + Z_0) + (1/i\omega C)} \tag{4.11}$$

From this relation we can extract an expression for Z_0, and we obtain

$$Z_0 = \frac{i\omega L}{2} + \sqrt{\frac{L}{C} - \frac{\omega^2 L^2}{4}} \tag{4.12}$$

If we consider that L and C are the inductance and capacitance, respectively, of a short section $\Delta\ell$ of transmission line, then as we take the limit $\Delta\ell \rightarrow 0$, we have L$\rightarrow$0 and C$\rightarrow$0, but L/C$\rightarrow$ constant. In this limit the equation for Z_0 becomes

$$Z_0 = \sqrt{\frac{L}{C}} \tag{4.13}$$

where L and C are the inductance and capacitance, respectively, of the transmission line, and Z_0 is called the characteristic impedance of the transmission line.

It may seem surprising that a network of inductors and capacitors can have a real impedance, and consequently consume power. The explanation for this apparent paradox is that since the network is infinite, power is flowing from one LC ladder rung to the next ad infinitum, so that power is constantly moving down the transmission line, though it is

not dissipated in either the inductors or capacitors. Of course, the power is consumed at the end of the transmission line when we attach a load resistor.

The LC ladder model is a high frequency model of transmission line and does not include the wire resistance which can contribute to signal attenuation.

Transmission Line Impedance Matching

A transmission line with a characteristic impedance of Z_0 should be terminated with a load impedance of Z_0 if the transmission line is longer than 1/10 of the wavelength of the signal (recall $wavelength = c/f$, where c is the speed of light in the transmission line and f is the signal frequency). If the transmission line is not properly terminated, then the signal will be partially reflected back towards the source upon arrival at the load.

There are three main types of transmission lines: wires, twisted pairs, and coaxial cables. In this section we go over their performance characteristics.

1. Wires

Plain wires are the simplest and the cheapest transmission lines available: they include the wires you use to connect components on your breadboard and electrical grid power lines. If the wires are kept parallel, then the transmission line will provide some protection to external fields and noise at low frequencies. While wire transmission lines are simple, they should be avoided whenever possible since they are susceptible to interference and high frequency pick-up. At high frequencies a wire transmission line actually becomes an antenna (for both transmission and reception).

2. Twisted Pairs

A twisted pair of wires provides good protection from outside fields and noise and can transmit relatively high frequency signals without difficulty. The electro-magnetic field of a signal kept close to the wire pair. A twisted pair can transmit analog signals at up to 250 kHz (sometimes even 1 MHz) and digital signals up to 100 MHz. Twisted pairs are quite common and are used in many computer communications cable, such as RJ45 Ethernet cables which has a characteristic impedance of $Z_0 = 100\ \Omega$. Twisted pair transmission lines are easy to make: just take two wires of equal length and twist them together!

3. Coaxial Cables

Coaxial cable is the best form of transmission line, short of a waveguide. In the limit of perfect conductors, signals on coaxial cables are impervious to external fields and do not radiate either. Coaxial cables can be used for frequencies up to 1 GHz. At high frequencies, a significant fraction of the transmitted energy/power is in the electric and magnetic fields that carry the signal though the cable, instead of in the potential energy of the current carrying electrons. The speed of light in a coaxial cable is usually 60-70% of

the speed of light in vacuum. The characteristic impedance of most coaxial cables used in industry and labs is $Z_0 = 50\ \Omega$ -- coaxial cable for cable TV is an exception, it has a characteristic impedance of 75 Ω.

Design Exercises:

Design Exercise 4-1: Design a high-pass RC filter that can filter out 60 Hz and 120 Hz but yet still pass signals in the kHz region. You can assume that your signal source has a 50 Ω impedance or lower.

Design Exercise 4-2: Design a band-pass filter which will only pass frequencies near 10 kHz. You can do this by combining 2 different RC filters (one high pass and one low pass). You can assume that your signal source has a 50 Ω impedance or lower.

Design Exercise 4-3: Design a notch filter which will attenuate 1 kHz by at least 60 dB, but will pass 1.2 kHz and 0.8 kHz with less than 6 dB of attenuation, and can drive a 100 kΩ load. You can assume that the signal source has a 50 Ω impedance.

Lab 4: Passive Filters

1. **Source Impedance Review** (30 minutes)

   Set the output of the function generator to a 1 Hz (or less) square wave with a voltage which varies between 0 V and 5 V. Try to use the output of the function generator to power a light bulb (if you do not see any light, then increase the voltage until you do).

   Measure the output voltage versus time when loaded by the light bulb. Is the output voltage consistent with what you would expect for a source with a 50 Ω impedance driving a light bulb?

   … By the way, what is the input impedance of the oscilloscope?

2. **High-Pass Filter** (1 hour)

   Design and construct a *high-pass RC filter* that can filter out 60 Hz but still pass signals in the kHz region.

   Connect a 10 kHz output from your signal generator to one terminal of the 6.3V transformer on your breadboard. This will add a large 60 Hz component onto the signal. Your filter should be able to clean it up.

   Measure the ratio of signal to noise (S/N), and compare it to your calculations, where "signal "is the peak-to-peak amplitude of the 10 kHz sine wave, and "noise" is the peak-to-peak amplitude of the 60 Hz component.

3. **Band-Pass Filter** (1 hour)

   Design and construct a *band-pass filter* which will only pass frequencies near 10 kHz. (You can do this by combining 2 different *RC* filters.)

   Measure its response (amplitude ratio and phase difference) when driving a load of 100 kΩ at 50 Hz, 100 Hz, 1 kHz, 5 kHz, 10 kHz, 20 kHz, 50 kHz, and 100 kHz. Does your circuit behave the way you expect it to?

Bonus (1 hour … maybe more)

   Construct the notch filter of design exercise 4-3.

   Characterize the performance of the filter (i.e. measure the amplitude ratio and phase difference) when driving a 100 kΩ load.

   Does the filter perform the way you designed it to? If not, explain why.

Chapter 5: Diodes

This week we will explore another new passive circuit element, the diode. We will also explore some diode applications including conversion of an AC signal into a signal that never changes polarity. To go beyond this and make a real DC signal will take a few more tools including low-pass filters.

I. Theory

Nonlinear Devices

The diode is our first nonlinear circuit element. A nonlinear element passes a current that is not proportional to the voltage across it (i.e. it does not obey Ohm's Law). Note, though, that a small change in the voltage across a diode will still generate a small change in its current. That is, if the change in voltage is small enough that the change in current will be proportional to that small voltage change. Thus, even circuits with diodes can use a Thevenin model for small deviations from the quiescent (or normal) condition of the circuit.

Diodes

Diodes are an important component of a DC power supply since they can convert an alternating current to a direct current. Most of the time, we will model a diode as simply a one-way gate for current.

In most wires and resistors, current is carried exclusively by electrons, which have negative charge. Semiconductors (silicon based devices) can use either negative charge carriers (electrons) or positively charged carriers (called holes) depending on what impurities have been added to the silicon when it was grown into a crystal. One type of impurity (called n doping) creates the negative carriers, and the other type of impurity (called p doping) creates positive carriers. A junction between p-doped and n-doped silicon makes a diode, as shown in figure 5.1, above. Electrons and holes can easily recombine with each other at a pn junction producing a net current flow

Holes and electrons recombine to let current flow

Current cannot flow the other way, since there is no energy source to create holes and electrons.
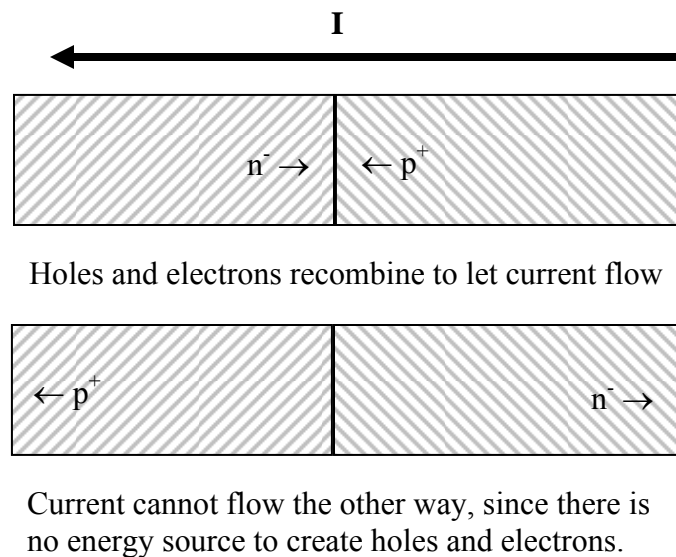
Figure 5.1: Hand-waving description of how a diode works in forward bias (top) and reverse bias modes (bottom).

from *p* to *n*. However, if the current wants to flow in the other direction, so that both electrons and holes move away from the junction, then the junction will act like an open switch. Thus, diodes are simple devices with the remarkable property that they only conduct current in one direction.

Diode Model 1: The current gate

For almost all practical purposes, we will treat a diode as a one-way current gate that also produces a 0.6 V voltage drop for silicon and 0.2 V drop for germanium. The voltage drop is due to the energy converted to heat as the electron-hole pair recombines. Diodes have a marking on them (sometimes just the line at the output end), showing the direction they conduct. In the accompanying figure, the diode will only conduct if $V_{IN} >$ 0.6 V. When the diode does conduct, any excess voltage drop shows up across the resistor. In other words, the diode just conducts enough current to set *I* such that

$$IR = V_{IN} - 0.6 \text{ V} \tag{5.1}$$
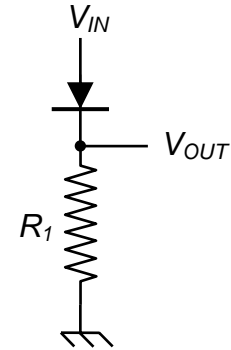
Figure 5.2: Diode as a one-way current gate

Diode Model 2: The Ebers-Moll model

A more complete model of a diode would treat the resistance of the diode as a logarithm of the voltage applied. In this more complete model, shown graphically in figure 5.3 below, we note that the 0.6V threshold is only an approximation for reasonable voltages.
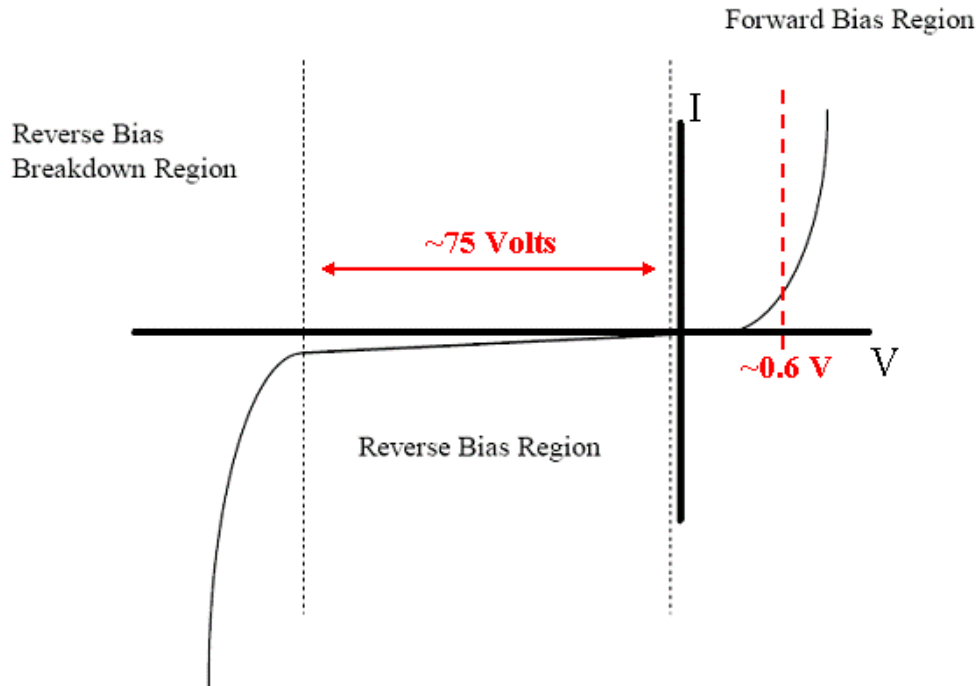
Figure 5.3: Typical silicon diode I-V characteristic curve
[figure adapted from etidweb.tamu.edu]

The I-V characteristic of a diode is given by the Ebers-Moll equation of solid state physics:

$$I = I_0[e^{V/(\eta V_T)} - 1]$$ (5.2)

Where for room temperature silicon, $I_0 = 10^{-9}A$, $\eta=2$, and $V_T=25.3\ mV$. This equation is valid in both the forward bias and the reverse bias regions, but does not apply in the breakdown region. Real diodes generally exhibit small deviations from this model.

II. Some Applications

Diodes have many applications. Here are a few of them:

A. Diode Rectifiers

Half-wave rectifier

One of the most common uses for diode is to rectify AC voltage to make a DC power supply. Since a single diode can only conduct current one way, when the input wave goes negative, there will be no current, as in the half-wave rectifier circuit of figure 5.4, on the right.
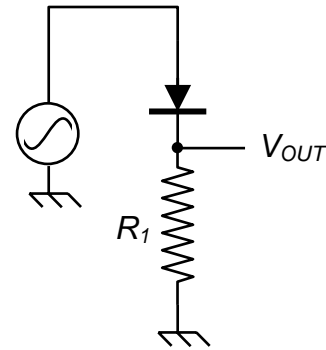
Figure 5.4: A half-wave rectifier

Full-wave rectifier

With four diodes, you can make both halves of the waves positive. This is called a full-wave rectifier diode bridge and is shown in figure 5.5 on the right. For both positive and negative swings of the input there is a forward path through the diode bridge. While two of the diodes are forward biased, the other two are reverse biased and effectively eliminated from the circuit. Both conduction paths cause current to flow in the same direction through the load resistor, accomplishing full-wave rectification.

Full-wave recitifiers are used in power supplies to convert AC voltages to DC voltages (or at least ones which are positive). A large capacitor in parallel with the output load resistor reduces the ripple from the rectification process.
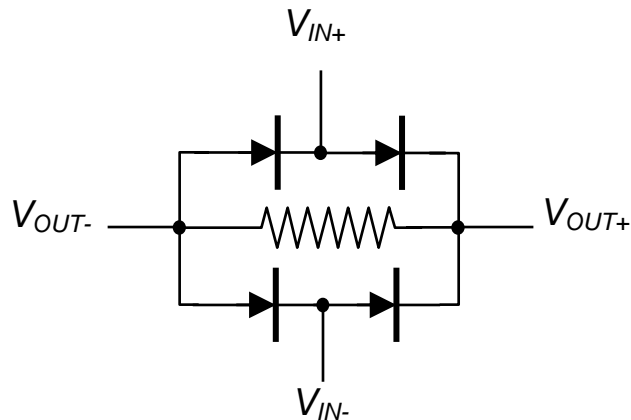
Figure 5.5: A full-wave rectifier circuit.

B. Frequency multipliers

A full-wave rectifier is also a frequency multiplier. The output of a rectified 60 Hz signal is at 120 Hz, but also has components at 240 Hz, 360 Hz, etc ... as is shown in the figure below
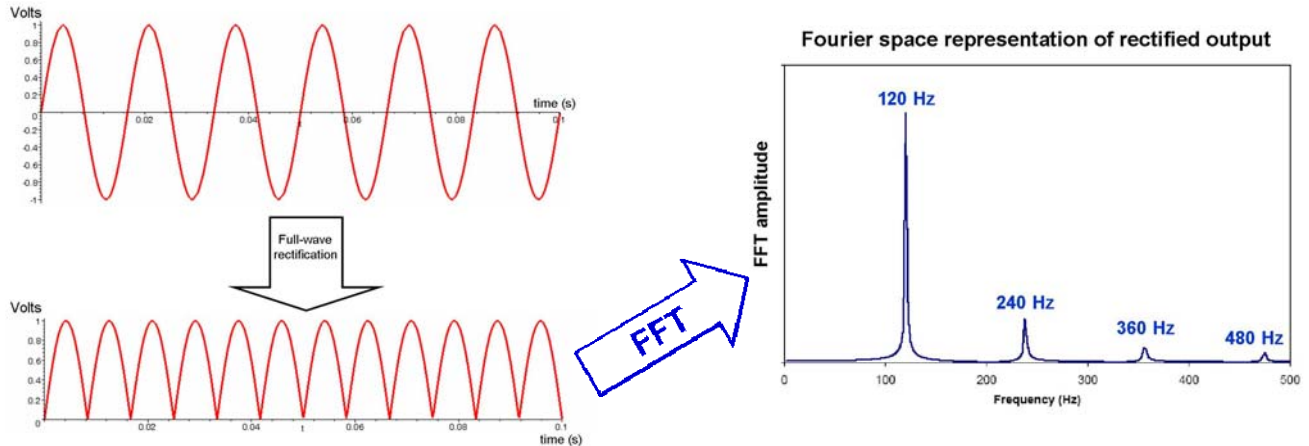

Figure 5.6: Ideal performance of a full-wave rectifier in the time and frequency domains.

In the above Fourier space representation, we have not included the zero frequency component (i.e. the DC bias) in the plot.

Filtering after the fullwave rectifier is used to remove the unwanted harmonics.

C. Frequency Mixers

A full-wave rectifier can also be used as a frequency mixer when two frequencies, f_1 and f_2 are input simultaneously into the full-wave rectifier: the output will include frequencies at $|f_1-f_2|$ and f_1+f_2, in addition to the harmonics of $2f_1$ and $2f_2$.

Design Exercises:

Design Exercise 5-1: Design a 12 V DC power supply out of an AC line source (120 V AC), a transformer with $N_p/N_s = 10$, some diodes, and a low pass filter (i.e. a large capacitor). Calculate the ripple voltage at the output of your supply when it has a 50 Ohm load.

Design Exercise 5-2: Design a frequency doubler with an input of 10 kHz and an output of 20 kHz, which rejects the input frequency and higher harmonics (not the second harmonic!) by at least 40 dB. The output should not include a DC bias.

Lab 5: Diodes

(estimated time: 30 minutes... 1 hour)
1. Measure the I-V characteristic of a diode (you may use a regular diode or an LED). Please make sure that you do not exceed ~100 mA though the diode. A good way to measure the I-V curve is to use the "one-way current gate" circuit from the course notes so that the resistor limits the current flowing through the circuit. Also, try to keep your circuit components close to room temperature since there performance will change if they heat up.

(estimated time: 30 minutes)
2. Construct the full-wave rectifier with a voltage signal at 10 kHz, 4 diodes, an audio transformer, and a load resistor of 100 kΩ. Measure the amplitude of the input signal and characterize the output signal through the load resistor. Why do we need the transformer?

(estimated time: 30 minutes)
3. (same set-up as question 2) Use the FFT (Fast Fourier Transform) function to measure the frequency spectrum of the output. Measure the amplitude and frequency of the principal harmonics (i.e. the one you can see). How do you convert dB to Volts? What FFT window did you use? Does it matter?

Chapter 6: Transistors and Gain

I. Introduction

This week we introduce the transistor. Transistors are three-terminal devices that can amplify a signal and increase the signal's power. The price is that we must also supply DC power to it (hence, the need for three terminals). Figure 6.1 shows a classic transistor package style.

The three terminals are called the **emitter**, the **base** and the **collector**. The base is the control terminal – a small current enters here and controls the big current that flows from collector to emitter. The notation may seem odd, but remember that electrons actually carry most currents and they have negative charge. A positive current flowing from the collector to the emitter means that electrons are flowing from the emitter to the collector.

Figure 6.1: A TO-92 transistor package.

In this chapter we will use a simple model for this device to try to understand its rather complicated behavior. As we continue through the semester we will encounter additional devices that require some sort of approximate models to describe their expected behavior. Since there are always approximations to the real devices they will have certain shortcomings or conditions were the models no longer apply.

The Basic Transistor Model

Bipolar transistors are essentially two diodes placed back to back. This may seem like a silly thing to do but the diodes are not the same. When current flows through one diode it provides carriers to carry current through the other part of the element. Thus, in its most basic form, **a transistor is a current amplifier**.

Bipolar transistors come in two basic types: *npn* and *pnp*. The current flow in a *npn* transistor shown schematically in the accompanying figure 6.2, below. Current (conventional, positive) goes in through the base terminal and out the emitter and the amplified current flows from the collector to the emitter.

Our first model of the transistor says that the collector-emitter current (I_{ce}) is β times the base-emitter current (I_{be}). Later, we will see that *good designs do not depend on β* since it varies from one device to the next, and it depends on the operating conditions (e.g. temperature). Most transistors have a value of $\beta \cong 100$. In this week's laboratory, you will measure β for a 2N3904 transistor.

In an *npn* transistor, a current will flow from the base to the emitter only if the base voltage is positive with respect to the emitter. A collector current will flow to the emitter only if the collector is positive with respect to the emitter. In a *pnp* transistor, things are reversed, and a current will flow from the emitter to the base only if the base voltage is

negative with respect to the emitter. A collector current will flow from the emitter only if the collector is negative with respect to the emitter. The arrow on the emitter tells you which way the current is supposed to flow, and which also indicates whether the base voltage must be positive or negative with respect to the collector.
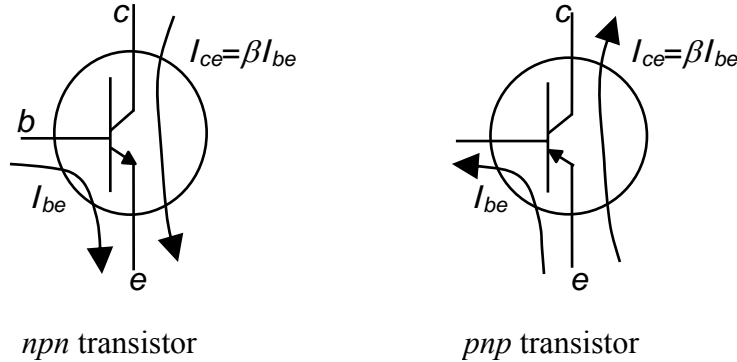

npn transistor   pnp transistor

Figure 6.2: Basic operation of npn (left) and pnp (right) bipolar junction transistors.

Let's summarize the conditions required for an *npn* transistor to conduct (the "transistor rules"):

1. $V_{be} > 0$. Since this is a diode, V_{be} should be roughly 0.6V when it is conducting.

2. $V_{bc} < 0$. This is a back-biased diode with enough voltage that the base current normally flows to the emitter. You do not want current to flow from the base to the collector (i.e. $V_{bc} > 0$).

3. $I_{ce} = \beta I_{be}$. $\hspace{7cm}$ (6.1)

Under normal operating conditions we will have a resistor (or load) connected to the emitter or to the collector, and we will see a voltage drop across the load as I_{ce} flows through the transistor. If I_{ce} changes it will change the voltage drop across the transistor (V_{ce}). If V_{ce} drops too low, then V_{bc} will become positive since V_{be} must still be roughly 0.6 V. Current then flows from the base to the collector, leading to an apparent drop in β. This is called saturation.

II. Basic Transistor Circuits

Here are some basic circuits which illustrate the operation of transistors.

A. Constant Current Source

The current source is the simplest transistor circuit possible and is shown in figure 6.3 below. You simply drive a small current, I_{be}, into the base of a transistor, and it produces a large current, I_{ce}, through the rest of the circuit. In this case I_{be} is given by,

$$V_+ - V_{be} = I_{be}R_b \hspace{6cm} (6.2)$$

and the current through the load can then be calculated from our current gain model ($I_{ce} = \beta I_{be}$). From this we can see that current flowing through the load does not depend on the load resistance. It does depend on β so it is not an ideal design.

The current flowing through the load should be independent of the load resistance. If your load resistor gets too big, though, the voltage drop across the load will make V_{cb} too small to conduct. In this case the current I_{ce} will drop until the transistor starts to conduct again. This reduces the effective β of the transistor (i.e. some base current flows to the collector). This condition is called *saturation*.
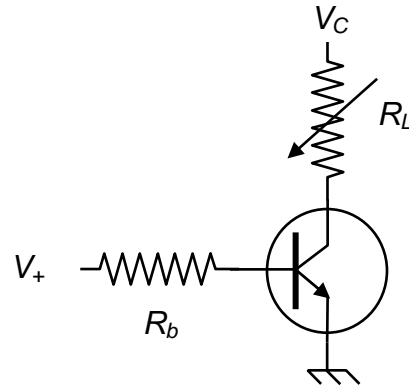
Figure 6.3: A basic npn transistor current source.

B. Transistor Switch

We can use a transistor switch to control a big current with a small current. This is just a current source with the device you wish to switch in place of the load resistor. A common application is to use a digital (0 or 5V) signal with moderate impedance to control a device requiring a lot more power. In next week's lab, we will use the 5V TTL (transistor-transistor logic) output of your function generator to drive a light bulb.

In designing a transistor switch you want most of the voltage drop to be across your device. If V_{ce} is large then the power dissipated inside the transistor ($I_{ce}V_{ce}$) will destroy the transistor. You can get around this in your design by designing the switch for saturated operation. Since the switch runs in saturation mode, some of the base current will flow up to the collector and the apparent β will be smaller than our nominal 100. A common design strategy is to design for a "generous base current," which could be satisfied with an effective gain of roughly 10.

C. The Emitter-Follower Amplifier

The most important circuit in this chapter is the emitter follower (see figure 6.4 below). This is a very easy circuit to design, and its normal operating conditions do not depend on β. An input voltage on the base produces a base current and an amplified collector current, to generate a voltage drop across the emitter resistor. But the voltage of the emitter cannot increase beyond $V_{IN} - 0.6V$, or the base current will drop. Consequently, the circuit produces a voltage of $V_{IN} - 0.6V$ at the output. In other words, the output (at the emitter) follows the input! In fact, this simple rule, $V_{be} = 0.6V$, is usually sufficient to understand most transistor circuits.

The emitter follower is extremely useful because it transforms impedance. The current amplification of the transistor means that it can drive a low impedance, while

pulling only a small current from the source. This is just like the transformer – but without the transformer's voltage decrease.

In the lab, you will build an emitter follower and measure its impedance. You should find

$$\frac{Z_{\text{OUT}}}{Z_{\text{SOURCE}}} = \frac{1}{1+\beta} \; .$$ (6.3)

The derivation of this formula is left to the reader.

D. An Inverting Amplifier

The final circuit is the inverting amplifier, which is also called a common collector, and is shown in figure 6.5. An inverting amplifier is similar to an emitter-follower with two changes. It has an additional resistor between the power supply and the collector, and the output is at the collector terminal. Of course our transitor rules still force the emitter to be $V_{IN} - 0.6V$, and this will set the transistor current to be

$$\frac{V_{\text{IN}} - 0.6\,\text{V}}{R_e} \; .$$

When this current passes through the collector resistor, it will generate a voltage drop of

$$R_c \frac{V_{\text{IN}} - 0.6\,\text{V}}{R_e} \; .$$

which means that the voltage drop across the collector resistor will be approximately $-R_c/R_e$ times the input voltage. The output is then given by

$$V_{OUT} = V_{CC} - R_c / R_e \times (V_{IN} - 0.6\ V).$$ (6.4)

The minus sign means that when the voltage at the base is zero, the collector will read the full supply voltage. When the voltage at the base is high there will be a big current and the collector voltage will drop almost to zero. Thus the output will be exactly the oposite of the input. If you find this disturbing you can use two inverting amplifiers in series to get a noninverting amplifier.
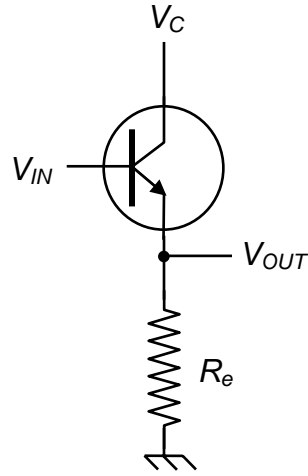
Figure 6.4: The classic emitter-follower amplifier circuit.
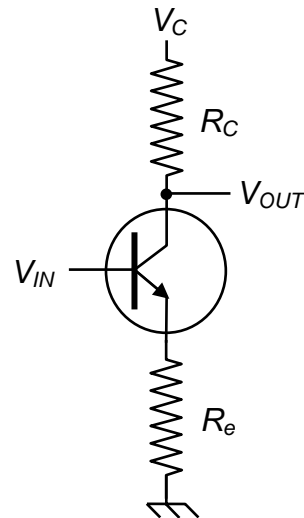
Figure 6.5: An *npn* transistor inverting amplifier.

Design Exercises

Design Exercise 6-1: Design a NPN transistor current source with a base supply voltage of 5 V, a collector supply voltage of 10 V, and a target constant current of 10 mA. Assuming that β=100, determine the value of the base resistor, and the range of collector/load resistance over which you expect the current source to produce 10 mA. What happens when the collector/load resistance goes above this range?

Design Exercise 6-2: Design a NPN transistor switch for a light bulb with a resistance of 50 Ω and a collector supply voltage of 6 V. Make sure the current into the base will guarantee operation in saturation mode (gain of roughly 10, instead of β=100) which is roughly independent of the exact light bulb resistance.

Design Exercise 6-3: Consider an NPN emitter follower amplifier that has R_e=8 Ω. Calculate the input impedance for the amplifier. Assume that your input signal has a DC bias of 3 V, and an AC amplitude of 1 V, and that β=100. Determine the collector supply voltage necessary to keep the average power dissipated in the transistor to below 0.5 Watts. What is the power dissipated in the 8 Ω load resistor?

Lab 6: Introduction to Transistors

(Estimated time: 1 hour)

1. Measure the current gain of a 2N3904 transistor using the constant current source circuit from design exercise 6-1. If you measure the collector current through the collector/load resistor for a variety of base currents (i.e. change the base resistor or the voltage applied to it), then you should be able to extract a value for β.

(Estimated time: 1 hour)

2. (same set-up as lab exercise 1) With the component values you calculated for design exercise 6-1 (appropriately updated for your measured value of β), verify that the circuit is indeed a constant current source by trying different collector/load resistors and measuring the collector current. Verify that the constant current source goes into saturation when you expect it to.

(Estimated time: 1 hour)

3. Construct an *emitter-follower amplifier* using the 2N3904 (*npn*) transistor whose gain you've previously measured. The circuit should be based on your calculations for design exercise 6-3 (adjusted for the real value of β if necessary).

   a. Adjust the function generator to output an AC signal at 1 kHz with an amplitude of 1 V and a DC bias of 3 V.

   b. Measure the input impedance of your speaker, and measure the relevant voltages across it when it is attached directly to the function generator.

   c. Use the speaker as the load resistor in your emitter-follower circuit, and adjust the collector supply voltage so as to keep the transistor power dissipation below 0.5 W.

   d. Power up your emitter-follower circuit and measure the relevant voltages across the speaker and at the transistor base. Is the speaker louder?

   e. Does the output signal sent to the speaker show any signs of distortion? How low can you set the collector voltage before performance is significantly affected? How do you measure the amount of distortion?

NOTE: Hang on to your transistor for next week.

Chapter 7: AC Transistor Amplifiers

The transistor amplifiers that we studied in the last chapter have some serious problems for use in AC signals. Their most serious shortcoming is that there is a "dead region" where small signals do not turn on the transistor. So, if your signal is smaller than 0.6 V, or if it is negative, the transistor does not conduct and the amplifier does not work.

Design goals for an AC amplifier

Before moving on to making a better AC amplifier, let's define some useful terms. We define the *output range* to be the range of possible output voltages. We refer to the maximum and minimum output voltages as the *rail voltages* and the *output swing* is the difference between the rail voltages. The *input range* is the range of input voltages that produce outputs which are not at either rail voltage.

Our goal in designing an AC amplifier is to get an input range and output range which is symmetric around zero and ensure that there is not a dead region. To do this we need make sure that the transistor is in conduction for all of our input range. How does this work? We do it by adding an offset voltage to the input to make sure the voltage presented to the transistor's base with no input signal, the resting or *quiescent voltage*, is well above ground. In lab 6, the function generator provided the offset, in this chapter we will show how to design an amplifier which provides its own offset.

Now that you understand capacitors it is pretty easy to see how to add and subtract an offset voltage to a signal, at least for AC signals. From here on, you will design transistor circuits with a bias network. This bias network is simply a voltage divider that is connected to the input. Its job is to insure that the output stays at approximately half the supply voltage for small input signals. Then, the output voltage can vary over a wide range (positive and negative) while always keeping the transistor conducting.

The trick to make this work is to separate these quiescent voltages and currents from the input and output signals. To do this, you will use blocking capacitors to isolate the input and the output. If you connect an AC input to a capacitor, it will not pass any DC offset voltages but it does pass the fast AC signals. Similarly, an output blocking capacitor will pass the fast signal while keeping the quiescent (resting) voltage from the amplifier from disturbing whatever comes next. This is shown schematically in the first figure (next page).

Some Design Basics

This week we are going to redesign our emitter follower and inverting amplifier to use bias networks. To help you with your design, we will make a step by step list for designing each of these basic transistor circuits.

Here are a couple initial design decisions we will make

- You will begin by determining the quiescent (DC, no signal) current through the collector. You usually want the quiescent current to be larger than any current you will use to drive a load. A quiescent current of 1 mA is typical, and we will use that in our example designs.

- We will also use a single +15 V power supply to power the collector (the *common collector voltage* or V_{CC}) for and the bias network.

- We need to be careful about loading the different stages of this amplifier. The transistor's base current will load the output of our bias voltage divider. To bias the base, we need a stiff voltage divider (i.e. low impedance). Our rule of thumb for designing voltage dividers was to have a factor of 10 difference in impedance at each stage.

AC Emitter-Follower

Design steps for the emitter-follower of figure 7.1 proceed as follows:

1. To have the maximum symmetric range of output voltages we would like our quiescent base voltage to be half of the (15 V) supply voltage. So, we will use a 1:1 input voltage divider. This means that both biasing resistors will be the same.

2. We then choose the emitter resistor. The quiescent voltage at the emitter is a diode drop below the voltage in the middle of the bias network (i.e. $V_{cc}/2$ if we have a 1:1 divider). This is roughly +7 V in our case. To get our design quiescent current, the emitter resistor must be

$R_e = V_e/I_e = 7$ V / 1 mA $= 7$ kΩ.

We will use the standard 6.8 KΩ. It is close enough.

3. To bias the base we want a stiff voltage divider (i.e. low impedance), therefore we want to use resistors that are smaller than the base-emitter-ground impedance $(Z_b=V_b/I_b \sim 750$ kΩ) by a factor of 10 or so. For this example, we will choose two 75 kΩ resistors for this divider.

4. Remember to AC couple (via capacitors) the input and output. The exact values are not particularly important, though you should remember that you are making a biased high-pass RC filter. Values around 0.1μF are typical if you want $f_{3dB} \sim$ *20 Hz*, but you can use what you have as long as it is not too small.
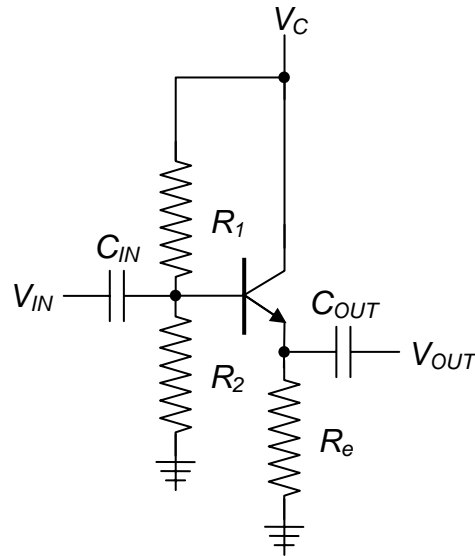
Figure 7.1: A biased emitter-follower *npn* transistor amplifier.

Common-Collector (Inverting) Amplifier

In this circuit, we need to know the quiescent current and the desired gain. Let's assume a gain of -5 and a 1mA quiescent current for this example. The circuit diagram is shown in figure 7.2.

1. In this circuit we want the quiescent output (at the collector) to be set roughly halfway between the power supply and the ground for maximum output voltage swing. For $I_c = 1$ mA,

   $$R_c = V_c/I_c \approx 7 \text{ V} / 1 \text{ mA} = 7 \text{ k}\Omega.$$

   We will use a standard resistor of about this value (e.g. $R_c = 6.8 \text{ K}\Omega$) as we did for the follower.

2. The emitter resistor can be determined by the desired gain. Previously we saw that

   $$\text{Gain} = -R_c/R_e$$

   In our case we want a gain of 5 so we chose $R_e = 1.35 \text{ k}\Omega$. We will approximate this by a standard 1.5 kΩ resistor. Note that with this choice, the emitter quiescent voltage will be given by the voltage drop across the emitter resistor

   $$I_c R_e = 1.5 \text{ V}.$$

Figure 7.2: A biased *npn* transistor inverting amplifier.

3. The tricky part is to design the bias network for this circuit. Since we know the emitter voltage, the output of the bias network (i.e. the base voltage) is just a diode drop higher than the emitter voltage. Therefore the bias resistors must be set to give a base voltage, V_b, of

   $$V_b = V_2 = V_e + 0.6 \text{ V}$$
   $$= 1.5 \text{ V} + 0.6 \text{ V}$$
   $$= 2.1 \text{ V}.$$

   This is the drop across the lower of the two bias resistors. The other has a drop of

   $$V_1 = V_{cc} - V_2$$
   $$= 15 \text{ V} - 2.1 \text{ V}$$
   $$= 12.9 \text{ V}$$

   The bias network's output impedance must be small enough to keep this bias up even when loaded. We will select the bias resistors so that the current running through the bias network is about 10 times larger than the current that goes into the base. If $\beta=100$, then the base current is

   $$I_b = I_c/\beta = 10 \text{ }\mu\text{A},$$
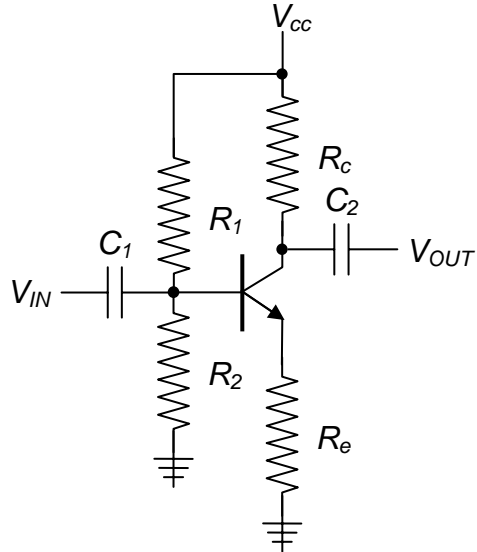
and the bias current, I_{bias}, is ten times larger: $I_{bias} = 100$ μA. The total resistance of the bias network is R_1+R_2, so we need

$R_1+R_2 = (15 \text{ V}) / (100 \text{ μA}) = 150 \text{ k}\Omega$.

We can now determine the value for R$_2$, since from the voltage divider formula, we require:

$V_b / V_{cc} = R_2 / (R_1+R_2)$ ⇒ $R_2 = (2.1 \text{ V} / 15 \text{ V}) \cdot (150 \text{ k}\Omega) = 21 \text{ k}\Omega$,

and therefore,

$R_1 = 150 \text{ k}\Omega - 21 \text{ k}\Omega = 129 \text{ k}\Omega$.

4. Remember to AC couple the input and output with capacitors. The values are not particularly important, but the relevant RC time constants should be chosen so as to guarantee unimpeded passage of the AC signal to be amplified.

Design Exercises

Design Exercise 7-1: Design an AC emitter follower with a 1.5 mA quiescent current. Remember that you can approximate the resistor values by picking the nearest standard values. Please show any approximations when they are employed.

Design Exercise 7-2: Design an AC inverting amplifier with a 0.2 mA quiescent current and a gain of 15.

Design Exercise 7-3: Calculate what happens in the hybrid emitter-follower circuit in figure 7.3 below. What is the current through the load resistor? What does this circuit do? Is it a "good" design?
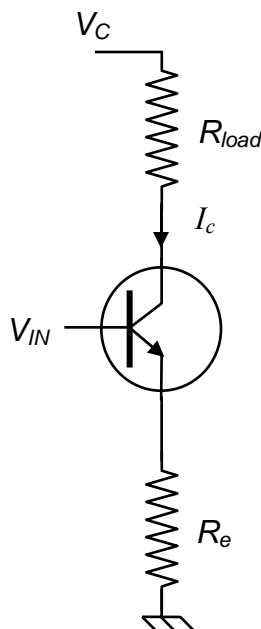
Figure 7.3: What does this circuit do?

Lab 7: Transistor Applications

1. **Transistor Switch** *(30 minutes ... 1 hour)*

Construct a voltage controlled transistor switch by connecting the collector of a 2N3904 transistor though a light bulb to a 6 V power supply (see design exercise 6-2). Use a square wave voltage to control the switch. Measure the voltage across the light bulb and compare with your quantitative and qualitative results from lab exercise 4-1 (if necessary repeat lab exercise 4-1).

2. **DC-biased AC transistor amplifier** *(2 hours ... much longer if not prepared)*

   a. Design and construct a common-emitter amplifier with a quiescent current $I_C = 0.2$ mA, and a gain of ~15 at 2 kHz, powered by a +15 V power supply. Measure the small-signal AC gain, and compare it to your calculation. Measure the voltage swing (i.e. the maximum output voltage swing before distortion). Measure the output impedance (Suggestion: use AC coupling).

   b. Short your emitter resistor to obtain the maximum gain. Measure the new small-signal gain. It will be necessary to change your bias network. Does this agree with your expectations? Measure the new voltage swing. Measure and describe any distortion.

   c. Return to your common-emitter amplifier with a gain of 15. Now try to eliminate the emitter resistor without affecting the DC bias, by using a by-pass capacitor parallel to the emitter resistor. Measure the new small-signal gain. Measure and describe any distortion.
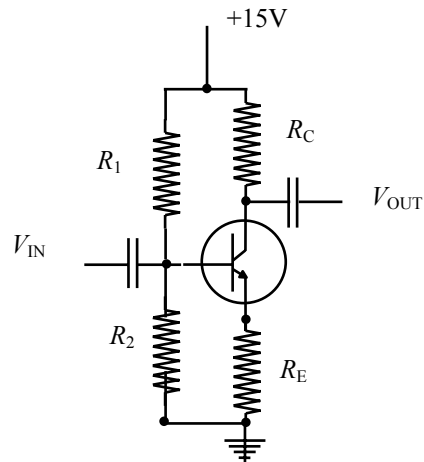
Chapter 8: Field Effect Transistors

Transistors are different from the basic electronic elements in that they have three terminals. Consequently, we need more parameters to describe their behavior than one simple current-voltage characteristic. We found that thinking of the transistor as a current amplifier was a particularly useful model. This model, however, does have its limitations and we could extend it.

A next step would be a *transconductance* model, such the Ebers-Moll model. In this improved model, the collector current depends on the base-emitter *voltage*, so that V_{BE} produces I_C. In this model, V_{BE} also produces I_B so this model includes the earlier current amplifier model in addition to the new features.

I. Field-Effect Transistors

A. Introduction to FETs

This week's we introduce a new element called a *Field-Effect Transistor* (*FET*). It is also a three-terminal element. The three terminals are similar to the transistor's base, emitter and collector, but they are called the *gate*, *source* and *drain*. The initial FETs we will use (the 2N5485) have identical packages to the transistors we used in earlier in the semester. The schematic symbol for the FETs we will use are shown in figure 8.1 on the right.
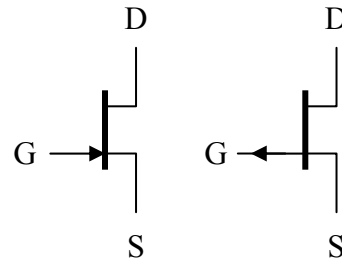
Figure 8.1: An n-channel JFET (left) and a p-channel JFET (right).

A FET is produced from a single piece of conducting silicon that connects to the source and the drain. This is called the *channel*. The gate is then created by diffusing a third connection, the gate. A voltage applied to the gate controls the conductivity of the channel.

The gate-channel junction looks like a diode that never conducts hence the gate draws no current. This is the major difference between normal transistors and FETs. Consequently FETs have extremely large gate input impedances ($>10^{12}\Omega$). Thus, we will be forced from the start into modeling the device with a transconductance model where the drain current depends on the gate voltage and not the current. This is the major difference between the bipolar diodes that we used previously and FETs.

FETs are extremely important as input stages to amplifiers. Since they have such a large input impedances that they almost attain the ideal of measuring a voltage without drawing any current from the source. This is great if you want to measure small charges deposited on capacitors – as you might want to do in a particle detector.

FETs come in five general types, but we will restrict ourselves to JFETs (for Junction FET) initially and our examples will only use *n*-channel JFETs. These have *n* channel doping and are similar to *npn* transistors. The *p* channel JFET requires the opposite voltage on the gate. They usually have poorer performance due to the lower mobility and shorter lifetimes of holes, as compared to electrons.

As mentioned above, the source-drain current is the only current that flows through a FET. The source-drain current is labeled I_D. The voltage applied to the gate terminal enables this current by creating an electric field inside the channel. There is no fundamental difference between the source and drain terminals of a JFET. The gate-drain capacitance, however, is usually lower than the gate-source capacitance so they are usually used as specified. This makes it different from a normal transistor, since current can flow either from the drain to the source or from the source to the drain.

Note that there is a maximum value and a minimum value for the gate voltage in order to keep the device operating. Since a JFET has a diode junction separating the gate from the channel, the gate must be held at a voltage of less than 0.6 V above the channel (usually the source terminal). If the gate voltage becomes greater than this, the junction will become conducting and the gate current will no longer be zero. Usually, we will not let the voltage between the gate and the source (V_{GS}) get any greater than 0. If the gate is biased too negative then no current flows and the channel is said to be "pinched off." This minimum gate voltage, called V_P, is a characteristic that varies from one model of JFET to the next. It is usually in the range from –3 V to –10 V. Even within the same type of FET this parameter varies significantly from one device to the next. For example, the range specified for a 2N5485 the range is between -0.3 V and -3.0 V.

Let's summarize these properties:

For $V_{GS} < V_p$: $I_D = 0$ (8.1)

For $V_{GS} > 0.6$ V: Device Fails!   (8.2)

When V_{GS} is between these bounds I_D depends on both V_{GS} and V_{DS}. A complete description of the device would require a two-dimensional plot showing how I_D varies with both V_{GS} and V_{DS}.
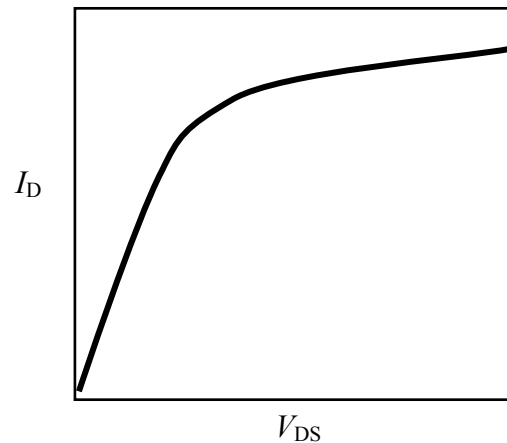
Figure 8.2: A sketch of drain current versus channel voltage.

B. Regions of Operation

For a fixed V_{GS}, the basic behavior of the drain current as a function of the channel voltage is shown in Figure 8.2 above. The behavior can be simply modeled in two regions. For small values of V_{DS}, I_D is almost proportional to V_{DS} just like a resistor. This is called the *linear region*. For reasonably large values of V_{DS}, I_D has little dependence on V_{DS}. This is called the *saturation region*. The actual relationships and conditions can be approximated as:

For $V_{DS} < V_{GS} - V_P$: $I_D = k [2(V_{GS} - V_P)V_{DS} - V^2_{DS}]$ Linear Region   (8.3)

For $V_{DS} > V_{GS} - V_P$: $I_D = k (V_{GS} - V_P)^2$ Saturation Region (8.4)

For small enough voltages the quadratic term is small. In this case, the current in the linear region is proportional to V_{DS} just like a resistor. This is true even for negative V_{DS}. Since I_D is proportional to both V_{GS} and V_{GS} this gives us a way to make voltage-controlled resistors. Moreover, since I_D is proportional to both V_{DS} and V_{GS}, we can generate a current corresponding to the product of two signals.

In the saturated region I_D is nearly independent of V_{DS}. We can use this to make excellent currents sources and source followers. This is similar to our transistor-based emitter follower but with a much larger input impedance. Note that in this region there is still a small slope and this slope increases for larger gate voltages. We will discuss the saturated region later in the chapter.

C. A Voltage Controlled Resistor

From Equation 8.3 we can see that the drain current is proportional to V_{DS}, when V_{DS} is small, and the proportionality constant is proportional to V_{GS}. This is perfect for a making voltage-controlled resistor. A typical data sheet will list $R_{D(ON)}$ as the resistance measured when the gate is shorted to the source. From Equation 8.3 we see that $R_{D(ON)}$ is approximately $-1/2kV_p$. If you put this "resistor" in series with a fixed resistor, then you will have a voltage-controlled attenuator. If this follows or precedes an amplifier the combination would be an amplifier with a voltage-controlled gain.

Figure 8.3: Uncompensated voltage-divider.

The resistor R_D and the FET in Figure 8.3 form a voltage divider to attenuate the input voltage. The size of the equivalent resistance of the FET is determined by V_{GS}, which is set by the potentiometer.

By making the right choice of gate voltages we can improve the circuit by eliminating the quadratic term in Equation 8.3 and thus improving the linearity of the attenuator. From Equation 8.3 we see that

$1/R_{DS} = I_D / V_{DS}$

Figure 8.4: Compensated voltage-divider.

$$= k \, [2(V_{GS} - V_P) \text{ - } V_{DS}] \tag{8.5}$$

If we set V_{GS} to exactly $V_{DS}/2$ then we find

$$R_{DS} = \text{-}1/(2kV_P) \tag{8.6}$$

Figure 4 shows a version of the attenuator that employs this improved linearity. The two resistors connected to the gate force half of the drain voltage onto the gate. They should be much larger than the potentiometer. The capacitor just blocks DC so that only the signal part of the input voltage is fed to the gate.

Application: Amplitude Modulator

If you feed a second, lower frequency, signal into the gate in addition to the DC voltage from the potentiometer, then you can modulate the attenuation of your voltage divider. Of course, you should feed this in via a capacitor so that you do not disturb the quiescent (DC) conditions.

II. The Saturated Region

As we saw in equation 4, for large values of V_{DS} the drain current only depends on V_{GS} :

$$I_D = k \, (V_{GS} - V_P)^2. \tag{8.7}$$

This saturation is useful, for example, for making current sources and followers. As discussed earlier, there is still some dependence of I_D on V_{DS} even in the saturated region. You can, however, reduce this effect with a few extra tricks.

A. Transconductance

The *transconductance* of a FET is denoted by g_m and is defined as

$$g_m = \Delta I_D / \Delta V_{GS} \tag{8.8}$$
$$g_m \approx 2k \, (V_{GS} - V_P) \tag{8.9}$$
$$g_m = 2(kI_D)^{1/2} \tag{8.10}$$

The transconductance will be useful in understanding the behavior of the FET amplifiers.

You can interpret it as the slope of I_D vs V_{GS} in the saturated region and depends on your choice of I_D. A curve of $g_m(V_{GS})$ and $g_m(I_D)$ are usually shown in the data sheets and they have complicated shapes. If you know the value for one set of conditions you can use the above relationships to scale to other similar values.

The transconductance has units of Ω^{-1} which is known as a *mho* (pronounced "moe"). You often seed the units of μmho (or umho pronounced "micro-moe") for 10^{-6} mho and mmho ("milli-moe") for 10^{-3} mho in FET specifications. You might also see an inverted capital omega or "S" to represent this unit.

The remainder of this section will cover some applications for FETs employing the saturated region.

B. JFET Current Sources

You can easily make a FET *current source* simply by connecting the gate to the source (i.e. $V_{GS} = 0$ V) and applying a high enough voltage to the drain that the JFET operates in the saturated region (Figure 1). This generates a current through the load. This specific current is I_{DSS}, where the subscript denotes that this is the drain current when the gate is shorted to the source.

According to the transconductance model given by equations 3 and 4, the saturation current, I_{DSS}, when $V_{GS} = 0$, is given by

$$I_{DSS} = kV_P^2 \qquad (8.11)$$

Two-terminal JFETs, such as the 1N5294, are exactly this circuit packaged to look like a diode and sold under the name *current-regulator diodes*.

Since we know there are large variations in V_P this may not seem like the best specified device. In fact, when marketing these devices, a manufacturer will actually measure its response and only sell devices under this product type when they pass the roughly specified current (within a factor of two of 7 mA). This is called *preselection* and is common in some categories of commercial electronics production and also in devices with large variations such as photodetectors. Since this is expensive, the companies invest huge amounts into development on the more expensive components (e.g. CPU chips or disk drive heads) to improve quality control during fabrication.
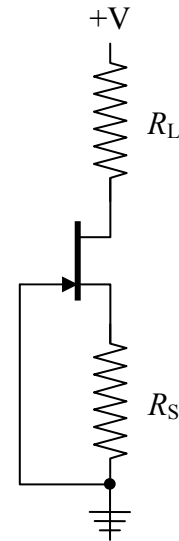
You can make this circuit an adjustable or *programmable constant current source* by adding a resistor between the gate and source to provide an offset (Figure 5). Since there is a voltage drop across the resistor the current is given by

Figure 8.5: A programmable constant current source driving a load.

$$
\begin{aligned}
I_D &= k(V_{GS} - V_P)^2 \\
&= k(-R_S I_D - V_P)^2 \\
&= I_{DSS}\left(\frac{R_S I_D}{V_P} + 1\right)^2
\end{aligned}
\qquad (8.12)
$$

This can be solved for I_D but it is rather ugly. In practice, one normally starts from a plot of I_D vs V_{GS} on the device's data sheet to get a rough idea of the source resistor. Since the saturation properties are not well determined in the fabrication process (up to factors of 10), use a potentiometer if you wish to sink a specific current.

C. JFET Follower

The drain current in a JFET operating in the saturation region depends only on V_{GS}. Because of this, you can add a source resistor to a FET to provide at type of negative feedback. This is called a *source follower*.

In the follower's quiescent state there will be nominal values for I_D and V_{GS}. An increase in the gate voltage will increase the current into the source resistor. This will cause an increased voltage drop and act to raise the source voltage. It is conventional to use lower case letters to represent small variations from the quiescent point.

In our transconductance model, small variations would be governed by

$$i_d = g_m(v_g - v_s). \qquad (8.13)$$

The source voltage is tied to the drop across the source resistor,

$$v_s = i_d R_S. \qquad (8.14)$$

Solving these two equations, we find that

$$v_s = \frac{R_S g_m}{1 + R_S g_m} v_g = \frac{R_S}{1/g_m + R_S} v_g \qquad (8.15)$$

For R_S much greater than $1/g_m$ we see that $v_g \approx v_s$ (i.e. gain = 1) and therefore this is a follower.
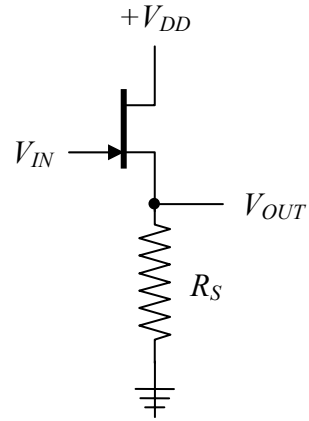
Figure 8.6: A simple source follower.

FET follower limitations

From Equation 9, we can see that the source follower acts like a voltage divider with the FET behaving like a resistor of $1/g_m$. Since it is a voltage divider we can determine its output impedance. Since the FET is the smaller resistor in the divider (or the output will not be very close to the gate), the output impedance of the source follower is just $1/g_m$, which is typically a few hundred ohms at currents of a few milliamps.

So, our simple follower, however, does not behave too well:

1. As V_{GS} varies, the nonlinear transconductance will distort the input.

2. It has quiescent output voltage that is specified by the pinch of voltage of the device and hence is unpredictable.

3. The required V_{GS} to produce a given quiescent current varies a lot from one device to the next

4. The output impedance is relatively high.

On the good side, the input impedance is immense.

A matched pair follower

A very common solution is to replace the ground at the end of the source resistor with a current source. This is called and *active load*. By putting in a constant current sink, we always have the same I_D and thus a fixed value for V_{GS}. The first problem is solved. It does not, however, fix the other problems.

We can do an even better job by noting manufacturing variations in FETs are significantly smaller for FETs made from the same piece of *silicon wafer*, or *die*. You can think of them as identical twins instead of just siblings. This is called a *matched pair*. The 2N3958 and U441 are examples of a matched pair JFETs.

Using the second JFET as the active load gets rid of most of the problems in the FET follower (Figure 7). The second JFET keeps the current constant, fixes V_{GS}, and eliminates the nonlinearities. The matched resistors also add better I_D predictability. Furthermore, when you use a matched FET pair and symmetric power supplies it largely eliminates the DC offset at the output. To fix the poor output impedance, one would need to add an op amp follower or amplifier to the output.

Matched pairs are used for the input stages of most charge measurement applications (e.g. oscilloscope inputs), sensitive electrometers, measurement of small charges (e.g. fC) like one sees in particle detectors, and current pre-amplifiers for photo-detectors.
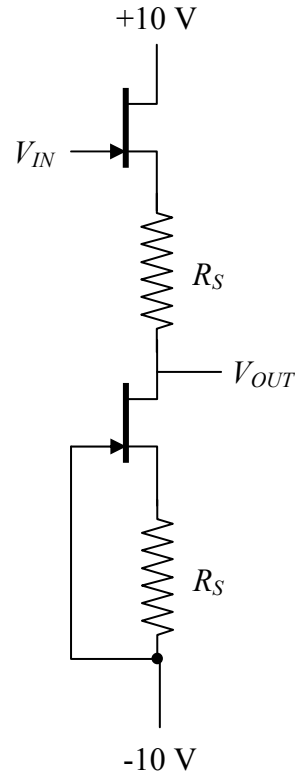
Figure 8.7: A matched pair follower.

Design Exercises

Design Exercise 8-1: Design a programmable current source with a design current of 1mA and a variable gate-source resistor. Pick a reasonable value based on the plot of I_D as a function of V_{GS} from the datasheet of a 2N5485.

Design Exercise 8-2: Determine the quiescent voltages (i.e. V_{GS}, V_S, and V_{DD}) and the quiescent power consumption for a source follower with I_{DS}(quiescent) = 1 mA and R_S = 4.7 kΩ. The N-JFET for the circuit has the following properties: V_P = -2 V, I_{DSS} = 3 mA, and g_m = 2.5 mmhos.

(hint: Start by assuming that the circuit is operating in the saturation region and then choose V_{DD} at the end of the calculation to ensure that the FET is well into the saturation region.)

Lab 8: Introduction to FETs

1. **N-JFET Basics** *(1.5 hours)*

   a) Measure the characteristics of a 2N5485 *n*-channel JFET by measuring I_D versus V_{DS} while the gate is tied to the source (i.e. $V_{GS} = 0$). Construct a sketch as you make your measurements. Using the Formulas 1 and 2 from the text and data in the linear and saturated regions, compute V_P and k.

   b) Measure the characteristics of the JFET by measuring I_D versus V_{DS} for an additional value of V_{GS} between V_P and ground. Does this agree with our model (*i.e.* does one value of k fit all your data)?

V_+

(A)

V- or GND
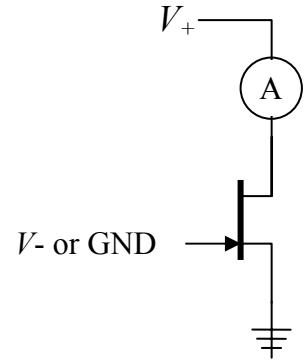
Do lab exercise 2 or 3.

2. **Voltage controlled JFET attenuators** *(1.5 hours)*

   a) Construct an uncompensated attenuator with $R_D = 10$ KΩ. Use a voltage divider with your variable −15 V supply to generate the V_{GS} control voltage in the approximate range of -0.5 V to -5 V. Connect the output of the voltage divider to the gate through a 1 MΩ gate resistor. Try to attenuate your input by a factor of 10 using this device when you drive the drain with a small signal (<1 V) around 1 kHz. Note that this attenuator even works for the negative values of V_{DS} function generator. Check for distortion using the FFT feature on the oscilloscope.

   b) Now compensate your attenuator using a second 1 MΩ feedback resistor between the drain and the gate. Use a ~0.1 µF capacitor to block the DC drain voltage. Measure the new attenuation and distortion. How do the new characteristics compare to the uncompensated ones?

   c) Connect the signal from another function generator to the voltage divider to form a bias-T using another capacitor. Set this frequency to be roughly 10 times lower than the frequency on the drain. Use small AC inputs for both signals (tenths of volts) and describe the output. Describe the signal in the time domain and in frequency space. Do your results agree with theory? What arithmetic operation does your circuit implement on the two input signals?

3. **Matched-pair source-follower** *(1.5 hours)*

    a) Construct a simple source follower with a 2N3958 (or U441) JFET. Use a 4.7 kΩ source resistor. Using a small sine wave around 1 kHz, measure its gain and DC offset. Estimate g_m from the attenuation. Measure its output impedance. Do they agree?

    b) Add an active load to your source follower using the second JFET from the matched pair. Measure its new gain, its DC offset and its output impedance.
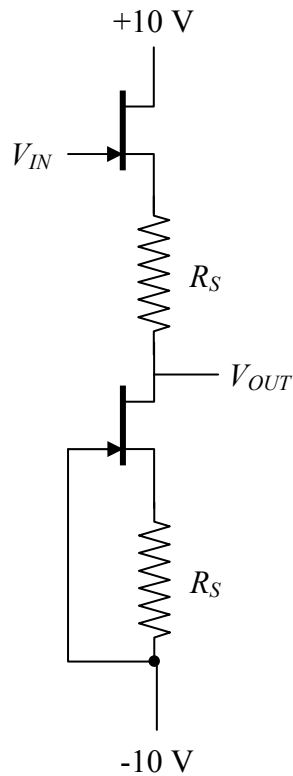
A matched-pair follower

Chapter 9: Operational Amplifiers

The *Operational Amplifier* (or *op-amp*) is the ideal, simple amplifier. It is an *integrated circuit* (*IC*). An IC contains many discrete components (resistors, capacitors, and transistors) all made at the same time on the same piece of silicon and put into a standardized package. Most of the circuit design has already been done for you, so they are particularly easy to use. An op-amp has (almost) infinite input impedance, very low output impedance, and extremely high gain (10^5 is typical). Since the gain of these devices is so large, you will almost never use them as simple amplifiers. Instead, you will reduce the gain, and make it more stable, using negative feedback.

So, you might ask, why did we not start here, and skip all the components that go into the op-amp? Well, op-amps require two power supplies (usually ±15 V), and they cannot dissipate too much power. If you have a very simple job or high power, a transistor is better. If you find yourself putting together several elements then you should probably be using an op-amp.

Op-amps

An op-amp has two inputs, the inverting input (*V*–) and the non-inverting input (*V*+), and one output, as shown in figure 9.1 on the right. The fundamental relation governing the operation of an op-amp is that the output is the gain, G, times the difference between these input voltages:

Figure 9.1: An op-amp.

$$V_{out} = G \cdot (V_+ - V_-) \qquad (9.1)$$
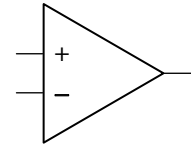
Since the gain is so large, this effectively requires that *V*+ = *V*– for the op-amp to be in its normal operating range. In lab this week we will use an LM741 op-amp and more modern equivalents, such as the OP27 and the LT1498.

An op-amp also has at least two other connections to it. Typically, it will require two separate power supply inputs (e.g. ±15 V), sometimes a ground, sometimes frequency compensation, and possibly output trimming. The output trimming enables you to set the output voltage to zero when the two inputs are tied together.

The symbol for an amplifier is a triangle with the inputs on one side and the outputs on point on the other side. If extra connections like power, compensation, or frequency control are shown, they are indicated by connections to the sides of the triangle.

Op-amp Golden Rules

Given the high input impedance and the near infinite gain, it is easy to analyze an op-amp's performance. Under normal operation with feeddback, the op-amp will follow these two golden rules:

1. An op-amp draws no current into either input.

2. An op-amp will do anything it can to its output to insure that its two inputs have the same voltage.

Negative Feedback

For negative feedback, you will couple the output signal back into the inverting input in some way. In its simplest form, negative feedback forces the op-amp output to be whatever is required to insure that the two inputs have the same voltage. We will define two different gains with the op-amp. The *open loop gain* (A) is the gain without feedback from the output to the inverting input. The *closed loop gain* (G) is the gain of the circuit with feedback from the output to the inverting input.

Follower

A *follower* (i.e. $V_{OUT} = V_{IN}$) is the simplest op-amp circuit with feedback. It can be made by simply connecting the output directly to the inverting input, as shown in figure 9.2 on the right. Since the op-amp will change the output to make the inverting input equal to the non-inverting input. This is a unity gain device just like those we made with single transistors except with will have even better input and output impedances. Followers are frequently called **buffers**, since they can be used to isolate one sub-circuit from another.
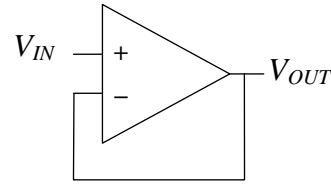
Figure 9.2: Op-amp follower or buffer.

Inverting Amplifier

An *inverting amplifier* with gain can be made by inserting a couple of resistors to the feedback system, as shown in Figure 9.3. From the golden rules, the op-amp will adjust its output to make the inverting input be the same as the non-inverting input (i.e. ground). Therefore the voltage drop across R_1 is V_{IN} and the voltage drop across R_2 is V_{OUT}. Since the input cannot sink any current the current through the two resistors must be equal. To make this happen, the output and input voltages are related by

$$\frac{V_{OUT}}{R_2} + \frac{V_{IN}}{R_1} = 0 \qquad (9.2)$$
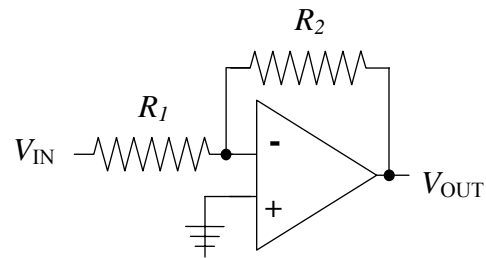
Figure 9.3: Inverting Amplifier

Gain is defined as the ratio of V_{OUT}/V_{IN}. Hence the gain is the ratio $G=-R_2/R_1$. Note that most of the time, one uses feedback resistors in the $1-100$ kΩ range.

Non-Inverting Amplifier

A *non-inverting amplifier* can be made using two resistors as shown in Figure 9.4. The analysis is very similar to the previous example. The voltage drop across R_1 is V_{IN}. The current flowing through the R_1 and R_2 resistors will be the same because the inverting input draws no current. Using this we can show that the gain is given by:

$$G = \frac{V_{OUT}}{V_{IN}} = 1 + \frac{R_2}{R_1} \qquad (9.3)$$
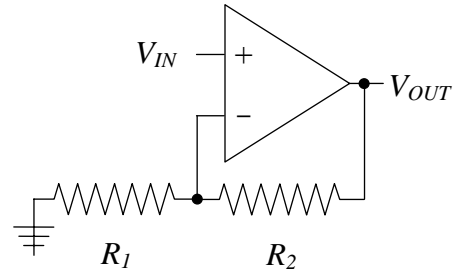
Figure 9.4: Non-Inverting amplifier.

We can also note that the follower was just a non-inverting amplifier with $R_2=0$ and an infinite R_1.

Summing Amplifier

Since the inverting input of the op-amp is a virtual ground, one could connect a number of inputs to the same point. The result is a *summing amplifier*, shown in figure 9.5. One can, for example, use this to add a DC offset, or bias, to a sine wave without blocking DC input with a capacitor.
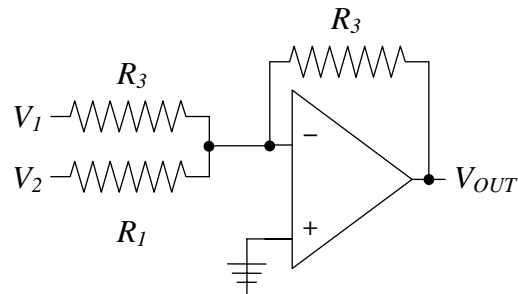
Figure 9.5: Summing amplifier.

Some Limitations

While the op-amp is nearly ideal, it does have limitations. It is good to define the limitations early in the topic to help us understand the properties of the real circuits we will try in the lab.

1. An amplifier can be modeled as an op-amp with an *open loop gain* of *A* (large but finite). The open loop gain is usually frequency dependent too.

2. The feedback loop actually redirects a fraction of the output current back to the inverting input. The fraction of the output that is returned is denoted by *B*.

3. A small but finite current is being drawn into the inputs. They are not quite the same.

4. They can only drive limited amount of current even thought they have low output impedance.

We will discuss each of these issues on the following sections.

Finite Open Loop Gain

The open loop (no feedback) and closed loop gains (in a feedback loop) are related by

$$G = \frac{A}{1+AB} = \frac{1}{B+A^{-1}}$$

(9.4)

where G is the closed loop gain and A is the open loop gain.

An example of an application of this relationship can be given for the gain of the inverting amplifier. The closed loop gain needs to be modified to include this effect by the open loop gain putting in parallel with the gain due to the feedback. The resulting gain is

$$G = \left(\frac{1}{A} + \frac{R_1}{R_1+R_2} \right)^{-1}$$

(9.5)

A common way to indirectly measure the open loop gain is to measure how the circuit gain deviates from B^{-1}. When the open loop gain is equal to B^{-1}, the total gain will be half of B^{-1}.

Finite Input Current

Since the inputs can sink (or draw) some current, it is also a good idea to match the impedances seen by the two inputs. To make a more perfect non-inverting amplifier, we can introduce a third resistor at the non-inverting input (see figure 9.6). Since the inverting input sees a voltage divider, the third resistor should be chosen to have the same resistance (i.e. R_1 and R_2 in parallel).
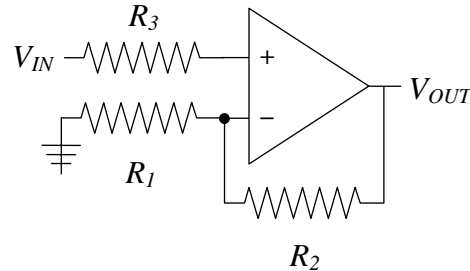
Figure 9.6: Impedance matched non-inverting amplifier.

Slew Rate

The output voltage can only change so fast. This is due to internal capacitance in the op-amp used to control phase changes. The slew rate of a circuit is the maximum rate at which the output can change. Its value determines what frequencies can be usefully amplified with a particular op-amp. For an LM741 the slew rate (SR) is about 0.5V/µs, and on more modern equivalents, such as the OP27 and LT1498, it is an order of magnitude larger.

Voltage Compensation

The internal components of the op-amp like the transistors at the inputs are, in general, not completely identical. This often leads to what looks like an artificial voltage difference (or *voltage offset*) between the inverting and non-inverting inputs. Most op-amps have an option to "trim" the inputs so that this effect can be eliminated. Examples of how to make these corrections area are shown in the data sheets of op-amps.

Op-amp Frequency Compensation

An op-amp's gain at DC can be very large but it decreases with increasing frequency. Since an op-amp consists of several discrete amplifiers following each other, each stage will have its own frequency limitations, which we can model simply as low pass filters.

Any low pass filter introduces a 90° phase shift at frequencies higher than f_{3dB}. Consequently, at high frequencies the various internal amplifier stages will produce a net phase shift that is at least 180°. Since a 180° phase shift is a sign reversal, this turns negative feedback (which stabilizes a circuit) into positive feedback (which drives the output to the rails). So, it is important to insure that the gain of any op-amp circuit has decreased so that the loop gain of your circuit is not larger than 1 by the time the net phase shift reaches 180°. Adding a capacitor that allows high frequencies to bypass the second stage of the amplifier chain will typically do this. This low pass filter thus limits the gain to be less than one where the phase shift reaches 180°.

Most of the op-amps we will use are internally compensated to ensure that the phase is constant over a wide range of frequencies even if you use the op-amp for open loop gains. This holds true until the open loop gain falls to about 1.

Some op-amps come in versions that are either uncompensated or decompensated (not completely compensated), and you must add a capacitor externally. Since your loop gain may be less than the open loop gain, this lets you use the op-amp for gain at a higher frequency than you could get using an internally compensated version.

Design Exercises

Design Exercise 9-1: Design a voltage divider that will take a signal with a 150mV peak-to-peak signal and reduce it to a 0.5mV peak-to-peak output.

Design Exercise 9-2: Design an inverting amplifier with a gain of 10.

Design Exercise 9-3: Derive the gain relationship for a differential amplifier (as seen in class).

Design Exercise 9-4: Design an amplifier which takes the arithmetic average of three input signals.

Design Exercise 9-5: Consider input voltage signals V_a and V_b which differ by a small amount, which is the real signal you want to extract. Design a"differential" amplifer circuit (more than one op-amp is necessary) which performs the following operation:

$$V_{out} = \gamma(\alpha V_a - V_b + V_0)$$

Where α is a variable gain of 0.1 to 10 on the V_a input, V_0 is an adjustable DC offset with a range of -10 V to +10 V, and γ is an overall gain with a range of 1 to 100. The input impedance for V_a and V_b should be in the 100 kΩ range and the output should be able to drive 10 kΩ at 10 V. You may use as many op-amps as you want in your design. Variable control can be achieved using a potentiometer (i.e. a variable resistor).

Lab 9: Basic Op Amp Circuits

1. **Open loop op-amp** *(1 hour)*

   Using an attenuated output from your function generator, try to measure the open-loop gain at 100 kHz, 10 kHz, and 1 kHz (preferably in this order).

   Attempt to measure the open loop gain of an LM741 op amp by connecting the negative input to ground, and the positive input to the wiper arm of a 1-30 kΩ potentiometer that has its two ends connected via resistors to the ± variable supplies and preferably an additional voltage divider. Can you find an input voltage that does not drive the output to one of the rails (i.e. approximately V+ or V-)? Make an estimate or put a lower bound on the DC open-loop gain.

2. **Op-Amp with feedback** *(0.75 hours)*

   Do either part a or b:

   a) Design and construct a non-inverting amplifier with a gain of 10. Measure your circuit's gain, output range, input impedance, and output impedance.

   b) Design and construct an inverting amplifier with a gain of 15. Measure your circuit's gain, output range, input impedance and output impedance.

3. **Laboratory differential amplifier** *(1.25 hours …but infinite if you are not prepared)*

   Construct the differential amplifier you designed in design exercise 9.5. Construct two signals sources using a 1 V AC signal at 1 kHz from a function generator and two voltage dividers in parallel – one divider will be divide by 3, the other will be divide by 2. Show that the you can use the gain control knobs to cancel the two signals. Also, adjust the gain settings to produce an output of 5 V pk-pk and look for distortion. Repeat at 100 kHz.
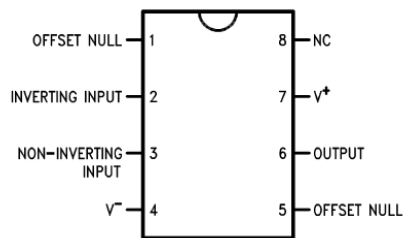
LM741 pin out

Chapter 10: Op-amp circuits for detectors, filters, and power applications.

We can improve the performance of most of the circuits we have built in this course, including RC circuits and rectifiers, by using op amps and negative feedback.

I. Active Filters

A. RC Integrator and Low Pass Filters

Recall that a low pass circuit was an integrator for high frequencies. The RC integrator we built in the laboratory was limited as an integrator because the current charging the capacitor changed as the capacitor charged up. This meant that the circuit integrated only for small voltages.

By putting an integrating capacitor into the op- amp feedback loop, you force the input side of the capacitor to always be at the same voltage (virtual ground) so the input current no longer depends on how much charging it has done. This integrator works all the way up to the maximum output of the op amp. For the circuit shown in figure 10.1, the output will be:

Figure 10.1: Op-amp integrator (with low frequency limit).

$$V_{OUT} = -\int V_{IN} \frac{dt}{RC} . \qquad (10.1)$$

Of course, you must limit the low frequency behavior, or else the integrator will always be at the rail as it integrates any small offset voltage. This is done by putting in an extra resistor that will slowly bleed the charge of the capacitor over the time.

B. RC Differentiator and High Pass Filters

Putting the capacitor at the input, and using a feedback resistor, you can construct an accurate differentiator. This time, your op amp will be stable with no input (if it has been internally compensated). For our circuit in figure 10.2, the output is:

$$V_{OUT} = -RC \frac{dV_{IN}}{dt} . \qquad (10.2)$$

Note, thought, that it has limited gain at high
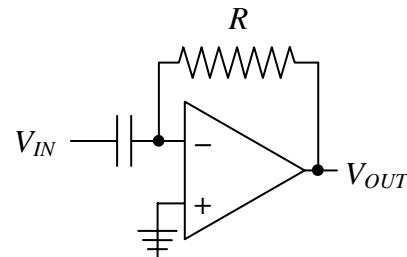
Figure10.2: Op-amp differentiator.

frequencies that will degrade its performance for fast derivatives.

II. Current Sources

The inherent negative feedback present in op-amp circuits is ideal for making high-impedance constant current sources. (why do ideal current sources have a high output impedance?)

A. Op-Amp Current Source

A *current source* can be made from an inverting amplifier as shown in figure 10.3. The current in the load resistor must be the same as that in R_1. The current is then just the input voltage divided by R_1.
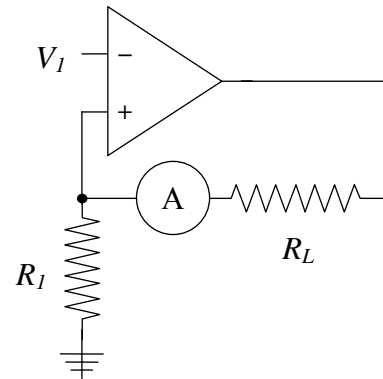
Figure 10.3: Current source.

B. Op-Amp and Transistor Current Source

An op-amp can be combined with a transistor to make a current source in which the load is outside the feedback loop, as depicted in figure 10.4 (left) below. The N-JFET transistor can be replaced by an NPN or a N-MOSFET transistor. The FET based circuits will have slightly better current regulation, and a MOSFET should be used for high current applications.
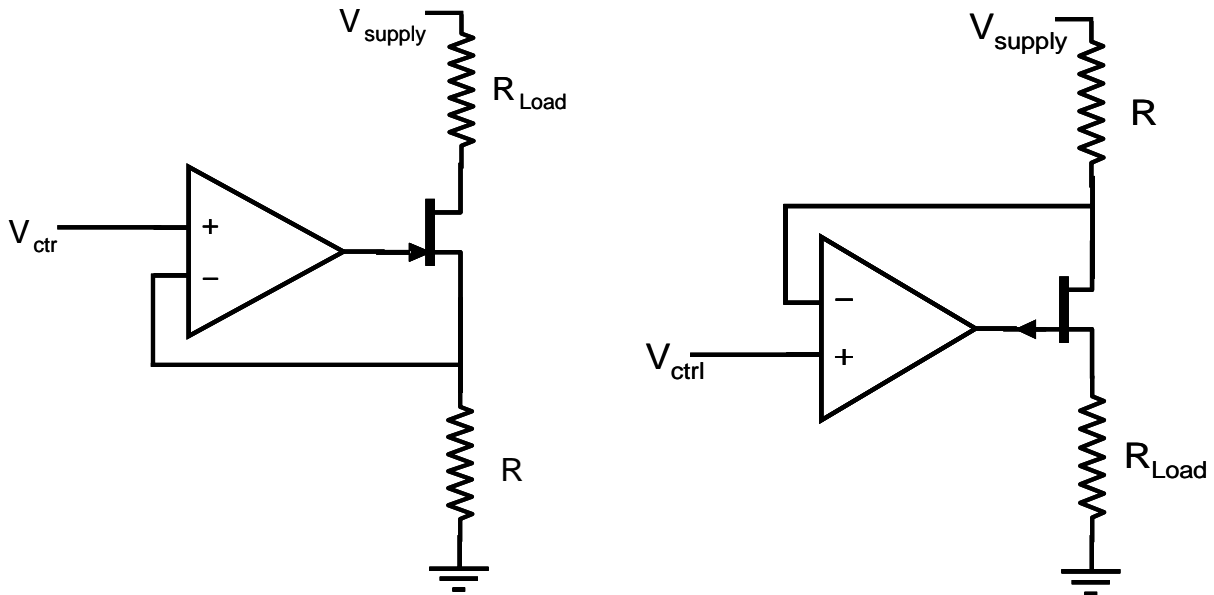
Figure 10.4: N-FET (left) and P-FET (right) constant current sources

C. Op-Amp and Transistor Current Source for Grounded Loads

The two previous circuits are excellent current sources, but they require that the load be at some voltage above ground. Frequently, one side of the load must be grounded and in this case another similar circuit is required. Figure 10.4 (right) shows the circuit

diagram for a constant current source with grounded load based on P-JFET. The P-JFET can be replaced by a PNP or P-MOSFET for high current applications.

III. Op-amp Power Amplifiers

Op-amps are quasi-perfect amplifiers, but they cannot source and sink a lot of current, which is necessary in high power application. The combination of an op-amp with one or more transistors can produce a high-quality amplifier. The high-impedance input of the op-amp provides high fidelity signal measurement and gain, while the output transistors provide high-current. The inclusion of the output transistors in the op-amp feedback loop helps linearize amplifier performance.

A. Op-amp with Emitter-Follower

An op-amp and an NPN emitter-follower (see chapter 6) can be combined to make a class A power amplifier as depicted in figure 10.5 below. Since the NPN transistor can only source current (i.e. the current can only go from *collector* to *emitter*), the input of the op-amp, V_{in}, must still have a positive bias.
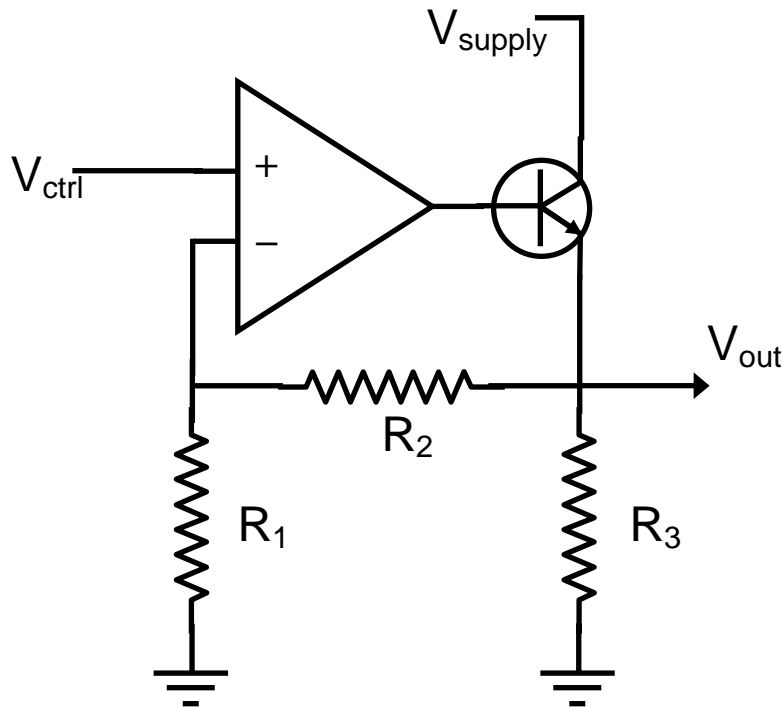
Figure 10.5: Non-inverting amplifier with emitter-follower in feedback loop.

B. Op-amp with Push-Pull Buffer

The push-pull amplifier we saw in chapter 7 can also be incorporated in the feedback loop of an op-amp inverting amplifier to make the class B bipolar power amplifier shown

in figure 10.6 below. The feedback loop automatically removes the cross-over distortion due to the 0.6 V diode drops so long as the slew rate of the op-amp can keep up.
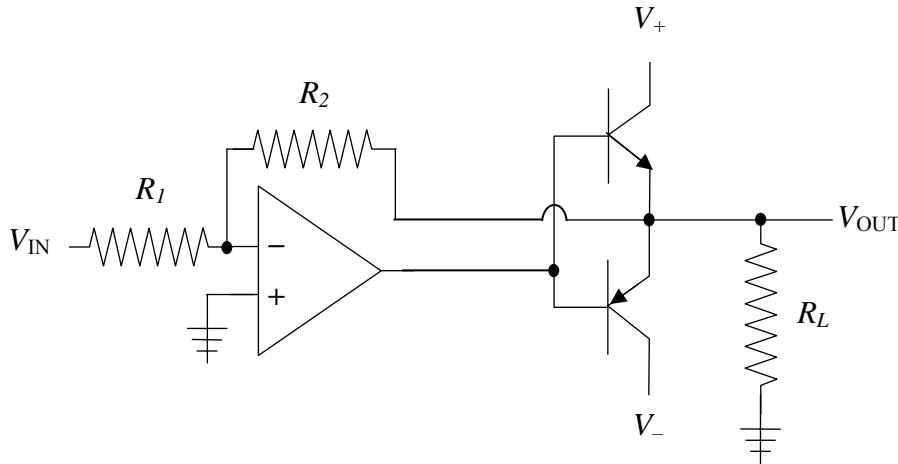
Figure 10.6: Inverting amplifier with a push-pull buffer inside the feedback loop.

IV. Op-amp Rectifiers

If you put a diode in the feedback loop of an op-amp (along with a resistor to let the inverting input float), you can build a rectifier that eliminates the problems for voltages less than 0.6 V as the diode turns on (see figure 10.7).

The improved rectifier circuit (see figure 10.8) keeps the output of the op-amp from being driven to the rails when the circuit is blocking the wrong sign input. Since your op amp has a limited slew rate, this improves performance. It also eliminates the 0.6 V offset that a single diode introduces.
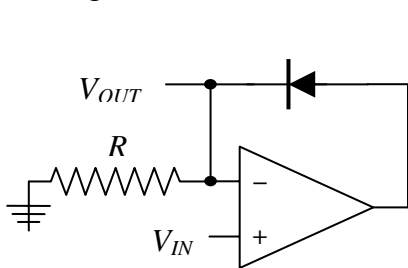
Figure 10.7: Op amp rectifier.
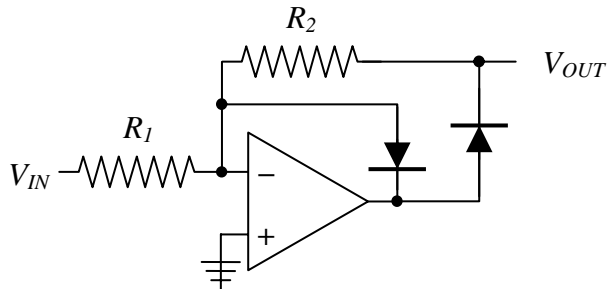
Figure 10.8: Improved op-amp rectifier.

IV. Photodiodes

Photodiodes are a type of diode which produce a reverse bias current when their PN junction is illuminated. The photo-current is proportional to the number of incident photons with quantum efficiencies of 0.5-0.9 photo-electrons per incident photon.

A. High sensitivity photo-diode circuits

The simplest way to read the current on a photodiode is to convert it to a voltage with a resistor as shown in figure 10.9 below. This circuit is very simple, but has the disadvantage that the resistor loads the photodiode which must produce a voltage across the resistor. The photodiode also has a capacitance proportional to its area which will also determine time response ($\tau = RC$): as this simple circuit makes clear, you cannot have high sensitivity and high speed at the same time.
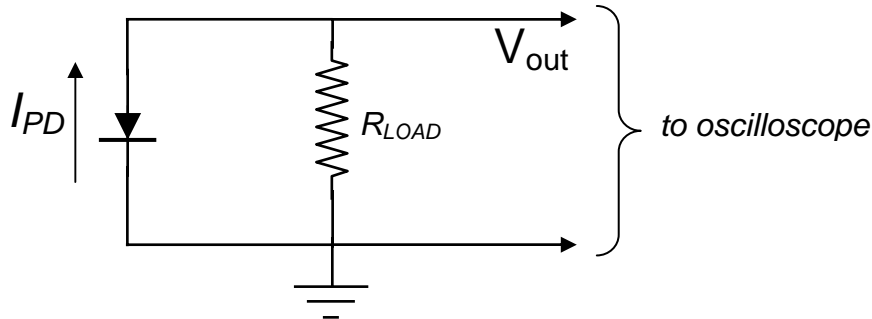
Figure 10.9: Photodiode with a resistive current-to-voltage converter.

As the voltage which the photodiode must produce becomes larger, the output current of the photodiode no longer remains proportional to the light intensity. An op-amp remedies this problem by keeping the voltage across the photodiode equal to zero for all light intensities, while also amplifying the signal in the current-to-voltage converter circuit of figure 10.10.
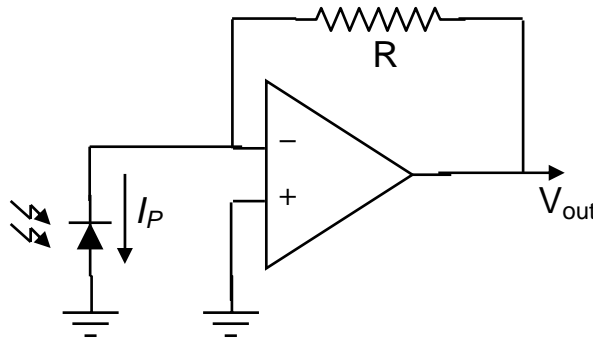
Figure 10.10: Photodiode with an op-amp current-to-voltage inverting amplifier.

B. High speed photo-diode circuits

The PN junction capacitance of the photodiode can be reduced significantly by using a reverse-bias voltage on the photodiode, as shown in the circuit diagram on the right (figure 10.11). Since the capacitance drops, so does the time response (still $\tau = RC$) of the photodiode. In practice, the fastest photodiode have very small detection areas and are

operated in reverse-biased mode with small load resistors. While reverse biasing helps too improve the time response of the photodiode to fast signals, it also reduces its sensitivity to very weak light signals, since the reverse bias voltage also drives a small leakage current across the PN junction. This larger leakage current, or dark current, has noise associated with it which limits the sensitivity of the photodiode at very low light levels.
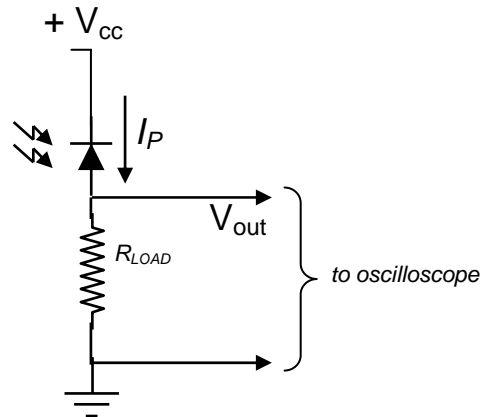
Figure 10.11: Photodiode in reverse-bias configuration with resistive load.

One problem with the simple reverse-bias circuit of figure 12 is that as the photo-current increases at higher light intensities, the effective biasing voltage across the PN junction changes as well, which introduces a non-linear response into the behavior of the photo-detector. An op-amp can be used to keep the photodiode reverse-bias voltage constant for all illumination intensities, while also converting the photo-current to an amplified voltage (see figure 10.12).
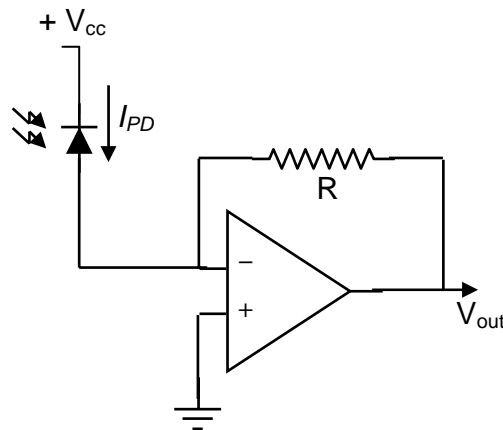
Figure 10.12: Amplified photodiode in reverse-bias configuration

In practice, the capacitance of the photodiode can send the op-amp into oscillation, so a small feedback capacitor is frequently added in parallel with the feedback resistor R.

Chapter 10: Op-amp circuits for detectors, filters, and power applications.

Design Exercises

Design Exercise 10-1: Analyze the push-pull amplifier shown in figure 10.13 below and explain the purpose of the diodes. What are reasonable values for R?
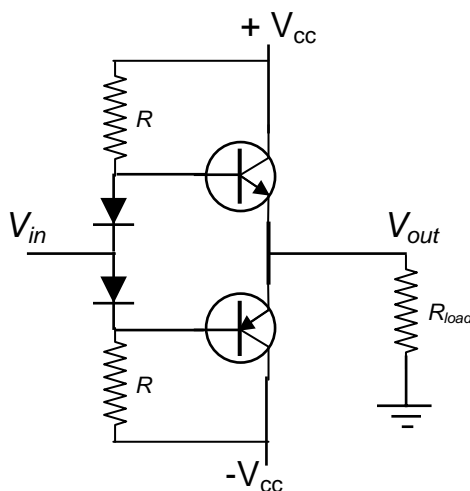
Figure 10.13: Improved push-pull transistor amplifier.

Design Exercise 10-2: Design an op-amp based high-sensitivity photodiode amplifier for a photodiode with a PN capacitance of 150 pF. The amplifer should have a gain of 1 V/μA. What is the response speed (i.e. time constant) of the photodiode-amplifier combination?

Design Exercise 10-3: Design an op-amp based high-speed photodiode amplifier. Use a reverse-bias configuration with a bias voltage of +15 V. We will assume that the biasing reduces the PN capacitance to 40 pF. The amplifier should have a gain of 1 V/μA. What is the response speed (i.e. time constant) of the photodiode-amplifier combination?

Design Exercise 10-4: Design an op-amp-based 10 μA constant current source in which the load is not grounded. The design may include additional semiconductor components which we have previously used in lab, if you wish.

Design Exercise 10-5: The TH10K thermistor is a type of silicon-based resistor which has a resistance of 10 kΩ at 25 °C, but whose resistance (counter-intuitively) changes by -4.4% per °C. Design a voltage divider using the TH10K thermistor and a second resistor such that at room temperature the output voltage of the voltage divider is roughly linear with any change in temperature (i.e. the output voltage function has an inflection point at room temperature). You should keep the power dissipated by the thermistor well below 1 mW to avoid self-heating. Design a circuit in which the output voltage changes by about 0.5 V per deg °C over the range 0 °C – 50 °C and can drive a 5 kΩ load or larger with minimal effect on the output voltage.

Chapter 10: Op-amp circuits for detectors, filters, and power applications.

Lab 10: Op-amps and detectors

This week's lab focuses on using op-amps and detectors together for high-quality measurements. In particular, we will focus on using op-amps with photodiode detectors and thermistor temperature sensors.

1. Construct an LED pulsing light source from a red LED (Light Emitting Diode) and the the function generator. Set the function generator to 5 V pk-pk with the low voltage set to 0 V. Attach the LED directly to the function generator and see if it flashes when the function generator frequency is set to 10 Hz.

2. Use an FDS100 Thorlabs photodiode to construct the circuit of figure 10.9 such that the current-to-voltage conversion is 10 mV/μA. Install the photodiode and the red LED in front of each other to maximize the amount of emitted light incident on the photodiode – *fasten the photodiode and the LED securely so that they will not move*. **Also, you should make sure that the LED is not saturating the photodiode**. Measure the rise time and fall time of the photo-current signal. You may increase the frequency on the function generator.

Assuming that the LED turn-on and turn-off is instantaneous (a high-speed detector gave about 100 ns), determine the capacitance of the photodiode.

3. In the same manner as part 2, construct the circuit of figure 10.11 such that the current-to-voltage conversion is 10 mV/μA and the bias voltage is + 15 V. Measure the rise time and fall time of the photo-current signal. Estimate the capacitance of the photodiode.

4a. In the same manner as part 2, construct the circuit of figure 10.10 or figure 10.12 (you pick) with an LM741CN op-amp and a gain of 10 mV/μA. Measure the rise time and fall time of the photo-current signal, and estimate the effective capacitance of the photodiode.

4b. Replace the LM741CN op-amp with an OP27 or LT1498 op-amp. Do you see any difference? Does the output oscillate? If it does, can you reduce it? Measure the rise time and fall time, and estimate the capacitance of the photodiode.

5. Thermometer

Construct the circuit you designed for Design Exercise 10-5 and roughly estimate the temperature-to-voltage coefficient (V/°C) in order to calibrate you thermometer. Verify that the output voltage can drive load resistors of 5 kΩ or more with little effect on the output voltage.

Construction advice: Your circuit is probably made up of a number of smaller subcircuits, so make and test these sub-units first before hooking them together.

Chapter 10: Op-amp circuits for detectors, filters, and power applications.

Chapter 11: Feedback and PID Control Theory

I. Introduction

Feedback is a mechanism for regulating a physical system so that it maintains a certain state. Feedback works by measuring the current state of a physical system, determining how far the current state is from the desired state, and then automatically applying a control signal to bring the system closer to the desired state. This process is repeated iteratively to bring the system to the desired state and keep it there.

Feedback can be used very effectively to stabilize the state of a system, while also improving its performance: Engineers use feedback to control otherwise unstable designs; op-amps use feedback to stabilize and linearize their gain; and physicists use feedback to stabilize and improve the performance of their instruments.

A. Feedback in engineering

Feedback is ubiquitous in engineering. Its application has led to device features and machines which would not otherwise function. Here are few examples:

Climate control: A sensor measures the temperature and humidity in a room and then heats or cools and humidifies or dehumidifies accordingly.

Automobile cruise control: The car measures its speed and then applies the accelerator or not depending on whether the speed must be increased or decreased to maintain the target speed.

Highly maneuverable fighter jets: The F-16 Falcon fighter jet is an inherently unstable aircraft (i.e. the airframe will not glide on its own). The F-16 does fly because 5 onboard computers constantly measure the aircraft's flight characteristics and then apply corrections to the control surfaces (i.e. rudder, flaps, ailerons, etc…) to keep it from tumbling out of control. The advantage of this technique is that the aircraft has the very rapid response and maneuverability of a naturally unstable airframe, while also being able to fly.

B. Feedback in electronics:

Op-amps use feedback to achieve very high linearity and predictability for their closed-loop gain by sacrificing some of their extremely high open-loop gain.

Another common application of feedback in electronics is in precision, fast- response power supplies. Constant current and constant voltage power supplies which have a high degree of stability use feedback to regulate their current or their voltage, by measuring the current and voltage across a precision shunt resistors and then using feedback to automatically correct for any deviations from the desired output. Feedback also allows the power supply to adjust its voltage or current very quickly and controllably in response to a change in load.

C. Feedback in physics

Feedback has become a familiar tool for experimental physicists to improve the stability of their instruments. In particular, physicists use feedback for precise control of temperature, for stabilizing and cooling particle beams in accelerators, for improving the performance of atomic force microscopes, for locking the optical frequency of lasers to atomic transitions, and referencing quartz oscillators to ground state atomic hyperfine microwave transitions in atomic clocks, to name just a few example.

Temperature control: Many delicate physics devices, such as crystals, lasers, RF oscillators, and amplifiers, require their temperature to be very stable in order to guarantee their performance. For example, the wavelength of diode lasers generally has a temperature dependence on the order of 0.2 nm/°C, but requires a stability of 10^{-6} nm for experiments.

Stochastic cooling: In a particle accelerator, the transverse momentum spread of particles must be reduced to a minimum. The reduced momentum spread increases the particle density, or beam luminosity, and consequently the probability of collisions with a similar counter-propagating particle beam in the detector area. Stochastic cooling works by measuring the transverse positions and momenta of the particles as they pass through a section of the accelerator, and then applying appropriate momentum kicks to some of the particles at other points in the accelerator ring to reduce the overall transverse momentum spread. The process is repeated until the momentum spread is sufficiently reduced. The 1984 Nobel Prize in Physics was awarded in part to Simon van der Meer for his invention of stochastic cooling which contributed to the discovery of the W and Z bosons (weak force mediators) at CERN.

Atomic force microscope: An atomic force microscope uses a very sharp tip (just a few nanometers in size at the very tip) which is scanned back and forth just a few nanometers above the surface to be imaged. Instead of scanning the tip at a constant height above the surface, which could lead to the tip actually running into a bump on the surface, the microscope uses feedback to adjust the tip height such that the force (from the surface atoms) on the tip is constant.

Laser locking: Many experiments in atomic and optical physics require lasers which have a very stable optical frequency. The optical frequency of the laser is locked by measuring the optical frequency difference between the laser and an atomic transition and using feedback to set this difference to a constant value. Lasers can be routinely stabilized with feedback to better than 1 MHz out of 3×10^{14} Hz (about 1 part per billion), though stabilities close to 1 Hz have been reported after heroic efforts.

Atomic clocks: In an atomic clock, the frequency of an RF oscillator (a quartz crystal for example) is compared to that of a ground state atomic hyperfine microwave transition (6.8 or 9.2 GHz). The frequency difference is measured and the frequency of the RF oscillator is corrected by feedback. The process is constantly repeated to eliminate any drift in the frequency of the RF oscillator. Atomic fountain clocks can achieve accuracies in the range of 1 part in 10^{15}, and plans are underway to construct optical atomic clocks with accuracies and stabilities of about 1 part in 10^{18}.

II. Feedback

In this section we introduce the main elements of a generic feedback model.

A. System

Consider a simple system characterized by a single variable *S*. Under normal conditions the system has a steady state value of $S=S_0$ which may vary and drift somewhat over time due to the variation of environmental variables *v* which we cannot measure or are unaware of. We possess a mechanism for measuring the state of the system as well as a control input *u* with which we can use to modify the state S of the system. In summary, the system has the following functional form *S(u; v; t)*. We will make the final assumption that S is monotonic with *u* in the vicinity of S_0 (i.e. that the plot of *S* vs. *u* does not have any maxima or minima, and that *dS/du* is either always positive or always negative).

Figure 11.1 shows a conceptual schematic of the relationship between the system, the variables *u* and *v*, and the measurement of the system state *S*.
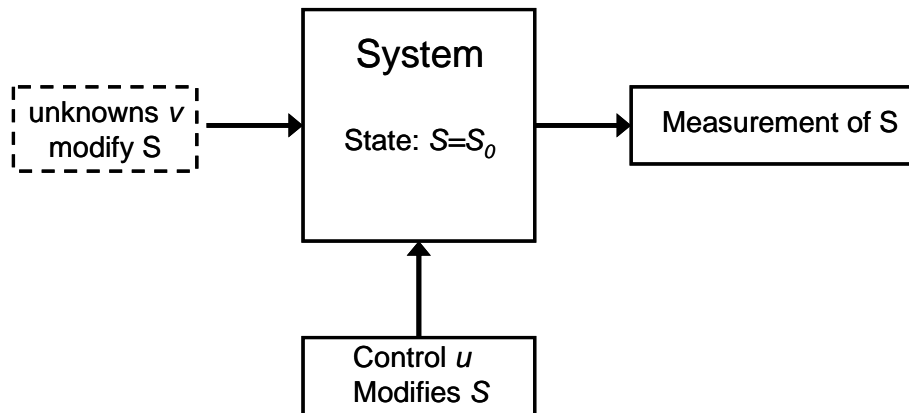
Figure 11.1: Conceptual schematic of system

B. Objective

Our objective is to set or lock the state of the system to a desired value $S=S_d$ and keep it there without letting it drift or vary over time, regardless of variations in the environmental variables *v*.

C. Feedback model

We will set or lock the state of the system to $S=S_d$ with the following procedure (see also figure 11.2):

1. Measure the state S of the system.

2. Determine how far the system is from its desired set point by defining an error variable, $e=S-S_d$.

3. Calculate a trial control value $u=u(e)$.

4. Feed the calculated control value, $u(e)$, back into the control input of the system S.

5. The state of the system changes in response to the change in the control value.

6. Return to step.

If we repeat this feedback cycle indefinitely with an appropriately calculated control value $u(e)$, then the system will converge to the state $S=S_d$ and remain there even under the influence of small changes to other variables (i.e. v) which influence the value of the state S.

This feedback model can be adapted to include several state variables and several feedback variables.
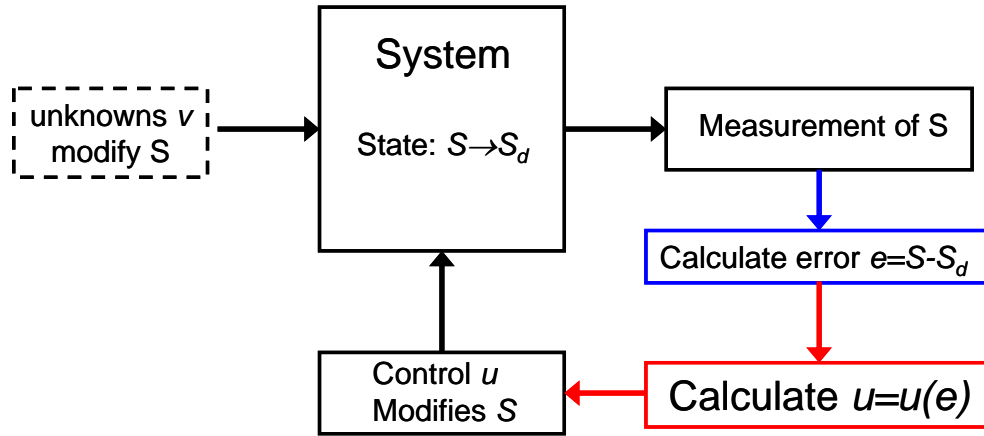
Figure 11.2: Conceptual schematic of system with the feedback loop.

In section III, we discuss a frequently used expression for calculating the feedback control variable $u(e)$.

III. PID Feedback Control

The most popular type of feedback stabilization control, $u(e)$, is Proportional-Integral-Derivative (PID) gain feedback. PID is very effective and easy to implement. The expression for $u(e)$ depends only on the error signal $e=S-S_d$ and is given by

$$u(e;t) = g_P e(t) + g_I \int_0^t e(t)dt + g_D \frac{d}{dt} e(t) \qquad \text{(11.1)}$$

where g_P, g_I, and g_D are respectively the proportional, integral, and derivative gains. We also note that g_P, g_I, and g_D do not have the same units. We will assume for simplicity that g_P is dimensionless in which case $u(e)$ has the same units as S.

A. Time evolution of the system with PID feedback control

We are now in a position to calculate the time evolution of the system under the influence of feedback. Without feedback, the system would remain in the state S_0:

$$S_{no\ feedback}(t) = S_0 \tag{11.2}$$

S_0 may vary in time, but we will ignore this effect until part III.C.

In the presence of feedback, the state of the system at time $t+\Delta t$ (step 5) depends on the state of the system without feedback, S_0, which has been modified by the control input variable $u(e)$. We now make the following simplifying assumption that the control input variable, $u(e)$, "controls" or modifies the state of the system S through the process of addition. In this case, the system state variable S evolves according to the following equation:

$$S(t + \Delta t) = S_0 + u(e;t) \tag{11.3}$$

We can convert this equation to an integro-differential equation, if we assume that the system has a characteristic reponse time τ (small). In this case, equation 3 becomes

$$S(t) + \tau \frac{d}{dt} S(t) = S_0 + g_P e(t) + g_I \int_0^t e(t)dt + g_D \frac{d}{dt} e(t) \tag{11.4}$$

B. Special case: pure proportional gain feedback

As a limiting case we consider pure proportional gain feedback ($g_I=0$ and $g_D=0$). We study this special case, because it is the basis for op-amp feedback and is also the simplest form of feedback. For $g_I=0$ and $g_D=0$, equation 4 becomes

$$S(t) + \tau \frac{d}{dt} S(t) = S_0 + g_P e(t) \tag{11.5}$$

We can solve this 1st order differential equation, for the **initial condition $S(t=0)=S_0$**, with the same technique we used in chapter 3 (equations 17-21). After a little bit of integration and algebra, which is left as an exercise to the reader, we find the following solution:

$$S(t) = \left(S_0 - \frac{S_0 - g_P S_d}{1 - g_P} \right) e^{-\left(\frac{1-g_P}{\tau} \right)t} + \frac{S_0 - g_P S_d}{1 - g_P} \tag{11.6}$$

Equation 11.6 shows that the system will converge to the state $S=(S_0-g_PS_d)/(1-g_P)$ when feedback control is applied, so long as the exponential exponent is negative (i.e. $g_P<1$), otherwise S will diverge. We note that $g_P<0$ corresponds to negative feedback.

Figure 11.3 shows the response of a system for a dimensionless gain of $g_P=-10$ and state values $S_0=0.5$ and $S_d=1$, with time measured in units of τ (the system characteristic response time).
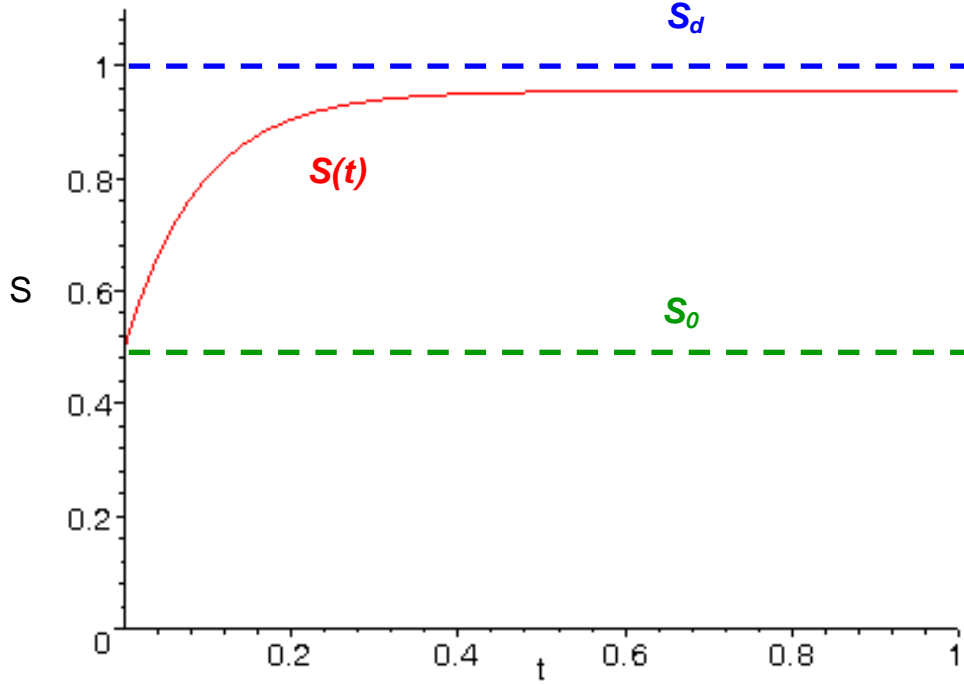

Figure 11.3: System response with pure proportional gain feedback control and parameters $g_P=-10$, $S_0=0.5$, and $S_d=1$. Time is measure in units of τ (the system characteristic response time).

As figure 11.3 makes clear, the system does not converge to the desired state $S=S_d$, though it does reach its final steady state value relatively quickly. If we restrict ourselves to negative feedback, then according to equation 10, the system will converge to the steady state value S_{ss} of

$$S_{ss} = \frac{S_0 - g_P S_d}{1 - g_P}$$

(11.7)

Equation 11.7 indicates that the system can be made to converge to a steady state value S_{ss} which is arbitrarily close to $S=S_d$ just by increasing the gain. In fact, for infinite proportional gain (i.e $g_P \rightarrow -\infty$) the system does converge to $S_{ss} = S_d$. This is the limit in which op-amps feedback operates.

A note of caution: On its own, equation 7 is a little misleading since it would seem to imply that large positive feedback, $g_P \rightarrow -\infty$, would also produce $S_{ss} = S_d$. Of course, this is

not true since according to equation 11.6, the system will never achieve a steady state, but instead will diverge forever.

C. Solution for PI feedback control

A large majority of PID feedback controllers are actually just PI controllers (i.e. proportional and integral gain, but no derivative gain), and so for simplicity we solve equation 11.4 without the derivative gain term ($g_D=0$). The inclusion of the derivative gain term is conceptually simple and follows the same treatment as PI feedback and is left as an exercise to reader. Derivative gain is used to improve the time response of the feedback, so that the system converges more quickly to its steady state value.

With the derivative gain term omitted, equation (11.4) becomes

$$S(t) + \tau \frac{d}{dt} S(t) = S_0 + g_P e(t) + g_I \int_0^t e(t) dt \qquad (11.8)$$

We can convert this integro-differential equation to a 2nd order linear differential equation with constant coefficients by taking the time derivative of equation (11.8) to obtain

$$\frac{d}{dt} S + \tau \frac{d^2}{dt^2} S = g_P \frac{d}{dt} S + g_I S - g_I S_d , \qquad (11.9)$$

where we employed the substitution $e(t)=S-S_d$. After combining similar terms, equation (11.9) becomes

$$\tau \frac{d^2}{dt^2} S + (1 - g_P) \frac{d}{dt} S - g_I S = -g_I S_d \qquad (11.10)$$

Equation 11.10 is an inhomogeneous 2nd order differential equation with constant coefficients. The full solution to equation (7) is given by

$$S(t) = A_+ e^{\lambda_+ t} + A_- e^{\lambda_- t} + S_d \qquad (11.11)$$

$$\lambda_\pm = \frac{(g_P - 1) \pm \sqrt{(g_P - 1)^2 + 4 g_I \tau}}{2\tau} \qquad (11.11a)$$

The first two terms of equation 11.11 represent the homogeneous solution to equation 11.10, while the 3rd term is the inhomogeneous solution to the equation (it does not depend on the initial condtions). A_+ and A_- are constants to be determined from the initial conditions.

Equation 11.11 shows that the system will converge to the state $S=S_d$ when feedback control is applied, so long as λ_+ and λ_- are negative (i.e. negative feedback), otherwise S will diverge (exactly the opposite of what we want to accomplish with feedback).

If we choose $S(t=0)=S_0$ and $dS(t=0)/dt=0$ as our initial conditions, we can calculate the constants A_+ and A_-. After a little bit of algebra, we find that

$$A_+ = -\left(\frac{\lambda_-}{\lambda_- - \lambda_+}\right)(S_d - S_0) \tag{11.12a}$$

$$A_- = \left(\frac{\lambda_+}{\lambda_- - \lambda_+}\right)(S_d - S_0) \tag{11.12b}$$

In figure 11.4, the behavior of the system under PI feedback control is plotted for several different parameters configurations.
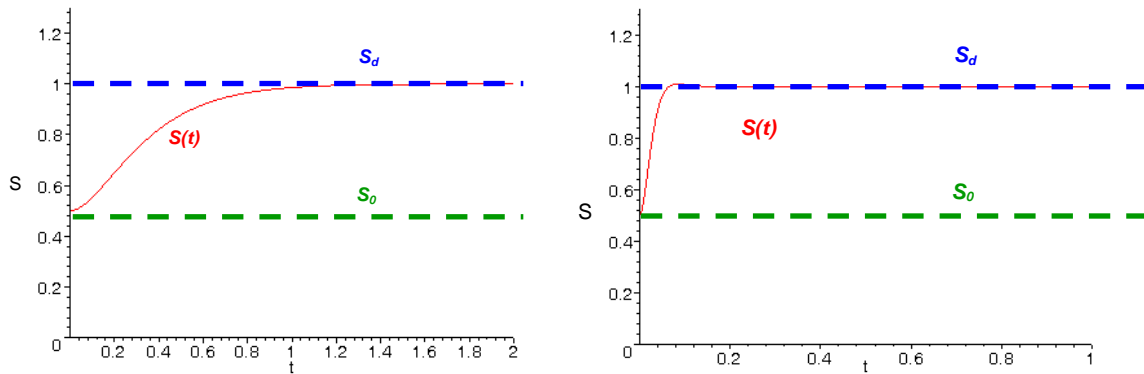
Figure 11.4: Time-evolution of a generic system with PI control feedback for $S_0=0.5$ and $S_d=1$. For the left hand plot the gain parameters are $g_P=-10$, $g_I=-30$, while for the right hand plot the parameters are $g_P=-100$, $g_I=-4000$. The small overshoot in the left hand plot is due to a small imaginary part in the exponential exponent of equation 11.11(a).

The primary purpose of integral gain is to provide essentially infinite gain at DC (0 Hz), which guarantees that $S_{ss}=S_d$, as can be seen in figure 11.4. Figure 11.4 also shows that **the larger the gain, the faster the correction time of the feedback control loop.**

D. Fourier space analysis of noise suppression

One of the primary objectives of feedback is to make the system insensitive to noise on the system state S, so that the system state stays locked to $S=S_d$ regardless of external influences.

In the absence of corrective feedback, external noise will cause the system state to deviate from $S=S_0$. External noise at a frequency ω will cause the system state to oscillate around its natural steady state such that $S=S_0+S_Ncos(\omega t)$, where S_N is the amplitude of the oscillations. Following the standard Fourier space recipe of chapter 3, we replace $cos(\omega t)$ with $exp(i\omega t)$, and then take the real part at the end of our calculations. In essence, we must re-solve equations 11.8 with the following modification:

$$S_0 \rightarrow S_0 + S_N e^{i\omega t} \tag{11.13}$$

Using the substitution of equation 11.13, equation 11.10 becomes

$$\tau \frac{d^2}{dt^2} S + (1 - g_P) \frac{d}{dt} S - g_I S = i\omega S_N e^{i\omega t} - g_I S_d \tag{11.14}$$

Equation 11.14 has the same homogeneous solution as equation 11.10, but the inhomogeneous solution, $S_{ih}(t)$, differs and is given by the following expression

$$S_{ih}(t) = \frac{i\omega}{(1 - g_P)i\omega - \tau\omega^2 - g_I} S_N e^{i\omega t} + S_d \tag{11.15}$$

We see that the noise term is present in the inhomogeneous solution, but with an additional factor modifying the amplitude of the noise. In the case of negative feedback the modulus of this suppression factor, which we will call A_N, is always less than unity and is given by the following expression

$$A_N = \frac{\omega}{\sqrt{(\tau\omega^2 + g_I)^2 + (1 - g_P)\omega^2}} \tag{11.16}$$

The plot in figure 11.5 shows the dependence of the suppression factor, A_N, on frequency for different feedback schemes. The plot shows that a combination of proportional and integral control gives the best suppression of noise, except in the vicinity of the "resonant" frequency $\omega = \sqrt{-g_I/\tau}$. The high frequency drop-off of the suppression factor is not due to feedback but simply the natural response time τ of the system which also suppresses noise.
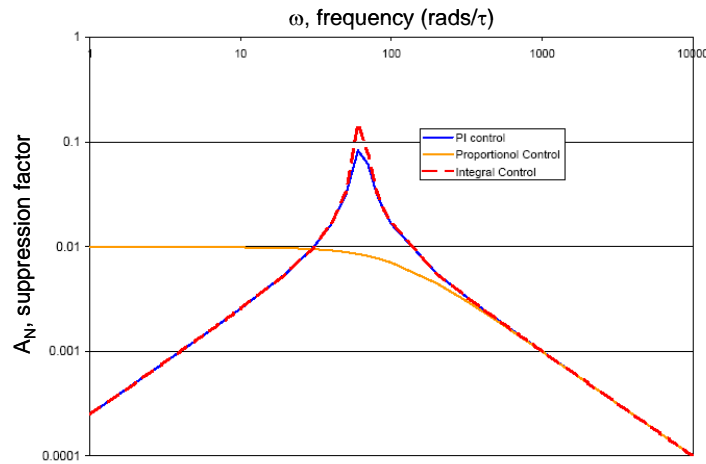
Figure 11.5: Comparison of the suppression factor, A_N, for different feedback schemes. The feedback control loop parameters are $g_P = -100$ and $g_I = -4000$.

IV. Reality

In practice, feedback is not quite as straightforward as presented in the previous section.

A. Gain vs, Frequency

In the theoretical treatment of part III, we assumed that the proportional gain was independent of frequency. In practice, gain will generally fall off at higher frequencies due to natural low-pass RC filtering in an amplifier and the larger circuit.

As an example, figure 11.6 shows a plot of the open-loop gain of an op-amp as function of frequency, which has a clear drop-off in gain at higher frequencies.
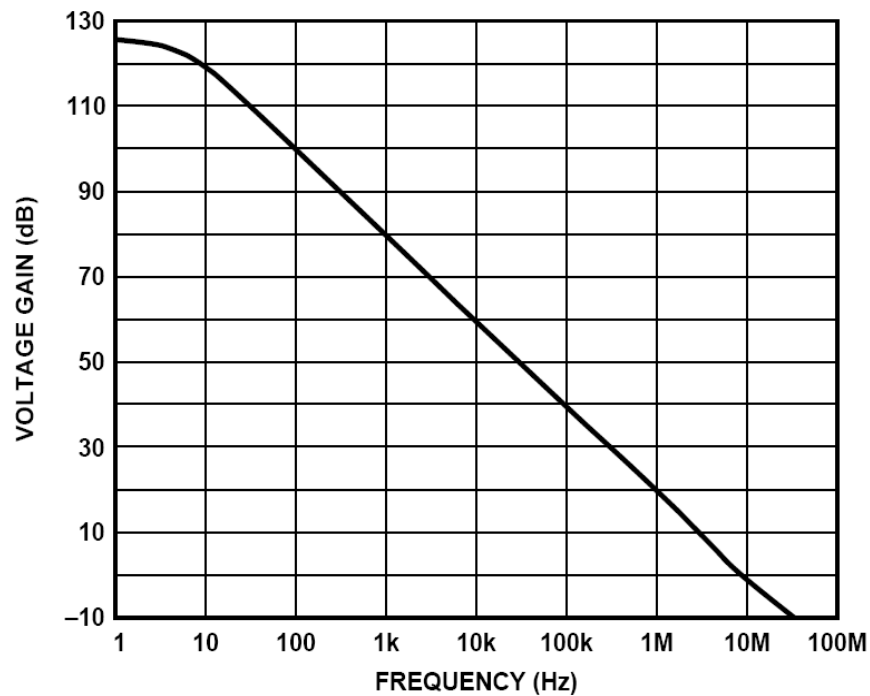
Figure 11.6: Open-loop gain of the OP27 op-amp (Analog Devices OP27 datasheet revision F, p. 10 (2006)).

B. Phase shifts and positive feedback

The natural or stray RC filtering of an amplifier not only rolls off the gain at high frequencies, but also introduces a $-\pi/2$ phase shift. If the feedback loop has a second stray unintentional RC filter present (for example, the natural time response of the system), then a second $-\pi/2$ phase shift is introduced. If the feedback gain is larger than 1 at the frequency at which the total accumulated phase is $-\pi$, then the feedback loop goes into positive feedback which causes the state of the system to diverge or sometimes oscillate out of control.

C. Stray RC positive feedback compensation

One way to avoid having the system go into positive feedback is to purposely introduce an additional RC low-pass filter into the feedback loop. If this RC filter has a f_{3dB} frequency which is sufficiently smaller than the frequency at which the positive feedback occurs then the attenuation of the filter can bring the gain below 1 when the $-\pi$ phase shift occurs. This way the feedback loop will no longer go into positive feedback above a certain frequency (of course there will not be any noise suppression or feedback action above this frequency either).

Design Exercises:

Design exercise 11-1: Consider an LED facing a photodiode in a manner similar to what you did last week in lab 10. Design a circuit which will maintain a constant optical power incident on the photodiode, even in the presence of external fluctuations in the room lighting. You should use the op-amp circuit of figure 10.10 and use PI feedback control to stabilize the intensity of the LED. Your circuit should be able to provide at least 10 mA at ~2 V to the LED.

Design exercise 11-2: **The non-inverting op-amp amplifier with finite open-loop gain.**

In this exercise you will NOT use the op-amp golden rules to solve the problem, unless explicitly indicated.
Consider the non-inverting op-amp amplifier in the circuit below.
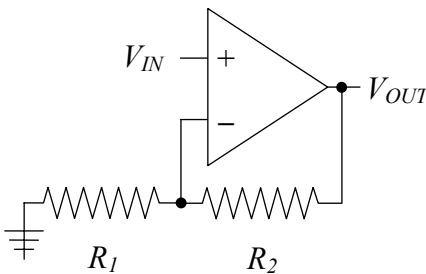
In the following parts, V_+ and V_- refer respectively to the voltages at the non-inverting and inverting terminals of the op-amp. The open-loop gain of the op-amp is A.

a. Write down the fundamental op-amp relation between V_+, V_-, V_{out}, and A when no feedback is present (i.e. when R_1 and R_2 are not present).

b. Assuming that the V_- input draws no current, derive an expression for V_- in terms of V_{out}, R_1, and R_2.

c. Obtain an expression for V_{out} in terms of V_{in}, A, R_1, and R_2, and determine the gain G of the amplifier. Calculate V_{out} in the limit of A $\rightarrow$ +∞. Calculate V_{out} in the limit A $\rightarrow$ -∞ and comment on its physical meaning.

d. Suppose that the open-loop gain, A, is not very constant with frequency and changes by ΔA between frequencies f_1 and f_2. Derive an expression for the resulting relative variation in the amplifier gain ΔG/G in terms of the relative variation in the open-loop gain ΔA/A. Calculate ΔG/G and ΔA/A for A=10^6, ΔA=10^5, R_2=100 kΩ, and R_1=10 kΩ.

e. Most op-amps feature a significant drop-off in their open-loop gain at frequencies above ~10 Hz. The drop-off follows a well established curve given by Af=constant, where the constant is called the gain-bandwidth product (f is frequency in Hz). The gain-bandwidth product of the OP-27 is 8 MHz. On a same log-log graph, plot the open-loop gain A vs frequency and the closed loop gain G vs. frequency (for R_2=100 kΩ and R_1=10 kΩ) for an OP-27-based non-inverting amplifier.

Lab 11: PI feedback control

This week's lab focuses on the use of PI feedback control to stabilize the incident light on a photodiode. We also introduce the Peltier thermoelectric cooler which is frequently used for precision temperature control in electronic circuits.

1. PI feedback control of an LED *(2.5 hours … infinite if you are not prepared)*
Construct the circuit we used in Lab 10, part 4b. Use your circuit design from design exercise 11-1 to regulate the optical power incident on the photodiode. The circuit should be such that the incident optical power can be set with some control resistor.

a. Determine experimentally the integral and proportional gain that you need in order for the feedback to function properly.

b. Use a second LED attached to a square-wave generator to see how fast your circuit can respond and modify the intensity of its LED to keep incident optical power on the photodiode constant. Adjust the integral and proportional gain to optimize the response time of your feedback circuit. What is the fastest response time that you can obtain with your circuit? How does the feedback control respond to a step function change in incident optical power from an external source (an oscilloscope plot will suffice)?

c. If you increase the proportional gain sufficiently, the feedback loop will go into positive feedback and start to oscillate uncontrollably. What is the frequency of this oscillation?

Recommendations:
1. Choose the desired light level: measure the output of the photodiode plus op-amp detector circuit (with LED on at about desired level), and set V_{ctrl} (i.e. S_d) to about this voltage value.
2. Integrator starting values: $R = 1 \text{ } k\Omega$, $C = 100 \text{ } nF$, $R_{feedback} = 10 \text{ } M\Omega$.
3. Proportional gain range: $0 < g_p < 10$ (adjustable with a potentiometer).
4. Procedure: Start off with only integral gain (zero out or remove proportional gain), and then once you get negative feedback, you can slowly add in the proportional gain to optimize the response.

2. Peltier thermoelectric cooler *(0.25 hours)*

A Peltier thermoelectric cooler (TEC) is a two-wire device which consists of two ceramic plates separated by a bi-metal junction. The TEC will cool one plate by removing heat and dumping it into the other plate (and thus heating it). Verify that if the current is reversed, then the cooling plate becomes the heating plate and vice versa.

Chapter 12:

Electronic Circuit Simulation and Layout Software

In this chapter, we introduce the use of analog circuit simulation software and circuit layout software.

I. Introduction

So far we have designed all of our circuits by studying basic electronic subcircuit building blocks and then constructing larger circuits from these. We have designed our circuits by calculating their steady state behavior, as well as their response to small AC (sine wave) signal deviations from the quiescent state. While this method is useful for coming up with the overall design of a circuit, it is a slow and limited method for predicting the ideal theoretical behavior of a circuit under all experimental conditions.

A. Computer-based analog circuit simulators

Computer software circuit simulators are very good at calculating ideal theoretical behavior from Kirchhoff's Laws. While circuit simulators will not help you come up with the insight or the creativity to design a good circuit, they are very useful for helping to elucidate quickly the benefits and disadvantages of one circuit design against another. Generally, you can simulate a circuit much more quickly than you can build and test it on a breadboard (after a little practice, of course). The circuit simulator also allows you to try out many variations on a circuit relatively quickly.

The industry standard analog circuit simulation software is SPICE (Simulation Program with Integrated Circuit Emphasis), which was originally developed at UC Berkeley during the 1970's and early 1980's. SPICE (v2G.6) is the basis for many commercial computer software programs. These programs provide the GUI (Graphical User Interface), but use the SPICE (or WinSPICE) simulation engine to perform all the circuit calculations.

SPICE does not simulate the electromagnetic fields in a circuit since these depend explicitly on the layout of the circuit. The results of SPICE can be trusted up to the low MHz range, but should be treated with suspicion for higher frequencies.

In this course, we will use the commercial software 5Spice (free for non-commercial use) available from www.5spice.com.

B. Computer-based circuit layout editor

In the electronics labs, you have designed the layout of your breadboard circuits on the fly. In a professional setting, the layout of a circuit determines its compactness, ease of use (and debugging), cost, longevity, and its performance (especially at high frequencies). A number of programs exist to help with circuit layout. In fact in industry, most electronics engineers will design an abstract circuit with a circuit simulator and then

use a software package to layout the actual circuit on a PCB (Printed Circuit Board). The PCB layout design is then turned into an industry standard Gerber file which is sent to a PCB production company. A prototype will be assembled and tested at the engineering company from a PCB board and electronics components, and once it is tested the entire production is usually contracted out to a third company.

In a physics research lab, the production is done in house, generally. The use of professionally made PCBs is relatively common, since it generally results in a reproducible circuit, which is likely to work better at high frequencies than one put together with wires or prototyping boards.

In this course, we will use the commercial software Eagle v4.16r2 by CadSoft (free for non-commercial use) available from www.cadsoftusa.com.

II. 5Spice v1.22

Circuit simulation is a two step process. In the first step you must construct the actual circuit diagram using wires and electronics components (i.e. resistor, capacitors, inductor, diodes, integrated circuits, etc …). In the second step, you vary the inputs to your circuit and see how it affects the circuit operation and the outputs.

The program is relatively easy to use. When you start the program you will get a blank schematic drawing canvas. The important GUI elements (i.e. the ones you will use the most) of the main screen are shown in figure 12.1 below.
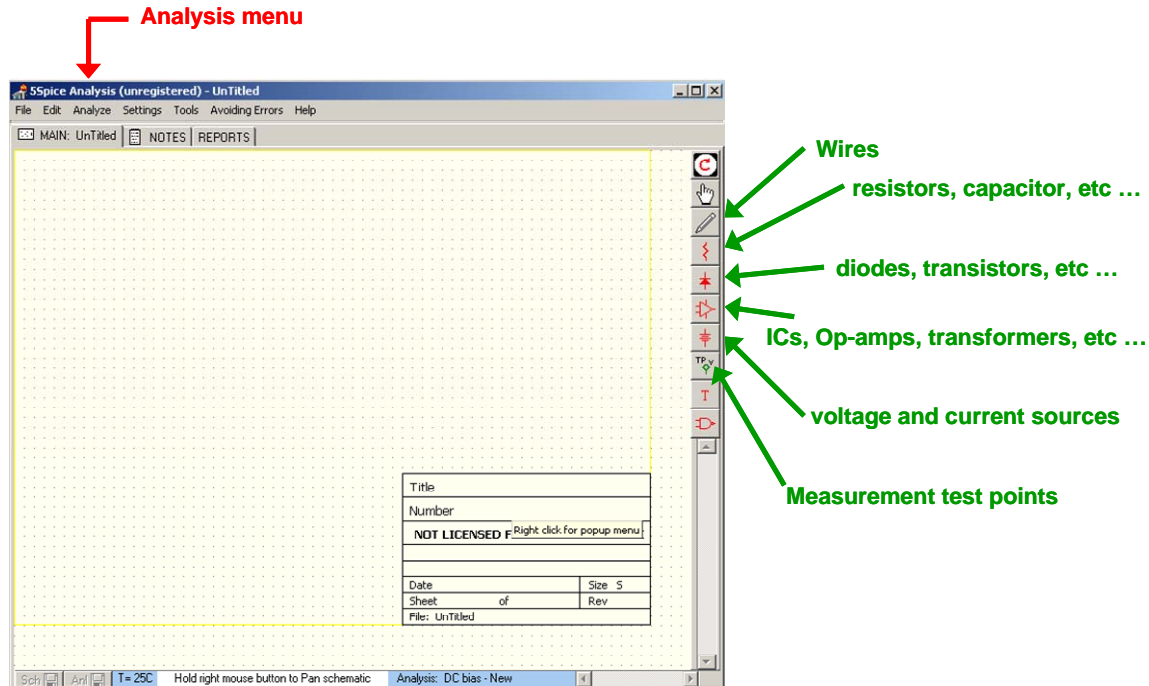
Figure 12.1: 5Spice start screen with frequently used icons and menus

The simplest way to introduce 5Spice is with an example, so we will make and analyze a *gain=-10* inverting amplifier based on a LM741CN op-amp, which is shown in figure 12.2 below.
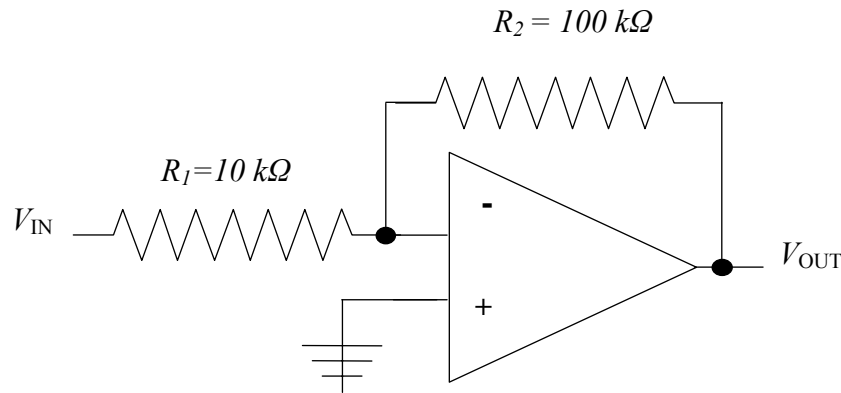
Figure 12.2: Gain=-10 Inverting Amplifier.

A. Circuit diagram

The first step is to build the circuit diagram of figure 12.2. We start with the central component, the LM741CN op-amp. We insert the LM741CN op-amp by going to the op-amp icon and selecting the op-amp sub-circuit element which we then drag-and-drop onto the schematic canvas as shown in figure 12.3.
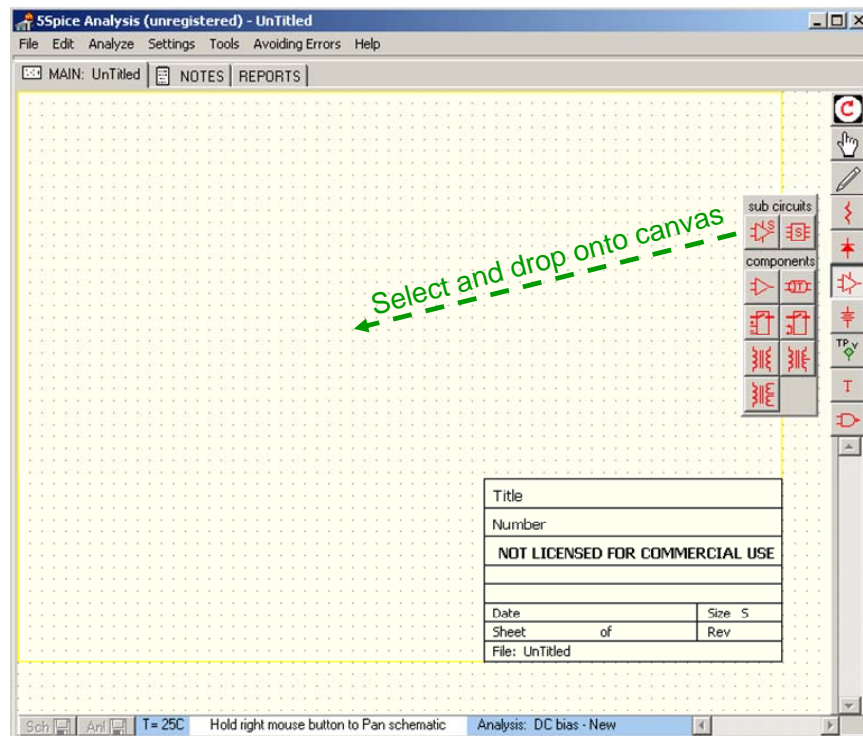
Figure 12.3: We select the op-amp sub-circuit and drop onto the schematic canvas.

The schematic canvas now has a single op-amp in the middle of it. Figure 12.4, below, shows the schematic with a single unidentified op-amp.
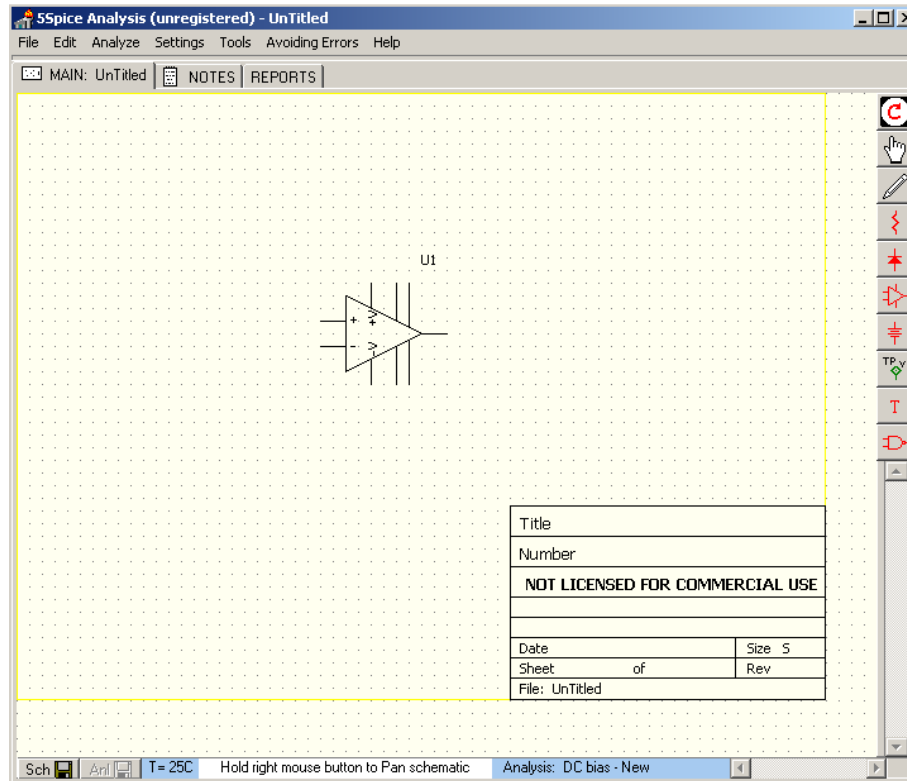
Figure 12.4: Schematic canvas with single unidentified op-amp sub-circuit.

We must now identify the op-amp so that 5Spice knows how to simulate it properly. We give the op-amp the identity of the LM741CN op-amp by finding it in the op-amp library. You can look up the LM741CN op-amp in the sub-circuits library by right-clicking on the op-amp component image and selecting the "Edit Parameter" menu listing as shown in figure 12.5 below.

The "Edit Parameter" selection calls up the sub-circuit library, where you can search and choose the desired op-amp, as shown in figure 12.6. In our case, the only op-amp sub-circuit available is the LM741, so we select this one. If you cannot find the SPICE model for the\component that you are looking for in the library, you can generally find it on the manufacturer's website. If not then you must simulate with another component.
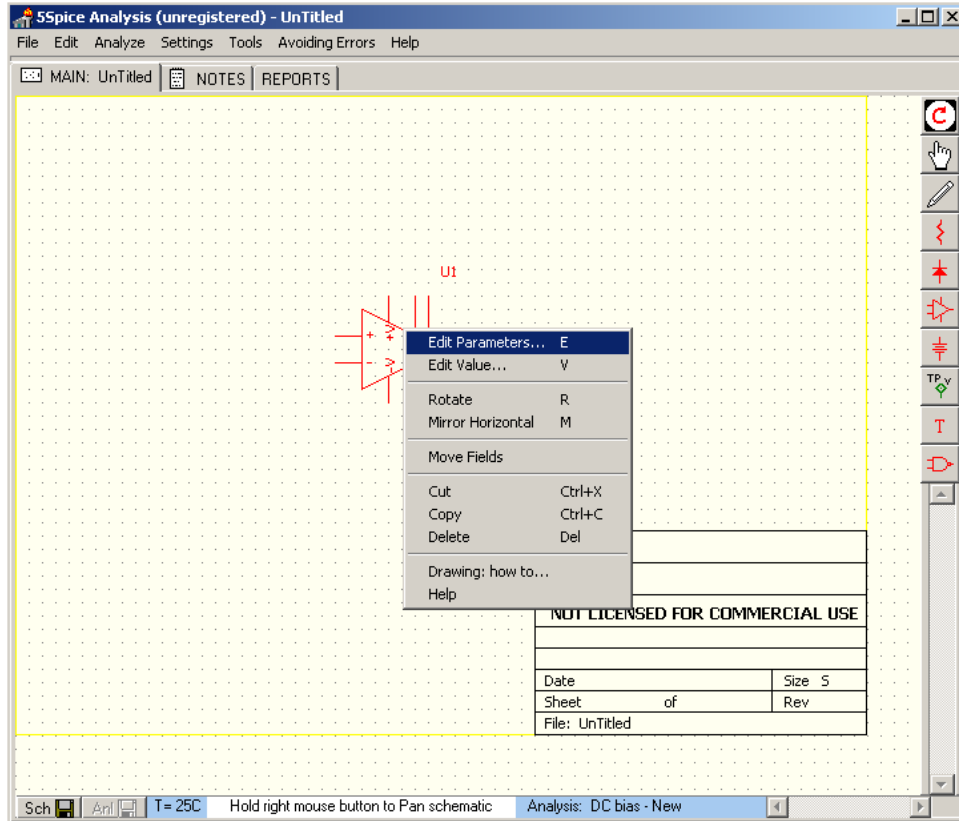
Figure 12.5: Right-click the op-amp image and select "Edit Parameters" to look up the SPICE model for the LM741 op-amp.
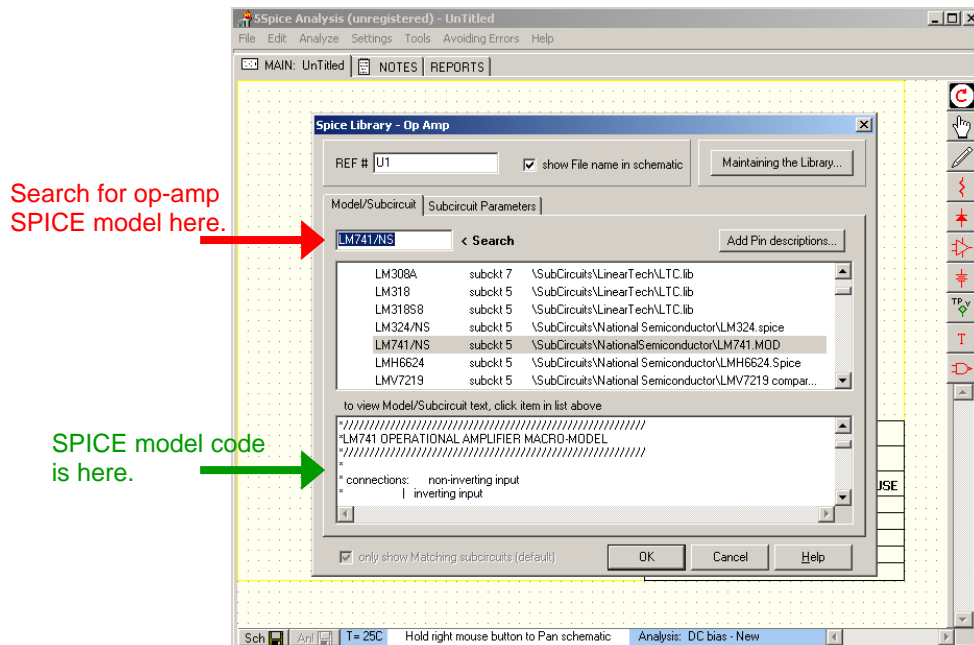
Figure 12.6: Find the SPICE model for the LM741 op-amp in the sub-circuit library.

The op-amp is now identified as an LM741 op-amp, and 5Spice will simulate it as such.

We must now construct the rest of the circuit. We add the resistors from the passive components icon menu on the left, using the drag-and-drop technique. When we put them on the schematic canvas, they are vertical so we rotate them by right-clicking on the resistor component image and selecting the "rotate" menu listing (see figure 12.7 below).
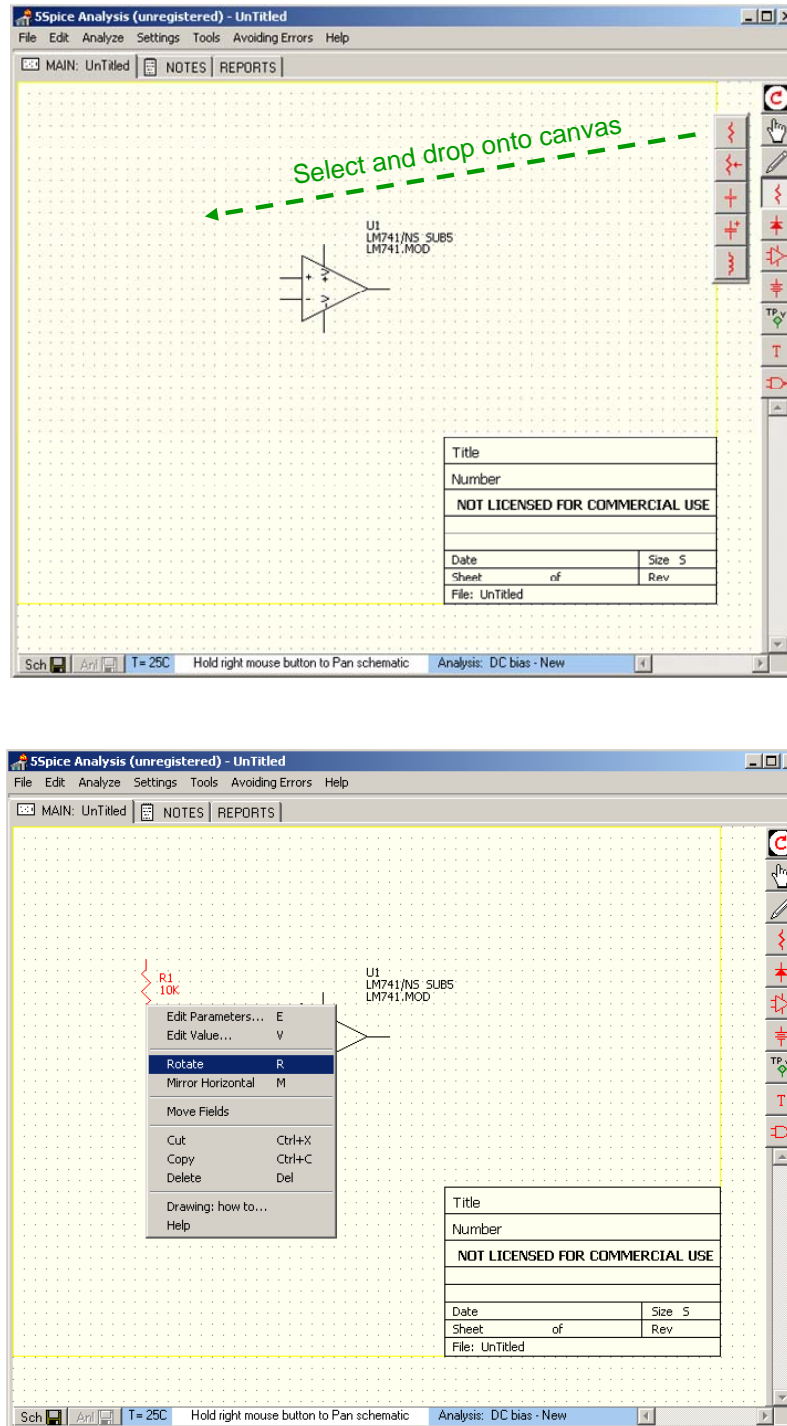
Figure 12.7: Inserting a resistor onto the canvas and rotating it (vertical to horizontal).

We now "rotate" and "mirror" flip the op-amp component image using the same technique so that it is in its traditional orientation. We can now copy and paste the resistor to make a second one, which we can then move around to start constructing our circuit diagram, which is shown in figure 12.8, below.
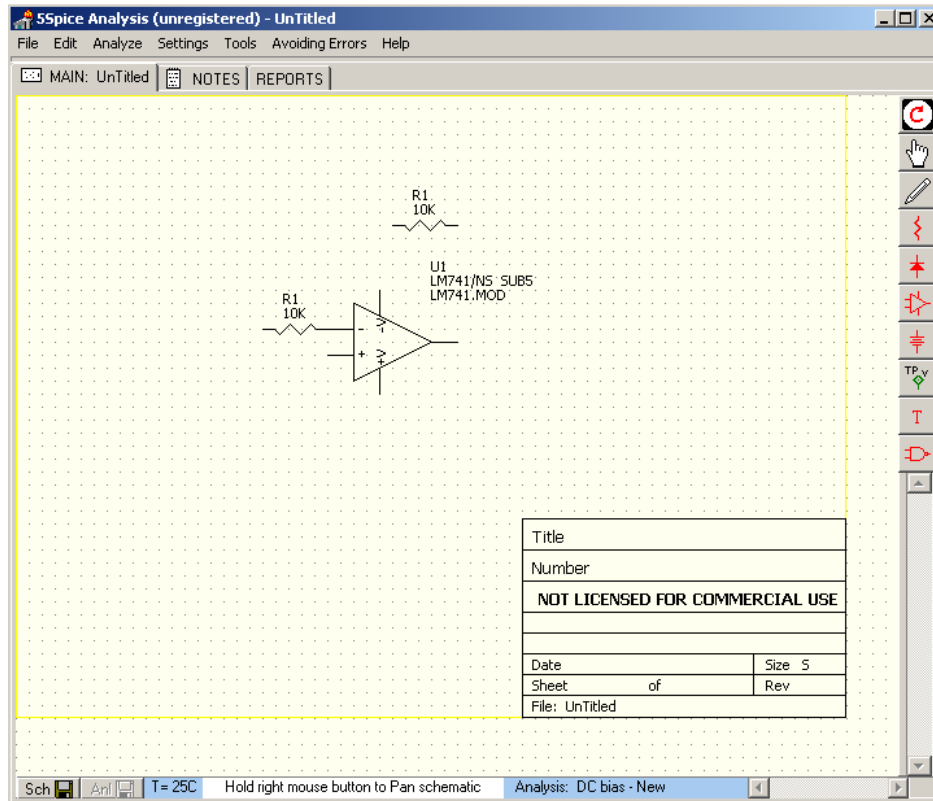
Figure 12.8: LM741 op-amp and two 10 kΩ resistors.

We now convert the top most 10 kΩ resistor to a 100 kΩ resistor by left-double-clicking the resistor component image or right-clicking and choosing the "Edit Parameter" menu listing. We adjust the "10K" value to "100K" in the resistor parameter window, as shown in figure 12.9 below.
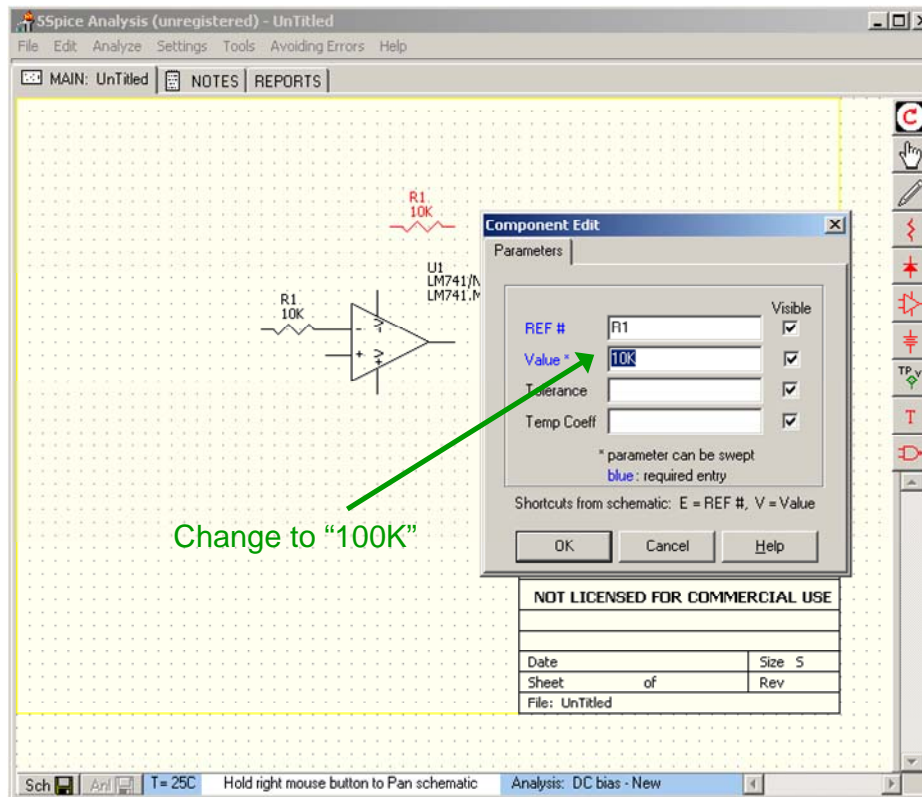
Figure 12.9: Adjusting the resistor value from 10 kΩ to 100 kΩ.

We only need to add the wires in to make our inverting amplifier. We do this by selecting the "wire" from the wire drawing icon menu, as shown in figure 12.10.
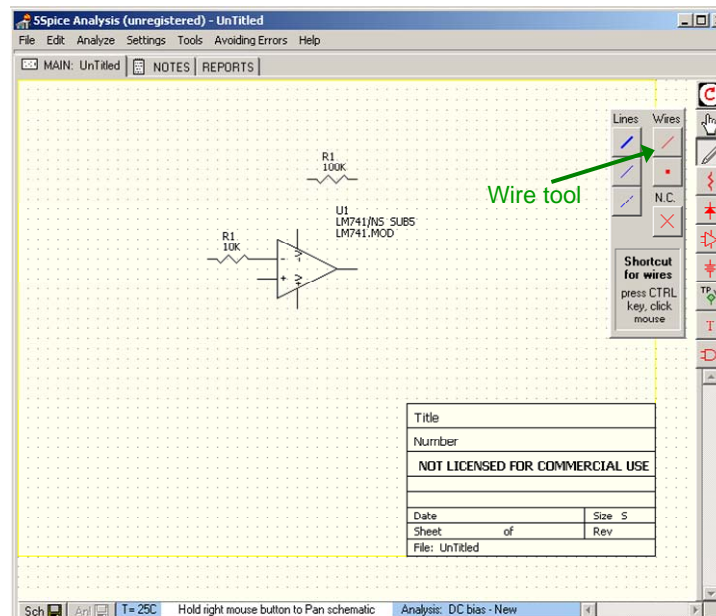
Figure 12.10: We select the wire drawing tool.

We draw in the wires for the inverting amplifier using the mouse cursor. You can stop a wire trace by hitting the "esc" (escape) key.

We are now ready to attach grounds and the positive and negative supply voltages for the op-amp. We do this with "power" icon menu as shown in figure 12.11 below. We drag-and-drop in the ground for the "+" terminal of the op-amp and the positive and negative supply voltages for the op-amp also.
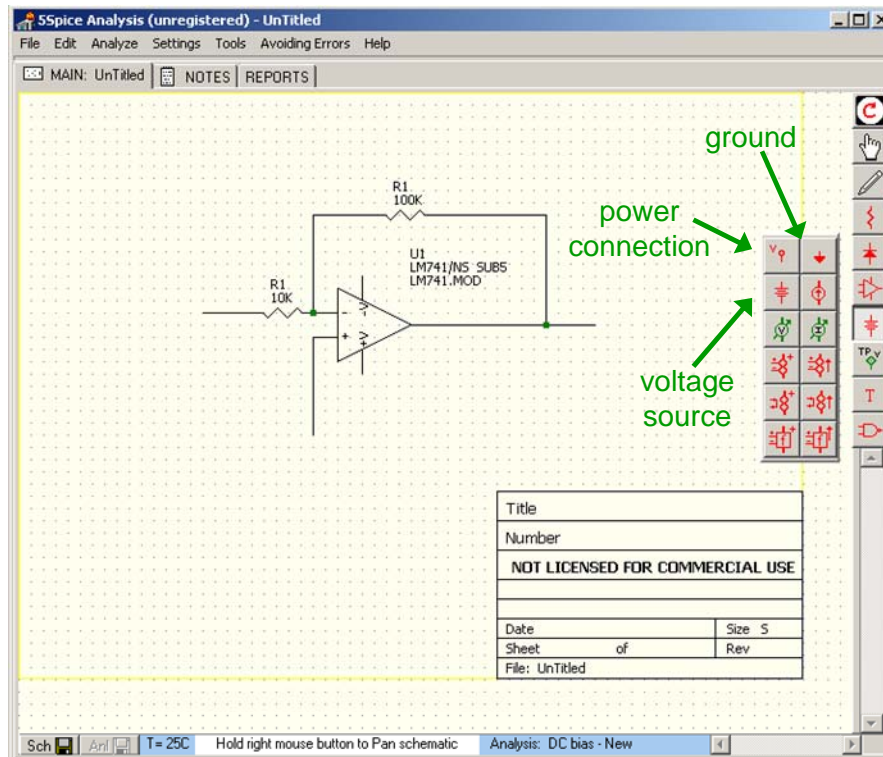
Figure 12.11: The "power" icon menu

We attach a +15 V supply and -15V supply to the op-amp and also connect the op-amp to ground, which then gives the following schematic in figure 12.12 below.
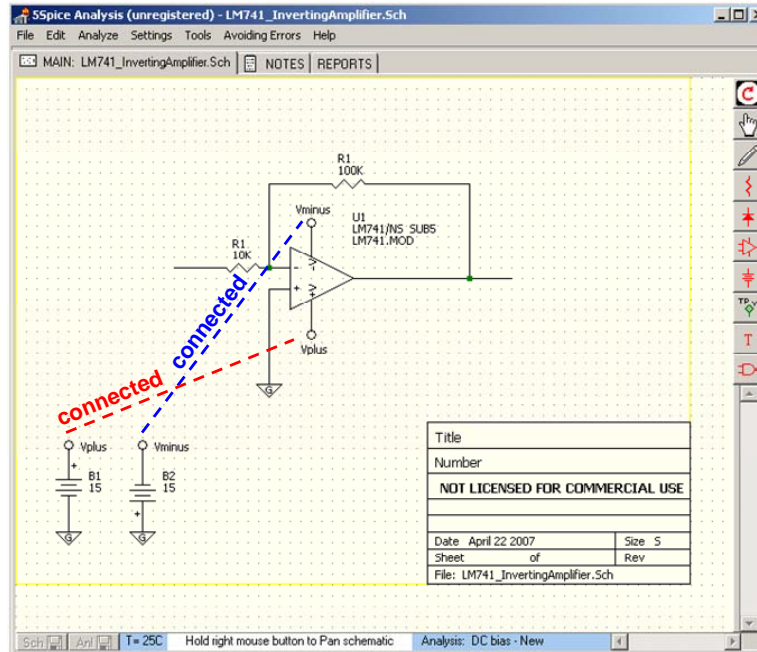
Figure 12.12: The power and ground connections for the inverting amplifier circuit.

The last additions to the circuit diagram are the input and output signals: we add a voltage signal source for input and insert a 1 kΩ load resistor at the output along with a "voltage test point". The final circuit diagram is shown in figure 12.13 below.
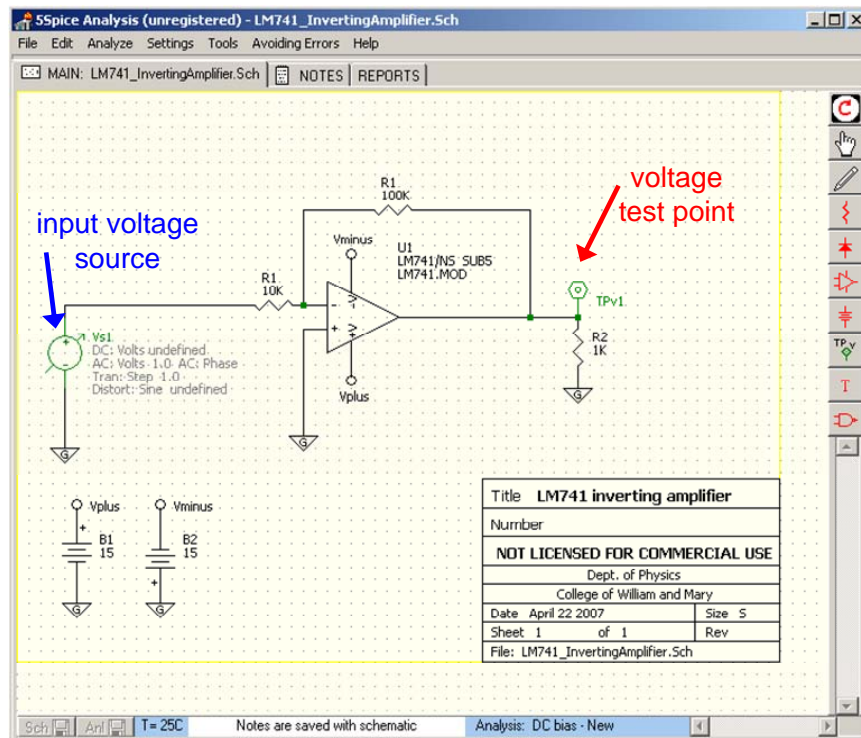
Figure 12.13: Final circuit diagram schematic

B. Circuit analysis

We are now ready to analyze the ideal theoretical performance of our LM741 inverting amplifier circuit. All circuit analysis starts with the "analysis dialog" which you can access through the "analysis menu" shown below in figure 12.14.
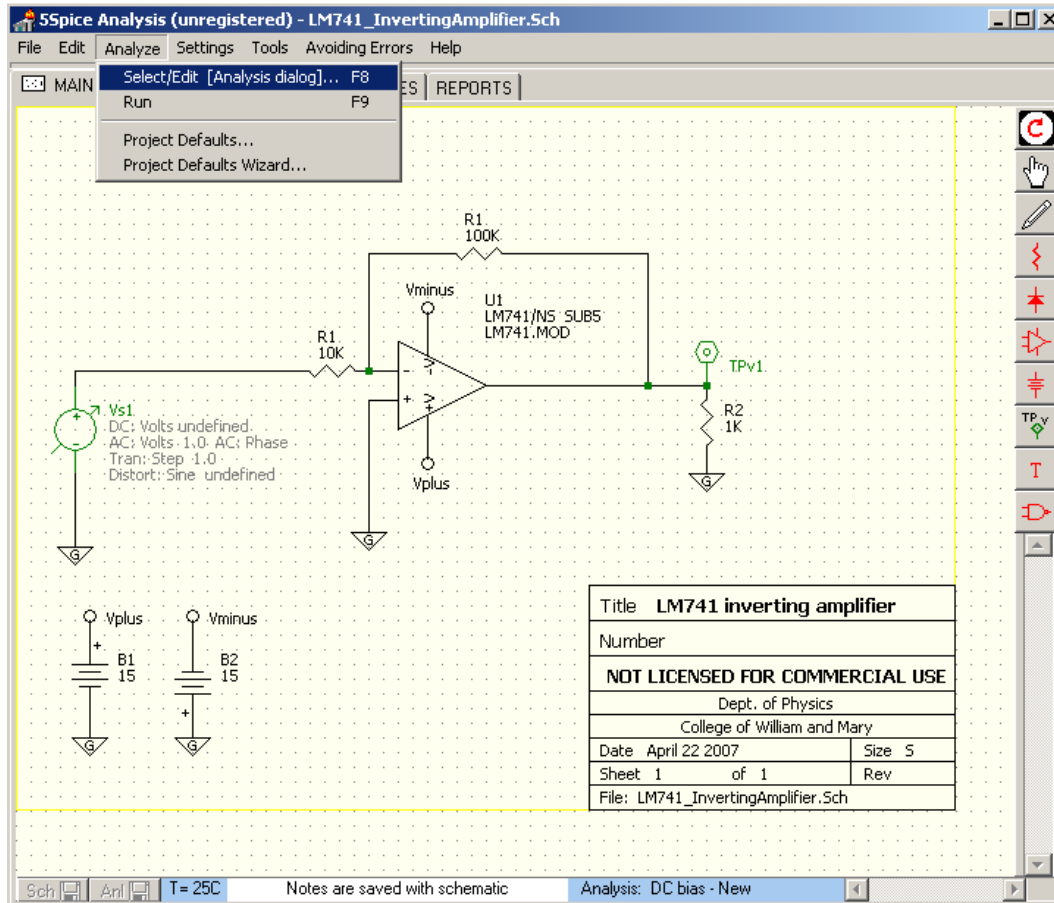
Figure 12.14: The analysis menu

The "analysis dialog" allows you to perform three principal types of analysis on the a circuit: i) "DC bias", ii) "AC", and iii) "Transient". The analysis type can be chosen in the "analysis" tab of the "analysis dialog" as shown in figure 12.15 below.
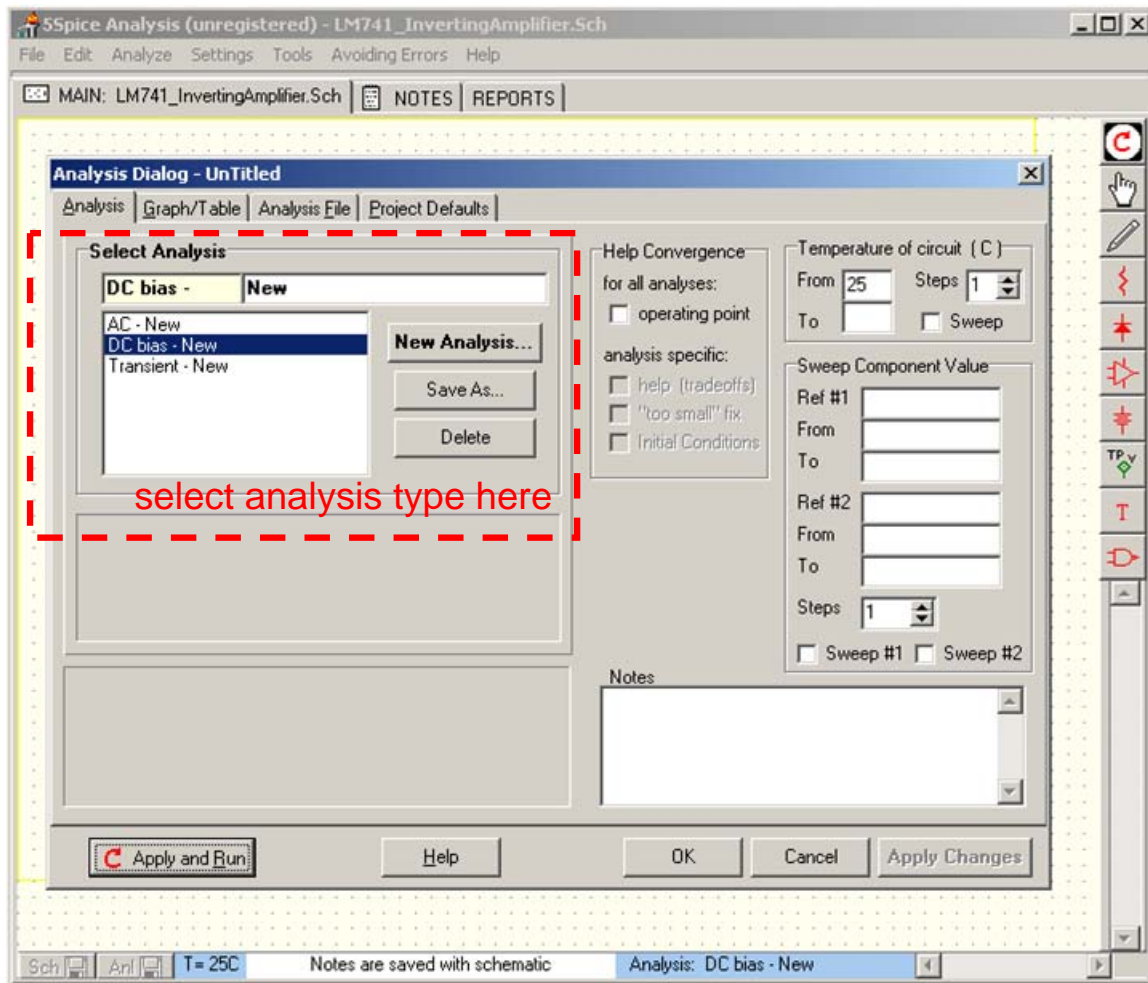
Figure 12.15: The analysis dialog is used to select the type of analysis to be performed.

We now review the three types of analysis.

i. DC bias analysis

You use this analysis to determine the steady-state or quiescent operation of a circuit. This type of analysis does not produce a plot. When we apply it to our inverting amplifier circuit by clicking the "Apply and Run" button, 5Spice calculates the DC voltages and currents and sends the output to the "DC Bias" tab of the main window as shown in figure 12.16 below.
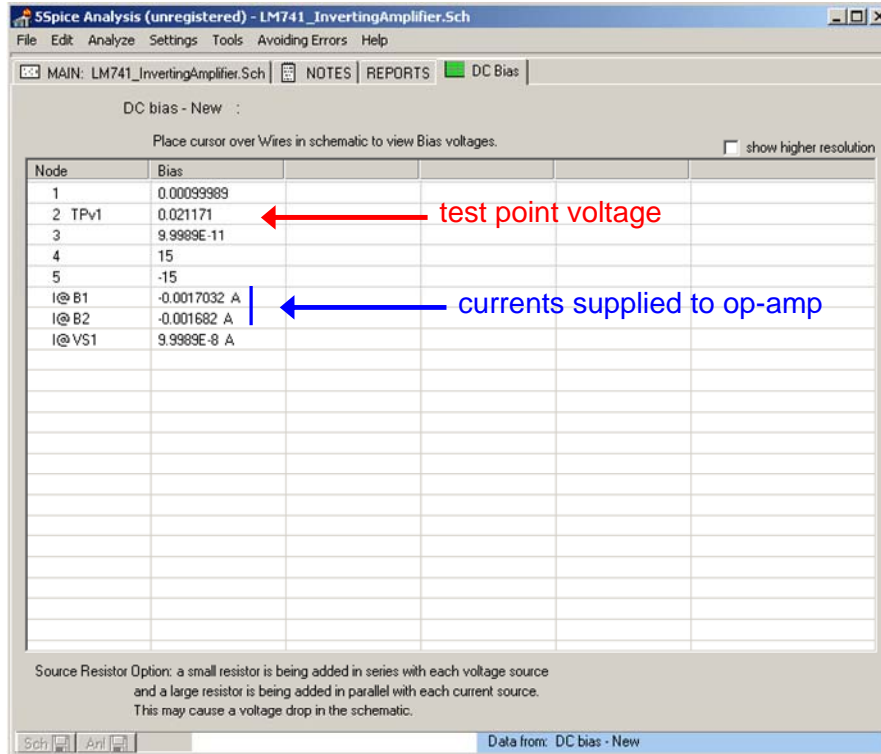
Figure 12.16: Results of the "DC bias" analysis in the "DC Bias" tab of the main window

ii. AC analysis

We use the "analysis dialog" to setup an AC analysis as shown in figure 12.17 below. The AC analysis consists of sending an AC signal into the input, determined by the voltage signal source in the circuit diagram, and scanning the frequency to determine the frequency response of the circuit output or at any other test point. In our case, we scan the frequency of the input from 10 Hz to 10 MHz. The AC analysis is done in frequency space (i.e. Fourier space).
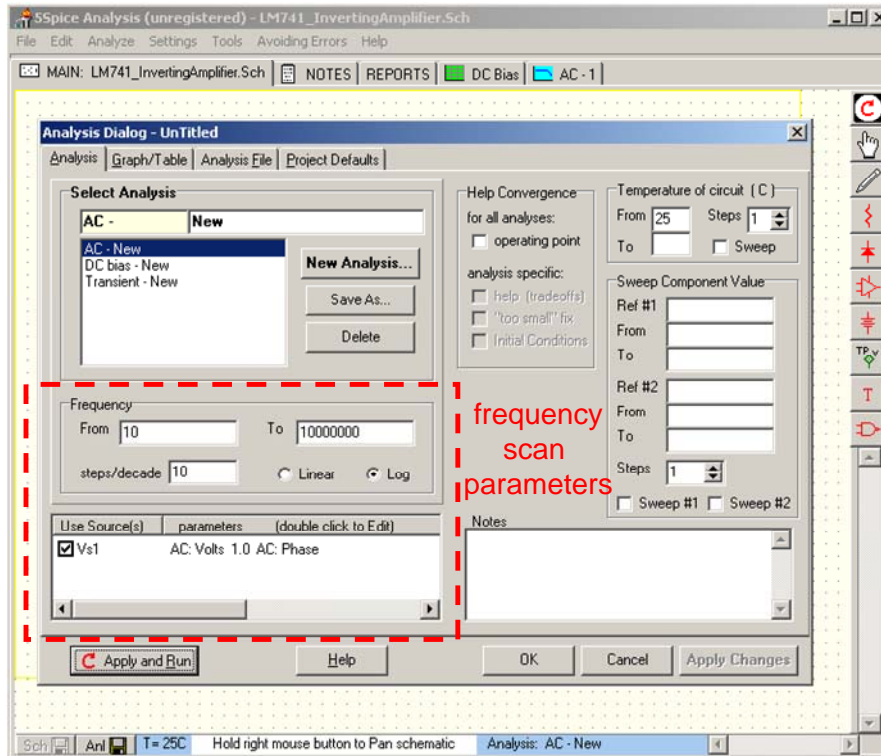
Figure 12.17: AC analysis setup.

AC analysis generally outputs a Bode plot of the frequency scan. The plot can be tailored to the circuit and the analysis in the "Graph/Table" tab of the "Analysis Dialog" as shown in figure 12.18 below. For our AC analysis we choose to plot both the magnitude and phase of the output AC signal.
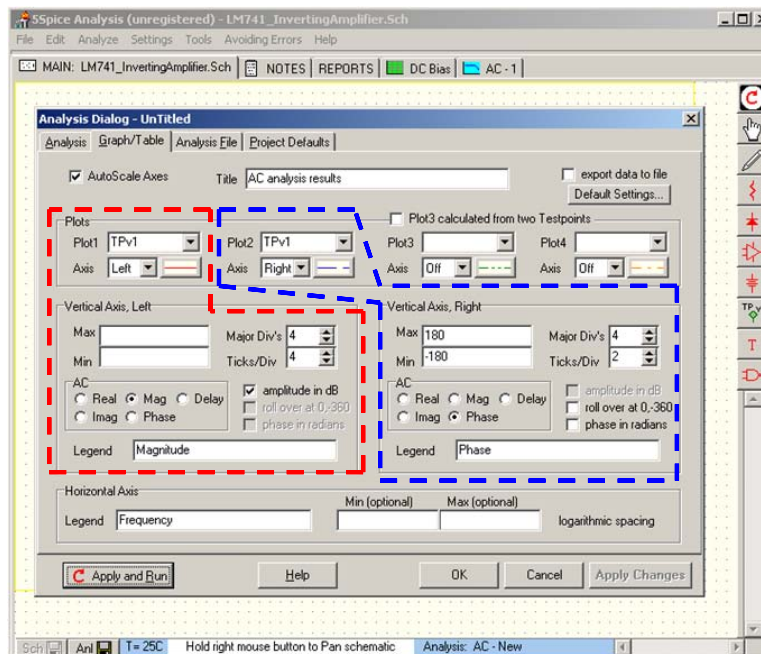
Figure 12.18: Output graphs setup for AC analysis.

When we run the AC analysis, 5Spice calculates the behavior of the output over the entire frequency scan, and produces the output Bode plots shown in figure 12.19 below. The graphs are displayed in the "AC-1" tab of the 5Spice main window.
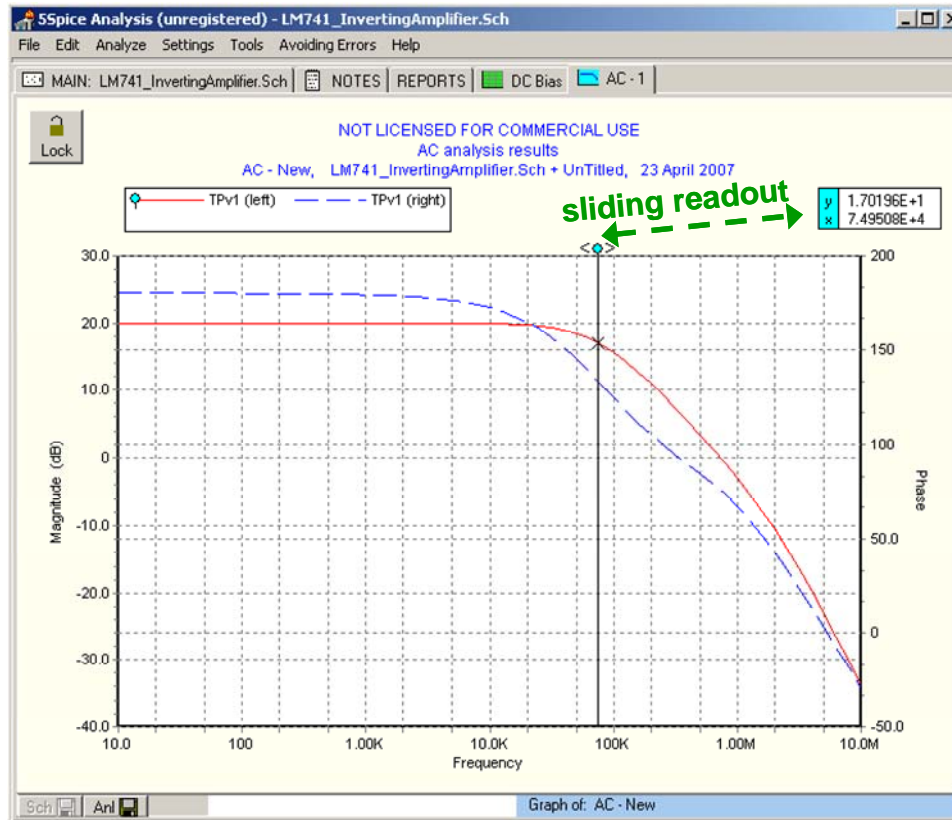
Figure 12.19: AC analysis output plots showing both the magnitude and the phase of the output signal. The graph also features a sliding readout, which has been indicated in green on the image.

If necessary the computed values used for the plots can be saved to a file – they are also displayed in the "Reports" tab of the main 5Spice window.

iii. Transient analysis

Transient analysis is done in the time domain. SPICE numerically computes the time evolution of the circuit voltages and currents as function of some time-varying input. For our transient analysis, we choose to use a 10 kHz square wave with 0.1 V DC offset and 0.2 V pk-pk, which we have selected in the "Analysis" tab of the "Analysis Dialog" window, as shown in figure 12.20 below. The transient analysis is set to compute the time evolution from t=0 s to t=0.0002 s.
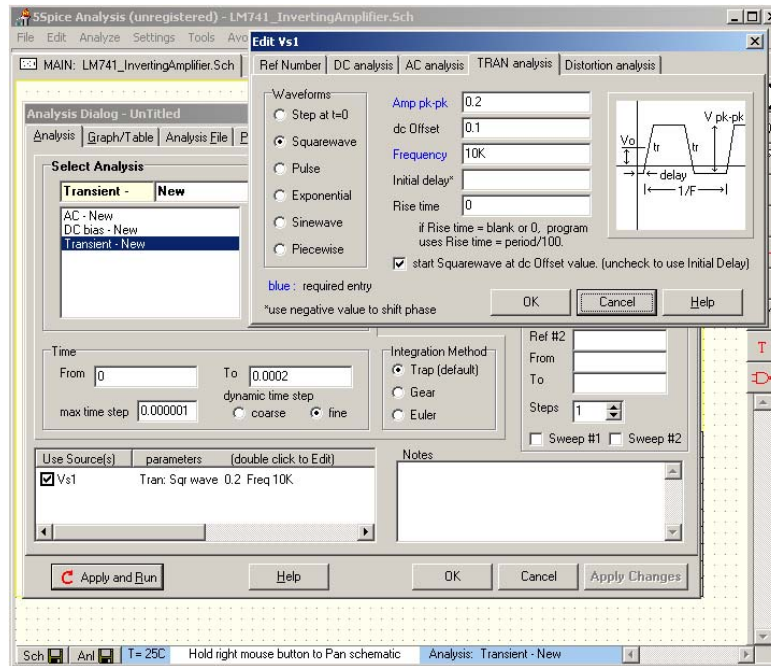
Figure 12.20: Transient analysis parameter selection for square wave input.

Transient analysis is the most difficult and frequently one must adjust the parameters governing the numerical computation in order for the simulation to work. Figure 12.21 (below) shows some of the basic changes to the numerical computation scheme one can select to help the transient analysis to compute.
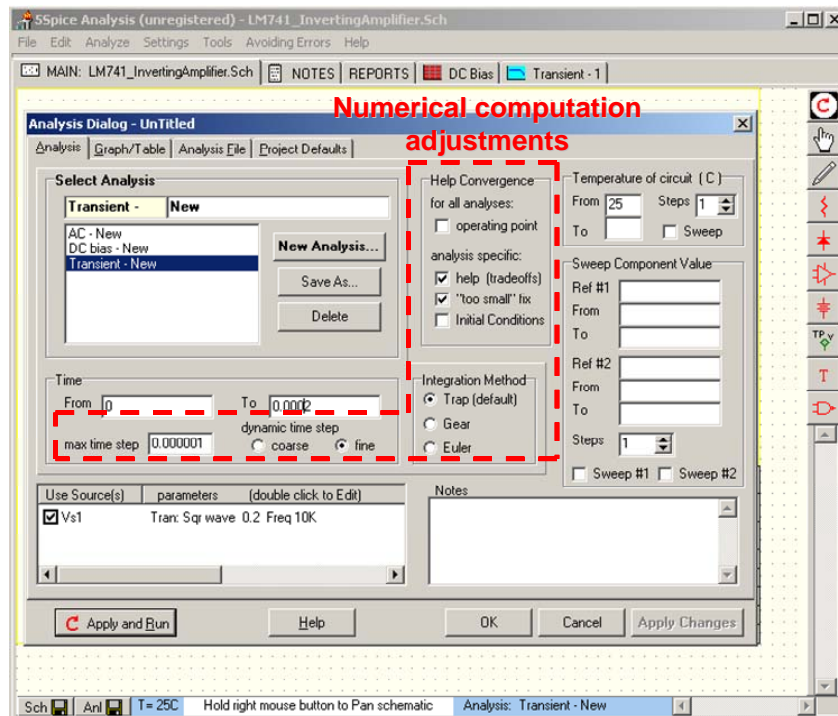

Figure 12.21: Coarse adjustments to the numerical computation used in transient analysis.

If the transient analysis cannot produce a solution after adjusting the above selections, then the parameters governing the tolerances and convergence criterion for the numerical computation must be modified. Figure 12.22 shows the "Project Defaults" tab and dialog in the "Analysis Dialog" window. The various parameters should be adjusted to obtain a transient numerical solution, though care must be taken since these adjustments will also affect the accuracy of the analysis results.
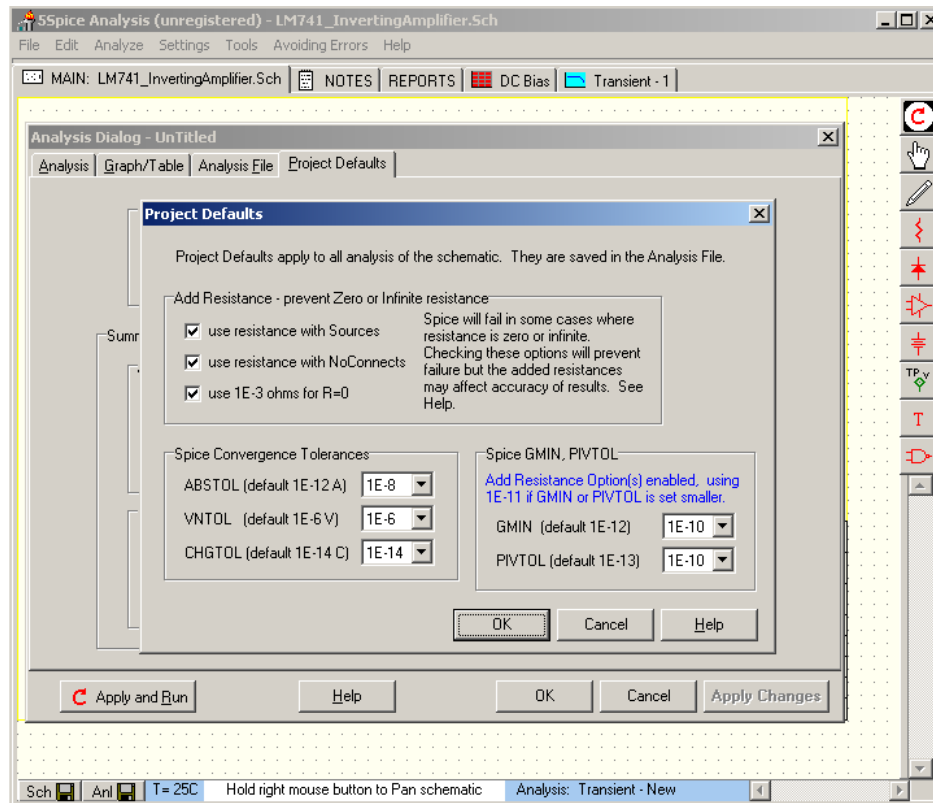

Figure 12.22: The numerical computation convergence tolerances and additional parameters.

The results of the transient analysis are plotted versus time. The graphs must be setup in a similar manner as shown in figures 12.18-19 for the AC analysis. The output is plotted in the "Transient-1" tab of the main 5Spice window. As figure 24 below shows, our LM741 inverting amplifier amplifies the input with *gain=-10*, though it does show some distortion due to some internal RC low-pass filtering. If we were to make the actual circuit in the lab, we should not expect better performance than what is shown in figure 12.23.
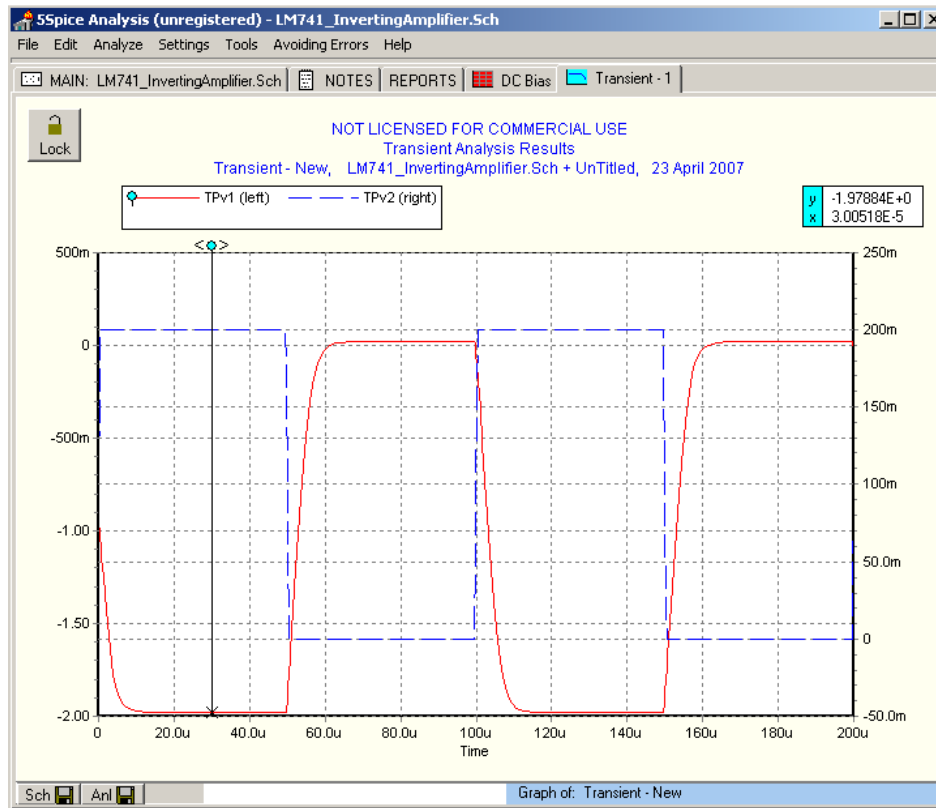
Figure 12.23: Plot of the transient analysis results. TPv2 is a voltage test point at the input.

In conclusion, we have shown how to construct a circuit diagram with 5Spice and do DC bias, AC, and transient analysis of a circuit. This introduction is not meant to be comprehensive and the reader is encouraged to read up further on how to use 5Spice from the "help" menu.

III. Eagle v4.16r2

Eagle v4.16r2 is circuit layout software. It is used for converting a circuit diagram into a circuit layout which can be sent to printed circuit board manufacturer for automated production.

As with 5Spice, you must first make a circuit diagram so that Eagle can help produce a circuit layout. This is a two step process, and we will go through it by using the example of the LM741 inverting amplifier of the previous section.

A. Circuit design

Circuit design in Eagle is similar to 5Spice. You open a blank schematic canvas and drag-and-drop components from a library onto it. You then attach the components with wires and add power. You must select the "use" menu item in the "Library" menu of an Eagle schematic window to open up a library and access its components. We will be

using the "QuantumOptics.lbr" library. Figure 12.24 below shows an Eagle schematic window with the important icons and menus highlighted.
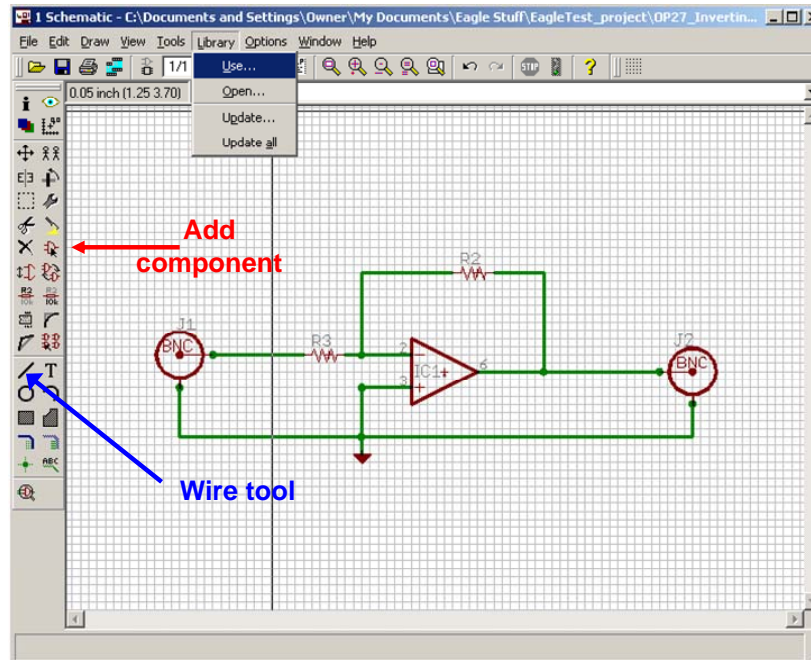

Figure 12.24: Eagle schematic window.

Once the schematic is complete, it can be converted to a board by selecting the board command, as shown in figure 12.25 below.
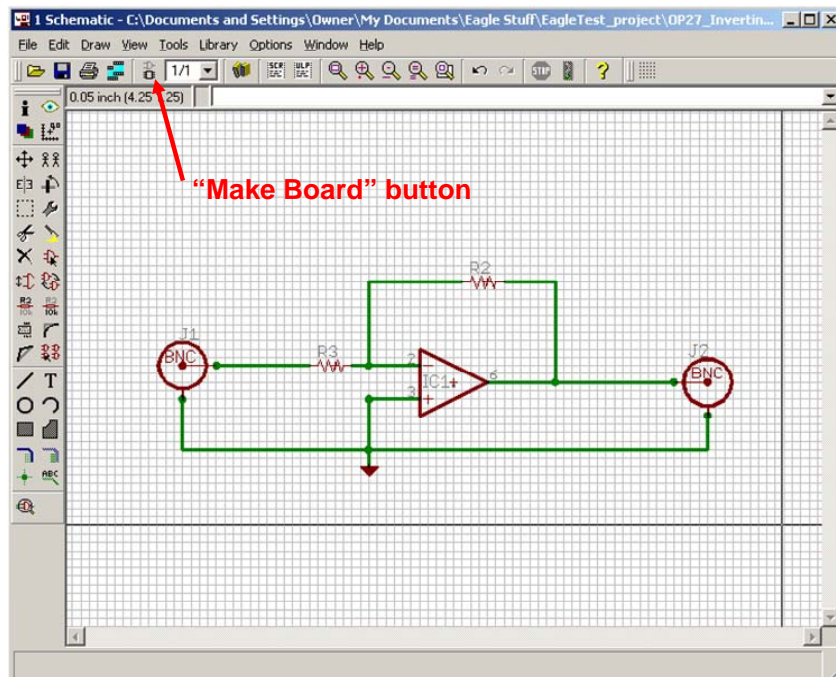

Figure 12.25: The "make board" button generates an initial board from the schematic.

B. Circuit layout

The printed circuit board produced with "make board" button still requires a lot of work. The first step is to place all the components appropriately into the board area as indicated in figure 12.26.
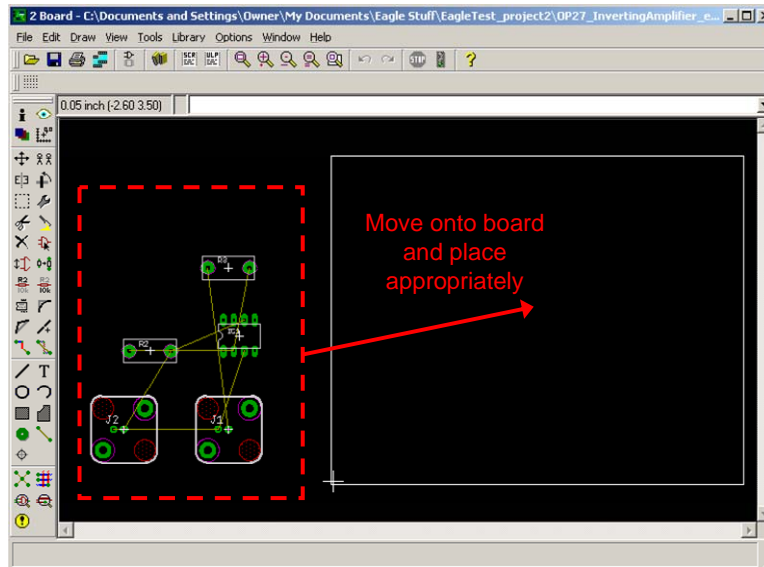

Figure 12.26: After using the "make board" button the circuit components should be placed in appropriate locations on the board.

An important fact to remember is that **the board has two sides** which you can use for placing components and routing wires/traces. The next step is to route all the wires. This can be done to a certain extent with the "autorouter" feature, though one must still route some of the wires by hand, generally. The wire routing is shown in figure 12.27 below.
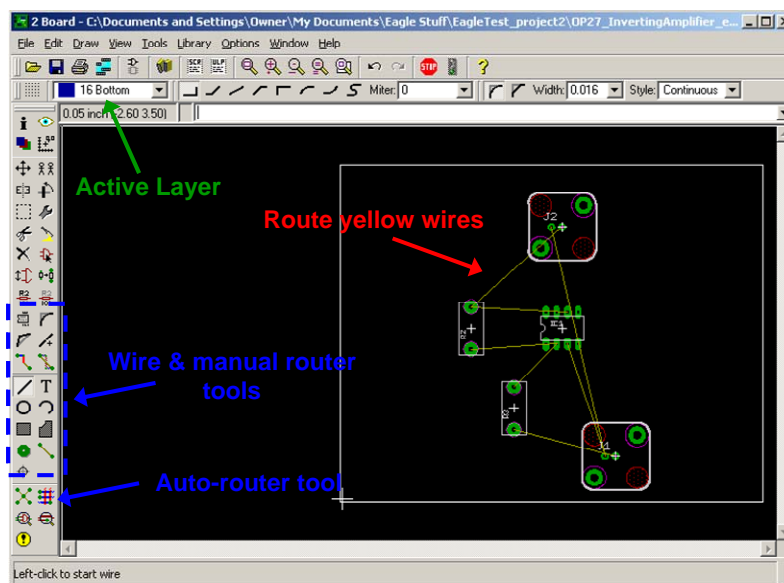

Figure 12.27: Available wire routing tools.

The wire routing is generally the bulk of the work and can take quite a long time. You should remember that the circuit is really 3-dimensionall and that one can use a holes and vias (conducting hole) to connect components and wires. After routing the wire, the board has the form shown in figure 12.28 below.
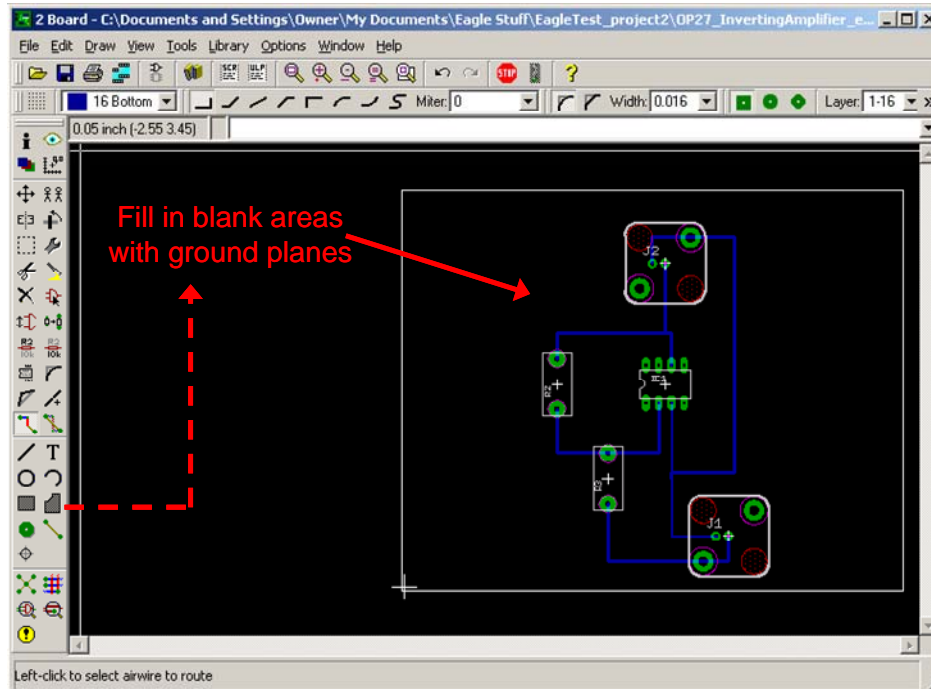

Figure 12.28: The rectangle tool is used to fill in blank areas with ground planes.

When making a board that will work at RF frequencies, a good rule of thumb is to have as much metal around which is at a ground voltage. These large area of metal are called "ground planes" and help shield parts of the circuit from one another. Futhermore, one should try to make sure that all the wires are also transmission lines (ideally with 50 Ω RF impedance), so that each wire has a ground on either side of it (and above or below it, depending on the orientation). Figure 12.29 shows the board after filling in all the blank areas with ground planes.
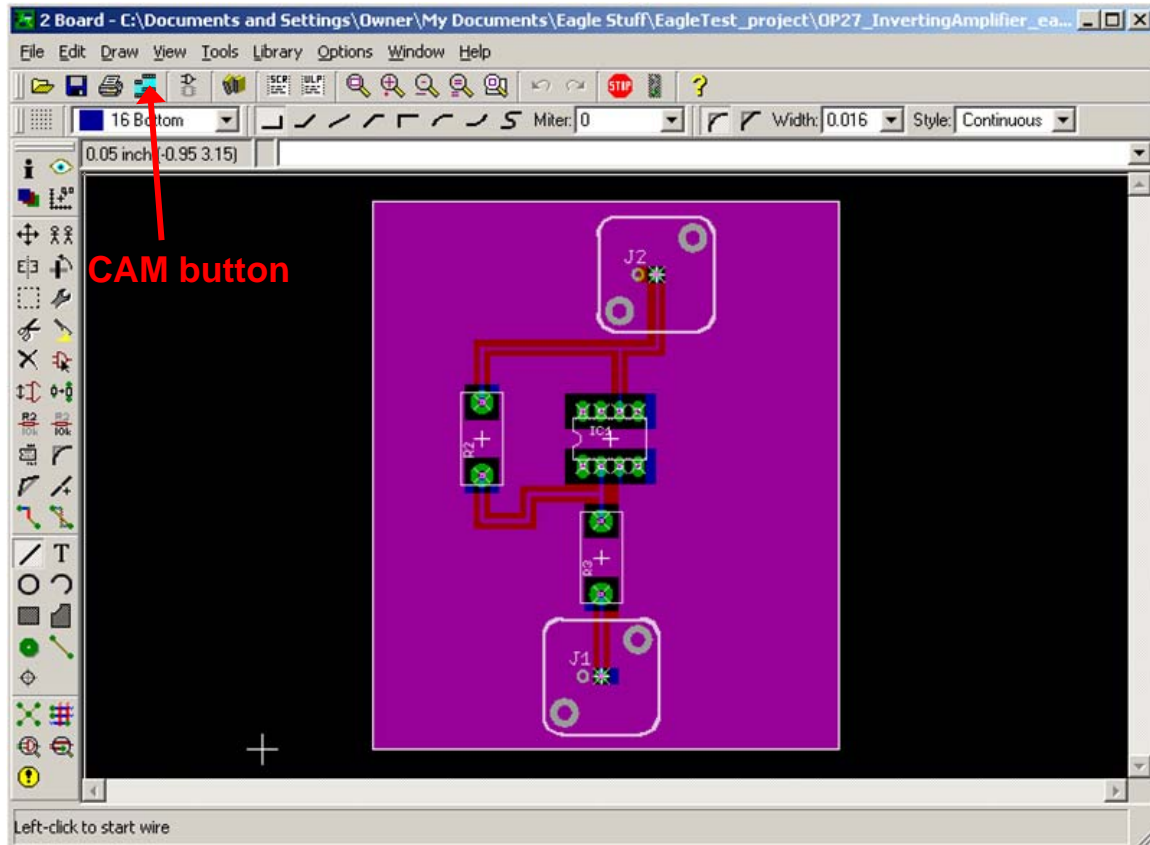
Figure 12.29: Finished board with ground planes and holes for components. All the ground wires have been replaced by ground planes. The CAM button, which is used for making Gerber files, is also indicated.

C. The Gerber file

The Gerber file is an industry standard computer file which contains all the information necessary for making a PCB (Printed Circuit Board). Once your board is finished, you can make a Gerber file by going to the CAM window (see figure 12.29 for CAM button). Once in the CAM window select the device to be "GERBERAUTO" and include a filename. The "process job" button will produce a Gerber file which you can send to a PCB manufacturer for automated production.

In conclusion, we have shown how to construct a circuit diagram and PCB with Eagle. This introduction is comprehensive and the reader is encouraged to find further information on how to use Eagle from the "help" menu.

Lab 12: 5Spice and Eagle

   This week's lab focuses on the use of the 5Spice v1.22 analog circuit simulator and the Eagle v4.16r2 circuit layout editor.

1. Use the 5Spice circuit simulator to simulate the ideal theoretical behavior of the circuit in Lab 10 part 4a – choose a feedback capacitor which optimizes the theoretical behavior of the circuit. You will replace the photodiode with the following sub-circuit of figure 12.30 below.
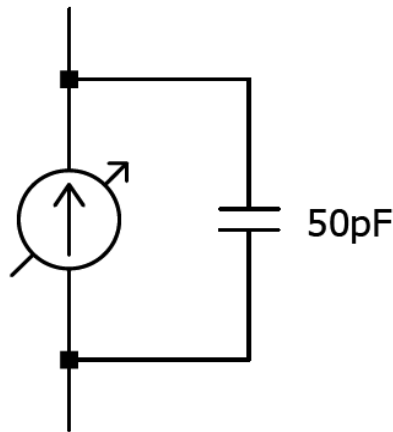

Figure 12.30: Current source model of a photo-diode.

1a. Determine the quiescent steady-state behavior of the circuit for photocurrent of 1 μA.

1b. Determine the response of the circuit to a photocurrent with the following functional form: $I(t)=10 + cos(\omega t)$ μA.

1c. Determine the response of the circuit to a square-wave photocurrent with a minimum current of 0 μA, a maximum current of 10 μA, and a frequency of 100 kHz.

2. Repeat part 1 (a, b, c) using the circuit of Lab 10 part 4b (i.e. use an OP27).

3. Design an RC low-pass filter with a 3 dB frequency of 100 kHz. Use the Eagle 4.16r2 layout editor to produce a schematic, board, and Gerber file. The input and output of the circuit should each use a female BNC connector.

Chapter 12: Electronic Circuit Simulation and Layout Software

Chapter 13: Comparators

So far, we have used op amps in their normal, linear mode, where they follow the op amp Golden Rules (no input current to either input, no voltage difference between the inputs). This week we look at comparators: specialty op-amps which use positive feedback or no feedback.

I. Comparators

Sometimes, you want to use an op-amp in its saturated mode – where the output is pushed either as far positive or as far negative as it can go. This use changes the op-amp from a linear device into a "bistable" device, that is, one that has only two possible output values.

The value of the output is based on a simple decision, made by the chip:

"If $V_+ > V_-$ set the output to the positive rail if not set the output to the negative rail."

Such a circuit is called a comparator. This is the first step into the world of digital circuits where all output values are expressed as a series of "yes" or "no" answers.

A. Positive Feedback

We often refer to the V_+ voltage as the *reference voltage* and connect the input to V_-. Comparators are supposed to determine:

- If the input (V_-) is less than the reference (V_+) then the output rails high.

- If the input (V_-) is greater than the reference (V_+) then the output rails low.

However, when a comparator looks at a real input signal it will always be faced with some small variations in the signal that make it hard to determine which input is greater. Even worse, this electrical "noise" could drive your comparator through many transitions when you really wanted your signal to show only one transition.

Figure 13.1 shows how a little noise added to a slowly decreasing signal can cause multiple transitions in the output. The top curve shows the comparator output, while the bottom curve shows the V_- input. The dashed line represents the V_+ input.
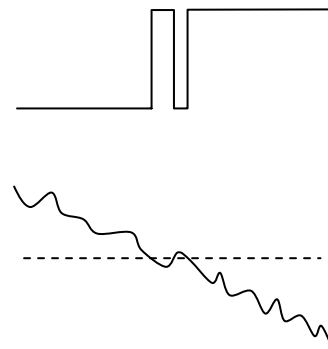
Figure 13.1: A noisy signal on V_- (bottom) generates multiple transitions (top) as it goes above and below the threshold on V_+ (dashed line).

When V_- drops below the threshold (dashed line) the output changes state. When noise makes the input voltage briefly go back above the dashed curve the output "bounces" to low and back to high accordingly.

The solution to this problem is to use positive feedback to simultaneously shift the reference voltage (V_+) as the output changes states. This is illustrated in Figure 13.2. If the shift is large enough, then the output will be less likely to bounce between the two output states during the transition. This shift, which is called a "hysteresis", depends on the design of the feedback network that ties V_+ to the output level.

An example of how this might work in an op-amp implementation is shown in Figure 13.3. We will start by ignoring the positive feedback, and note that the V_+ input is tied to 0.83 V via a voltage divider formed by a 10 kΩ resistor and a 50 kΩ resistor:

$$V_+ = 5\text{V}\frac{10\text{k}\Omega}{10\text{k}\Omega + 50\text{k}\Omega} = 0.83\text{V} \qquad (13.1)$$

Now let's consider the effects of the feedback resistor. When the output is high, this places the 100 kΩ resistor (ignore the 4.7 kΩ for now) in parallel with the 50 kΩ resistor so the reference voltage goes up to 1.16 V.

$$V_+ = 5\text{V}\frac{10\text{k}\Omega}{10\text{k}\Omega + 33\text{k}\Omega} = 1.16\text{V} \qquad (13.2)$$

Figure 13.2: When the output makes a transition (top), the reference level can be shifted (dashed) using positive feedback to prevent extra transitions as the input on V_- (bottom) makes multiple transitions across the original reference voltage on V_+ (dotted line).

Figure 13.3: Comparator with hysteresis.

When the output is low the 100 kΩ resistor is now in parallel with the 10 kΩ resistor so the reference voltage goes down to 0.76 V.

$$V_+ = 5\text{V}\frac{9\text{k}\Omega}{9\text{k}\Omega + 50\text{k}\Omega} = 0.76\text{V} \qquad (13.3)$$

Therefore, this circuit's output will go from high to low when V_- increases beyond 1.16 V. The circuit's output will go from low to high when V_- decreases below 0.76 V. When V_- is between 0.76 V and 1.16 V, the output will stay at its current value (either high or low). This memory of the past is why this type of feedback effect is called *hysteresis* and is often represented as a plot of output voltage as a function of input voltage. The hysteresis in this case given by
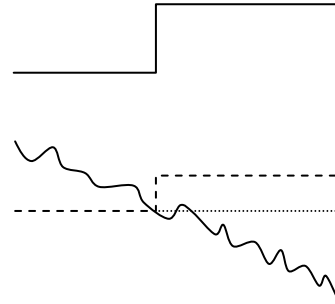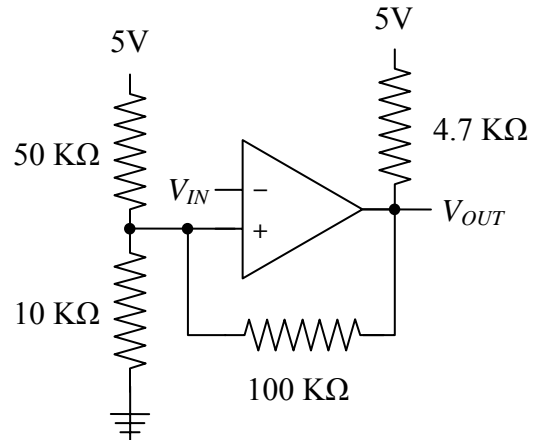
$$1.16V - 0.76V = 0.40V \qquad (13.4)$$

A plot of the reference voltage (V_{REF}) vs the input voltage (V_{IN}) for a comparator with a positive feedback network (figure 13.4). To make the two parts clear they have been slightly offset vertically. This type of plot is called a *hysteresis plot*. If there is an open box in the middle of the two paths then we say there is an input hysteresis. The hysteresis is defined as the difference between the two values of V_{IN} where there are transitions.
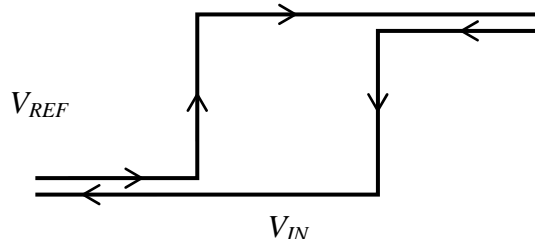

Figure 13.4: A plot of V_{REF} vs V_{IN} for a comparator with a positive feedback network.

This positive feedback circuit is called a *Schmitt trigger*. A slightly better version can be made using two transistors but that's beyond the scope of this chapter.

B. Comparators

Commercial comparators are specialty op amps and come in a standard IC package. They are designed with the sole purpose of switching very quickly and do not have a number of the other good op amp qualities we have been exploiting in negative feedback circuits. The LM2903 and LM311 are the standard comparators that we will use in this week's lab.

Open-Collector Output

Since comparators convert a linear signal into a digital signal, they are routinely used to connect two dissimilar electrical systems with different supply or reference voltages. Consequently, most standard comparators come designed to connect to any system you want to use. The output of a comparator is actually the collector input of a transistor switch (with "open" or "closed" states), rather than actual output voltages – note: in a regular op-amp the output would be some form of push-pull stage.

Figure 13.5: Equivalent circuit for an open collector output and a pull-up resistor.

When you use a comparator in a circuit, you usually connect the output to a voltage source via a *pull-up resistor*, as shown in figure 13.5. When the comparator is in the high state, the switch will be open, no current will flow through the resister. There will not be a voltage drop across the resistor, and hence the output will be pulled to the positive supply. When the comparator is in the low state, the switch will be closed, current will flow through the resistor. There will be a voltage drop across the resistor, and the output will be pulled to the lower supply level (ground in this case). The 4.7 kΩ resistor in Figure 13.3 is a pull-up resistor.
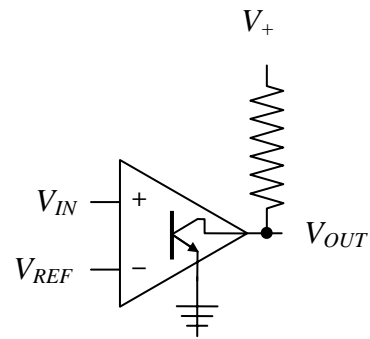
Even though the real open collector states are "open" and "closed," they are still often called "high" and "low" in the assumption that you are using pull-up resistors.

Comparator Limitations

Comparators typically have high gain at high frequencies and no phase compensation. Since comparators are designed to go all the way from the negative rail to the positive rail during transition, their data sheets list a *transition delay time*, which is the time to go from one rail to the other, instead of a slew rate. For the LM2903 the transition time is roughly 1.5 μs.

The most serious limitation for some comparators is that they will not allow a large voltage difference between V_+ and V_-. This difference is called the *differential input voltage* and should be looked up on the data sheet before you design a circuit using a specific comparator.

Finally, another issue is that the input circuit's impedance changes as the output state changes.
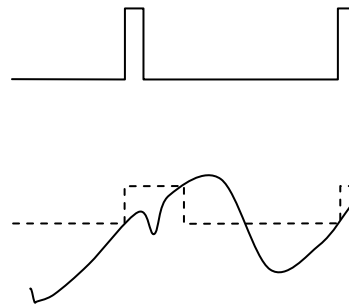
Edge Triggers

A closely related class of circuits is *triggers*. A rising edge trigger will make a *pulse* (fast transition from low to high and back again) each time the input voltage rises past a particular threshold voltage. On the other hand, when the input voltage goes back below the threshold the output stays low. When the output is triggered, the threshold voltage is raised for a certain period of time to prevent multiple triggers. After that time has elapsed, the threshold is lowered back to its normal value. This is called *rearming* the trigger (Figure 13.6).

Figure 13.6: The output of the rising edge trigger sends out a pulse (top) when the input (bottom) rises past the threshold (dashed line). The output stays low when the output returns to be low the line. A hysteresis ensures that only one pulse is made per transition even in presence of noise.

II. Oscillators

In many electronics applications it is useful to have a small function generator to synchronize various portions of a circuit or to insert time delays between different actions. This is called a *clock signal*.

You should be used to this concept from a trip to a store shopping for PCs. The speed that it will perform individual operations (single core) is proportional to the clock speed. Therefore we know that 3.0 GHz PC will be 10% faster than a 2.7 GHz PC.

One simple way to make a clock signal is using positive feedback and a comparator to make a square wave generator. An example square wave generator is shown in figure 13.7. The circuit is based on a comparator with hysteresis. The comparator is assumed to be powered by 0 V and +5 V. The divider in this example defines the transition levels to be $V_+/3$ and $2V_+/3$.

The wrinkle here is that the output is connected to the reference voltage through an RC circuit. After the output switches state, the capacitor gets charged by the voltage at the output. When the charge on the capacitor gets higher than $2V_+/3$, the hysteresis causes the output to change states and the capacitor gets discharged. When the charge on the capacitor gets lower than $V_+/3$ then the output changes states again. This process repeats forever and the resulting output is a square wave as shown in figure 13.8. The square wave will have the period given by roughly $2ln(2)RC$.
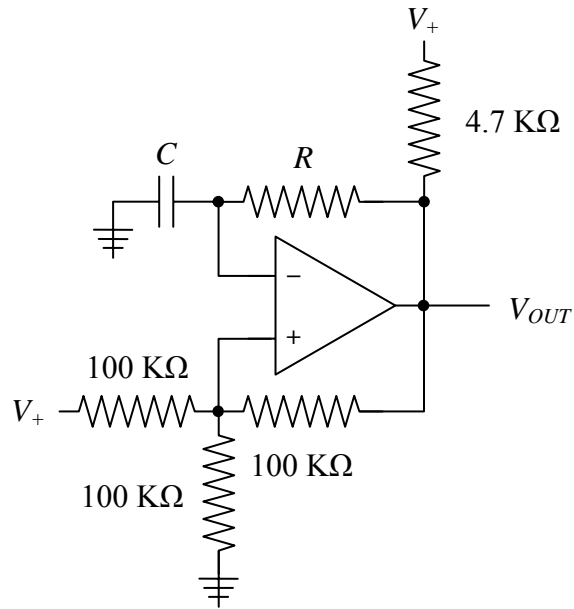
Figure 13.7: A square wave generator made using a relaxation oscillator.
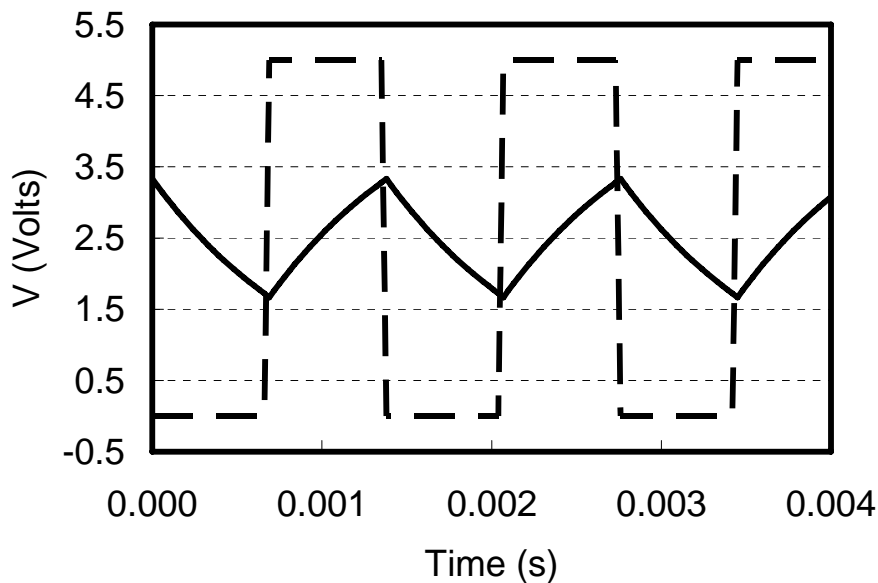
Figure 13.8: The output of the relaxation oscillator shown in Figure 7 (dashed) and the voltage across the capacitor are plotted as a function of time.

These kinds of circuits are called *relaxation oscillators* or *RC oscillator*. They generally work well for frequencies from around 1 kHz to 1 MHz. When designing an oscillator, make sure that *R* is significantly larger than the pull-up resistor or the output will be asymmetric. (Can you to figure out why?)

In precision applications the simple RC oscillator would be replaced by a vibrating crystal. Vibrating crystals are available as precision oscillators with a wide range of frequencies. Suggested designs for using crystals with comparators are available in the data sheets.

Design Exercises

Design Exercise 13-1: Design a comparator that will switch when the input signal crosses +2.5 V with no hysteresis (as in equation 13.1).

Design Exercise 13-2: Modify the design of your comparator with a positive feedback network to add a total hysteresis of approximately 0.2 V.

Design Exercise 13-3: Design a relaxation oscillator with 10 Hz frequency. Use resistor and capacitor values in the same range as those found in the electronics lab.

Bonus: If you run design exercise 13-3 in 5Spice, then you will notice that the square wave is not symmetric (i.e. "high" time is longer than the "low" time). Modify the circuit so that you can make the square wave symmetric.

Lab 13: Comparators and Oscillators

1. Connect a LM2903 comparator to measure where a sinewave crosses +2.5 V, with no feedback. Use +5 V and ground for your supply voltages. Remember that the output must use a "pull-up" resistor. Use a 4.7 KΩ pull up resistor tied to +5 V. Observe and characterize its output when driven by a 1 kHz sine wave with a 1 V amplitude and a DC offset larger than +0.5 V.

   Add positive feedback to your comparator to generate a total hysteresis of 0.2 V and observe its effect.

2. Build a square wave oscillator with 10 Hz frequency that drives a green or red LED. Examine the voltage across the capacitor and the output voltage on your scope at the same time to understand the switching structure.

   Construction mini-project: You should build the circuit directly onto a prototyping board with all components connected via solder. The webpage has an image of the prototyping board layout. The LED flasher will be powered by a +9 V battery in combination with a +5 V voltage regulator (MC7805, see datasheet for more details). You should use the regulated +5 V output to power your circuit. The regulator guarantees a voltage of +5 V even if the input voltage is not +9 V volts, while also providing additional protection for the rest of the circuit. Also, you should use an IC socket for your LM2903 comparator IC. You may use lab components to construct your circuit.

   Bonus points if your prototyping circuit works! … and you can keep it.

Chapter 13: Comparators