Experiment 1

Introduction to analog circuits and operational amplifiers

Electronic circuit design falls generally into two broad categories: *analog* and *digital* (a third category, *interface* circuitry, includes hardware to join these two major circuit realms). Digital circuitry, as you probably already know, uses electronic components and systems to represent and store numerical data and to perform algebraic and logical operations on the data. Incredibly complicated digital structures are created by combining a few simple circuit building blocks (such as *registers*, *gates*, and *clocks*) into vast networks of components.

Analog circuitry, in contrast, is used to respond to continuously-variable electrical signals from sensors (such as microphones, thermistors, antennas, and accelerometers) or to provide continuously-variable control signals to actuators (such as loudspeakers, heaters, antennas, or motors). Analog circuitry is used, for example, to connect digital computers and circuits to many of the physical devices they use for data I/O and storage. Our focus in this course will be mainly on analog designs.

Probably the most important application of analog electronic circuitry (especially in the sciences) is to *amplify* and *filter* the power of a minute signal so that it can be accurately measured or used to control something interesting. In this experiment you'll jump right in and start designing and building simple, useful amplifier circuits using that truly marvelous, tiny building-block of modern analog electronics: the *operational amplifier*.

Before you can begin to understand how to construct such amplifiers, you must understand some pretty basic concepts concerning what sorts of elements make up electronic circuits and how they work together in a design. The first several pages that follow cover these basic ideas; the information may be dry and dense, but it is important that you read it! Hopefully much of the following section is a review of what you already know, but, if not, familiarize yourself with the content so you can quickly refer back to it when the time comes.

Next the text covers the behavior of an *ideal operational amplifier* (op-amp) and why this behavior makes it so versatile and easy to use by incorporating *negative feedback*. Finally we cover some additional concepts which will prove useful as you analyze and design circuits. The *Prelab Exercises* will test your understanding of this material and give you some practice in preparation for your lab session. Complete them and come to your recitation session ready to ask (and answer!) questions about the material.

One auxiliary, but very important, objective of this experiment is to give you some initial experience using the lab electronic equipment: signal generator, oscilloscope, cabling, and, of course, the analog trainer/breadboard you use to construct your circuits.

Contents

Table of Circuits

CIRCUIT BASICS

Current, voltage, power

Current is the flow of electric charge from place to place. Electronic circuitry employs networks of narrow, highly conductive elements (copper wiring or traces on printed circuit boards) to effectively confine the flow of charge to well-defined paths. Current is defined as the measure of the rate of charge flow through a surface (typically the cross section of a wire) and is measured in the SI unit *ampere* (amp, or A). The SI unit of charge is the *coulomb*, which is defined such that $1\,\mathrm{A} = 1\,\mathrm{coulomb/second}$. An amp is a very large current for small, table-top circuit designs; our electronic circuits will have currents of about 10^{-7} amp to 10^{-2} amp, so we'll most often be dealing with currents of *microamps* (uA or μA) to *milliamps* (mA).

Currents are generated by the motions of charge carriers in the circuit in response to electromotive forces induced by electromagnetic fields. As a charge carrier moves about, its potential energy due to the fields varies. The work done on the charge by the fields is equal to the reduction in potential energy of the charge as it changes position. The potential energy per unit charge due to an electric field is called the *electrostatic potential* (or just *potential*) and is measured in the SI unit *volt* ($= 1\,\mathrm{joule/coulomb}$). We have various ways of establishing either steady or time-varying potentials in our circuits: *power supplies*, *signal generators*, and *batteries*. These devices also serve as sources and receivers of charge carriers, so that the circuit to which they are connected remains electrically neutral (no net charge). In our circuits maximum voltages are no more than ~12V, and signals have amplitudes of a fraction of a *millivolt* (mV) to a few volts.

Assume a circuit has charge carriers flowing steadily from a point *A* to a point *B* at potentials v_A and v_B. If the current flowing is i_{AB}, then the power being expended by the source of the potential difference must be $P = (v_A - v_B)\,i_{AB}$. If the potentials and the currents are time-varying, but the instantaneous current out of *A* remains equal to that arriving at *B*, then the current between the points remains a well-defined function of time, and we have the instantaneous power: $P(t) = (v_A(t) - v_B(t))\,i_{AB}(t)$. Our circuits will have power flows on the order of a few tenths to about a hundred *milliwatts* (mW).

A component which can continually add power to a circuit is called an *active element*. Other components (most of which dissipate power or otherwise remove it from the circuit) are called *passive elements*.[1]

[1] Electrical engineers often refer to semiconductor devices such as transistors and integrated circuits as *active elements*, because these devices can transfer power from a steady power source into a signal circuit.

Frequency, wavelength, lumped circuit elements

Electromagnetic fields propagate at the speed of light, 30 cm/nanosecond (or about a foot/nanosecond). One nanosecond is the period of a signal oscillating at a gigahertz (GHz, 10^9 hertz). The maximum frequencies we'll be using in our circuits are no more than a few megahertz (MHz) or kilohertz (kHz), so the wavelengths of these fields will usually exceed hundreds of meters, hundreds to thousands of times bigger than the physical sizes of our circuits. Consequently, but maybe not so obviously, each individual element in a circuit will have, to a high degree of accuracy, no change in its total net charge as the fields oscillate. Thus we may safely assume that there is *zero net total current flow into or out of all of an element's connections* to the circuit at any instant. Such a component is called a *lumped element*. Examples of lumped elements are the resistors, capacitors, LEDs, and integrated circuits (ICs) we'll be using.

In the high-frequency case, where wavelengths become comparable to the size of a component, the fields and currents may vary across it, making it a *distributed element*, and our assumption above is no longer valid. Examples of distributed elements include antennas, microwave waveguides, the motherboard in your computer or tablet, and the national electrical power grid.

Most of the elements we use in our circuits — resistors, capacitors, inductors, diodes, batteries, etc. — have two terminals for connections to circuit conductors, making them *lumped, two-terminal elements* (note that any lumped element will have *at least* two terminals, since the total current flow into the element's connections must vanish, as described above). Pictured in Figure 1-1 is a selection of symbols for typical two-terminal elements as used in an electrical circuit drawing, called a *circuit schematic*. Because the currents at the two terminals must be equal and opposite (flowing in at one terminal and out at the other, so the total net current into the element is zero) we can simply refer to *the current flowing through the element and the potential difference (voltage) across it*.

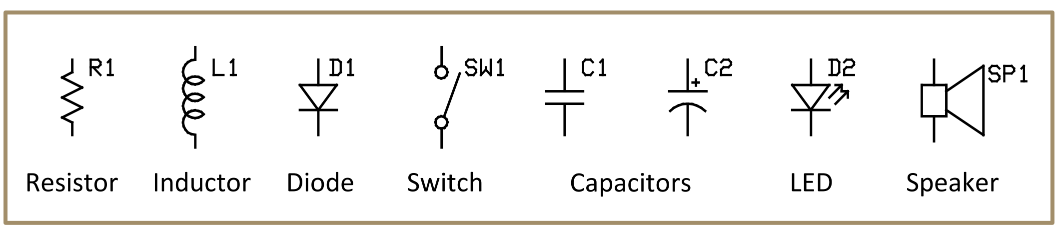


Figure 1-1: Schematic symbols for a selection of two-terminal, passive circuit elements. Also shown with each symbol is an example of an associated *reference designator* to uniquely identify that component in a circuit.

Sources and signals

In most cases the independent variables in the set of equations we will write out to describe the behavior of a circuit are a few voltages or currents we control as *inputs* to the circuit; the equations then allow us to determine the circuit's *outputs* in response to the inputs. The independent inputs are termed *sources* or *signals*, and may be generated by batteries, power supplies, signal generators, microphones, antennas, thermocouples, or whatever else we can think of which produces a potential (*voltage source*) or injects a current (*current source*) into our circuit. Sources are usually described as two-terminal devices in our circuits and are *active elements* since they inject power into the circuit. If a source produces a constant output (such as a battery or power supply), then it is called a *DC* source (for "direct current"). If its output is sinusoidal, then it is an *AC* source or signal (for "alternating current").

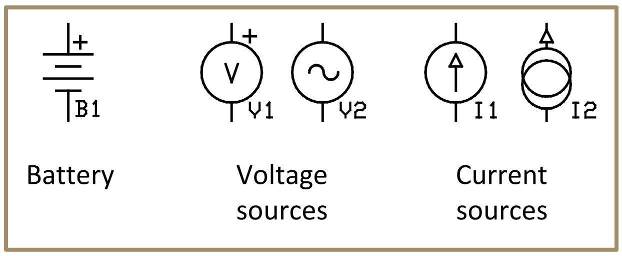


Figure 1-2: Schematic symbols for common voltage and current sources; each is a two-terminal, active element. Polarity or current flow has the direction indicated when the source output is positive. The battery symbol will often be used to represent any constant-voltage (DC) source such as a power supply. The other sources may be constant or time-varying (an "AC source" usually means that its output varies sinusoidally – V2 in the figure represents an AC voltage source).

The schematic symbols for some common active sources are shown in Figure 1-2. These sources are considered to be *ideal*, in the sense that each can maintain its specified output voltage or current regardless of what they may be connected to in the circuit and how much power they must supply. Of course, real sources are not quite so capable! The polarity or current direction included with the schematic symbol shows the relative potential or current flow when the source output has a positive value.

Grounds and power supply terminals

The fields in our circuits produce potential differences and corresponding current flows. Because only potential differences are significant, we will pick one convenient point in a circuit and *define it as having zero potential* (0 Volts); all other voltages in the circuit will be measured or specified with respect to this point. The symbol used in this text for the 0-Volt reference point is a triangle: ⏚. We refer to this point as the *circuit ground*, and a terminal connected to this point is said to be *at ground potential* or to be *grounded*.

We will use a couple of conventions when drawing our circuit schematics which will considerably reduce the clutter in them. The amplifiers you will build will require DC power from a power supply or batteries in order to operate properly, so these constant-voltage power sources must be indicated in our circuit drawings. Additionally, often several components in the circuit (including the power supply) will have terminals connected together and to the 0-Volt reference point (ground). A power supply schematic drawing simplification is shown in Figure 1-3; instead of explicitly showing the power supply source symbols, we'll put little arrow symbols with their associated voltages at whatever points need to be connected to the power supply (as shown on the right in Figure 1-3). Wherever these symbols appear, you must remember that physical wiring connects all symbols with the same voltage to the appropriate terminal of the power supply source. Similarly:

> Multiple ground symbols appearing in a schematic *are all implicitly connected together* and are connected to the appropriate power supply return terminal, which also serves as the 0-Volt reference point for the circuit (as shown in Figure 1-3).

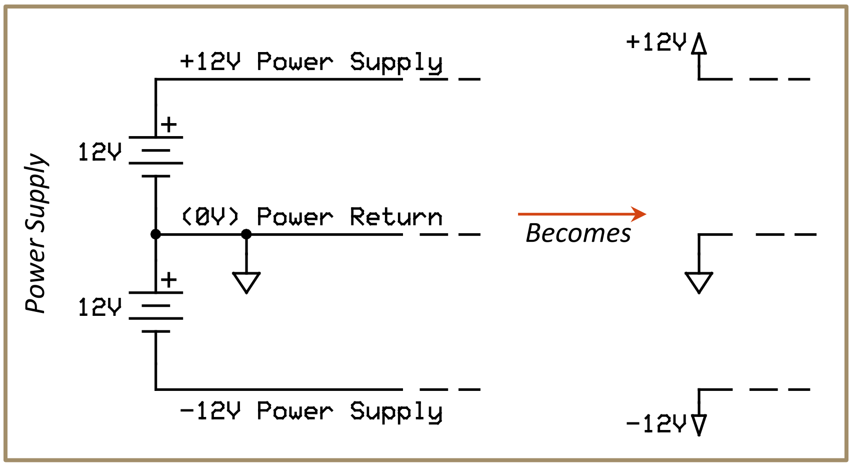


Figure 1-3: Simplified schematic diagram depiction of a power supply. Multiple arrow symbols labeled with the same voltage (such as +12V) may appear in a diagram. All must be considered to be connected together and to the appropriate power supply source terminal. Similar considerations apply to the appearance of multiple ground symbols in a diagram. One power supply terminal is connected to ground as well, as shown above. The voltage labels always give the voltage values with respect to ground, which is the 0-Volt reference point.

Resistors

The two most common two-terminal, passive elements we will use are the *resistor* and the *capacitor* (another common element, the *inductor*, is mostly used in radio-frequency circuits; we'll discuss the capacitor and inductor in later experiments). These elements are important to study first, because each of them has a simple, linear relationship between the voltage across and current through it (at least to a very good approximation, for low frequencies and voltages). You will generally find that your designs will contain more resistors and capacitors than the combined total of all of the rest of the components in the circuits.

Let's start with the resistor and its famous *Ohm's law*. Figure 1-5 on page 1-5 shows a variety of resistors with different physical sizes, shapes, uses, and power ratings; the ones you will use when "bread-boarding" prototype circuitry will look like the small, light-brown

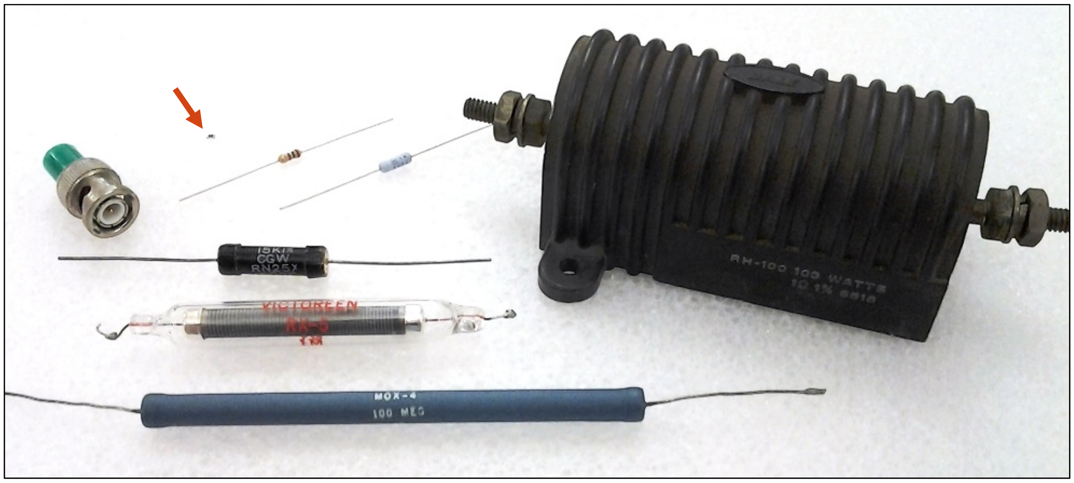
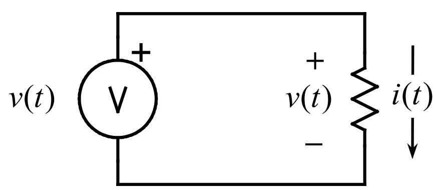


Figure 1-5: A variety of resistors, from the tiny (.08 inch × .05 inch) surface-mount component barely visible near the upper left corner of the photo (see arrow) to the beefy, 100 Watt power resistor to the upper right. The long resistor in front is designed to be used with very high voltages.

resistor towards the upper left in the photo. If we connect a resistor to a voltage source (a power supply or signal generator) as shown in Figure 1-4, then, obviously, there will be a voltage across the resistor and a current flowing through it. Figure 1-4 shows the usual conventions for the polarity of the voltage and direction of the current flow when the potential $v(t) > 0$.



$$v(t) = R \times i(t)$$

Figure 1-4: A simple circuit illustrating Ohm's Law for a resistor with value *R*.

Since the lines connecting the elements' terminals in a schematic diagram are considered to be perfectly-conducting wires, terminals connected by a wire in the diagram must have identical voltages, and current will flow through the connection without any change. Thus, if the output voltage of the source in Figure 1-4 is $v(t)$, then that is also the voltage across the resistor R. Ohm's law states that the voltage $v(t)$ across a resistor and the current $i(t)$ through it are strictly proportional at any instant:

Ohm's law for a resistor

1.1
$$\boxed{v(t) = R\, i(t)}$$

The constant of proportionality, R, is called the *resistance* of the resistor and has SI units of *Ohms*: $1\,\text{ohm} = 1\,\text{volt/amp}$. For an ideal resistor R is a real, constant number (independent of voltage, current, or frequency) with $R \geq 0$. Equation 1.1 is the *defining relation* for an ideal resistor; the actual resistors you will use behave in a very nearly ideal manner. The circuits we build will have voltages mostly in the range of a few tenths to a few volts, and currents of a few microamps to a few milliamps. Thus the useful range of resistor values is a few hundred to a few million ohms (MΩ). Commonly used values should be 10^3 to 10^5 ohms (1 to 100's of kΩ).

> The elegance of the lowly resistor lies in Ohm's Law, equation 1.1: a resistor is a current to voltage converter (and vice versa)! If we have a wire with, say, a few milliamps of current flowing in it, then inserting a 1.0 kΩ resistor (kΩ: kilo-ohm =10^3 ohms) in the circuit *will produce a voltage across it of 1.0 V/mA, faithfully following any variation in the current flow*. Conversely, if we put a known voltage across a resistor, then we can immediately calculate the resulting current flow through it. These facts will be of enormous importance when you design and analyze various amplifiers during this course!

Series and parallel resistors

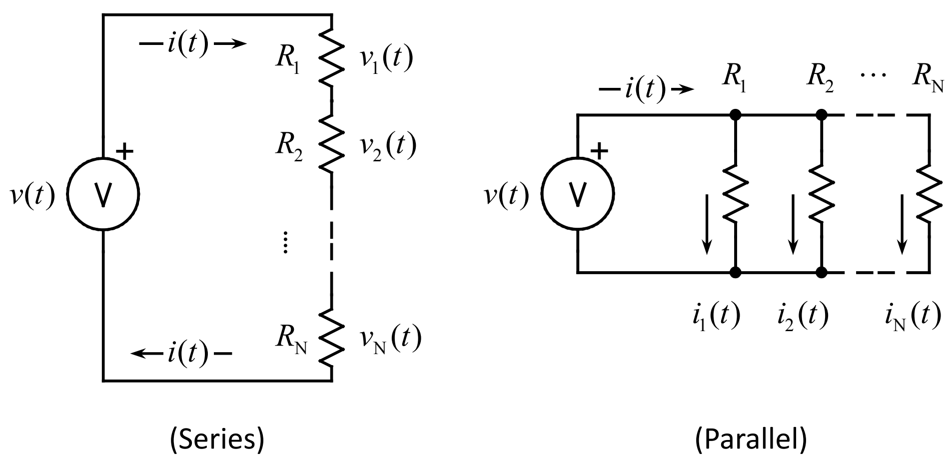


Figure 1-6: Series and parallel resistor combinations. For the series case, the same current flows through all resistors, and the total voltage is the sum of the individual resistor voltages. For the parallel case, the voltage is the same across all resistors, whereas the total current from the source is the sum of the individual resistor currents. The results are that series resistances add; parallel conductances (1/resistance) add.

A common situation when analyzing part of a circuit is to find that two or more resistors (or other components) are connected in *series* or in *parallel*, as illustrated in Figure 1-6. The problem is to determine the *equivalent resistance* of such a combination, that is, the relationship between the voltage and current across the entire array of components. The solution is straightforward once you recognize that:

- *Series combination*: the same current must flow through all resistors, and the total voltage across them must be the sum of the individual resistor voltages.
- *Parallel combination*: the same voltage appears across all resistors, and the total current must be the sum of the individual resistor currents.

Consider the series combination first. Referring to Figure 1-6, the voltage across the k^{th} resistor is $v_k(t) = R_k\, i(t)$, so the total voltage is $v(t) = i(t)\sum R_k = R_{\text{series}}\, i(t)$, where R_{series} is the equivalent series resistance we seek. In the parallel case, $v(t) = v_k(t) = R_k\, i_k(t)$, so $i_k(t) = v(t) / R_k$. The total current is therefore $i(t) = v(t)\sum(1/R_k) = v(t) / R_{\text{parallel}}$, where R_{parallel} is the equivalent parallel resistance. The equivalent resistance (resistance seen by the source) for each case is then:

1.2

$$\boxed{\begin{aligned} &\text{Series combination:} && R_{\text{series}} = \sum_{k=1}^{N} R_k \\ &\text{Parallel combination:} && \frac{1}{R_{\text{parallel}}} = \sum_{k=1}^{N} \frac{1}{R_k} \end{aligned}}$$

The reciprocal of resistance is called conductance, and has the SI unit *Siemens* ($= \text{ohm}^{-1}$).

Series resistances add. Parallel conductances add.

Using the second of equations 1.2 for the case of only two resistors, we can derive the more familiar equation for two parallel resistors,

$$R_{\text{parallel}} = \frac{R_1 R_2}{R_1 + R_2}$$

which is less elegant than the more general expression in 1.2.

Now let's ask what the voltage drop across any particular resistor would be for our set of series resistors in Figure 1-6. The voltage across the k^{th} resistor is $v_k(t) = R_k\, i(t)$, and $i(t) = v(t)/R_{\text{series}}$, so we get expression 1.3, which may seem pretty obvious:

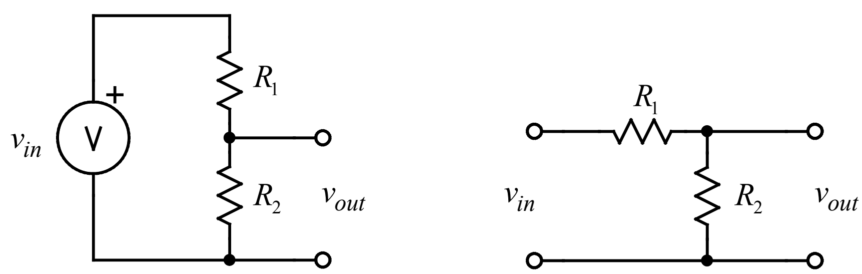
1.3

$$\frac{v_k(t)}{v(t)} = \frac{R_k}{R_{\text{series}}} = \frac{R_k}{\sum R}$$

Thus the ratio of the voltage across one resistor to the total voltage is just the ratio of the resistor's value to the total resistance in the series. This result segues nicely into our next topic.

Voltage dividers and the potentiometer

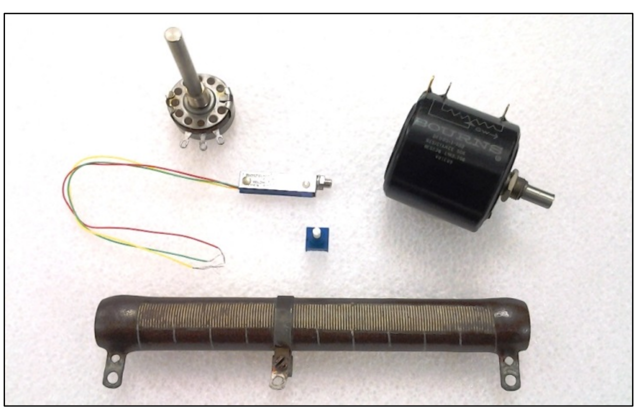
The result 1.3 for a series-connected pair of resistors provides the solution for the behavior of the very common *voltage divider* circuit, Figure 1-7.



1.4
$$\frac{v_{out}}{v_{in}} = \frac{R_2}{R_1 + R_2}$$

Figure 1-7: The *voltage divider* is just a series combination of two resistors. The *input* voltage, v_{in}, is the source voltage applied across both resistors; the *output* voltage, v_{out}, is the voltage across the bottom resistor of the pair. The gain of the divider is defined as $G = v_{out}/v_{in}$, and is trivial to derive from equation 1.3. The circuit on the right is the same voltage divider, but assumes that a source of voltage is applied between the two input terminals rather than explicitly showing the source.

> The voltage divider configuration of two resistors (or other types of elements) will show up again and again in the circuits we will design and build. It would be wise for you to commit the schematics in Figure 1-7 and the formula 1.4 to memory!



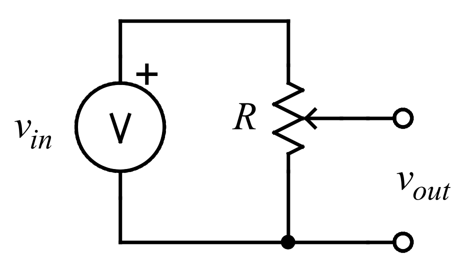


Figure 1-8 (left): A selection of potentiometers. All except the one at the bottom of the photo are adjusted by turning a shaft. The bottom device clearly shows how a potentiometer is constructed, in this case from a long coil of high-resistivity wire, such as nichrome. The two ends of the resistive element are attached to terminals on the device; the adjustable wiper contact is attached to the remaining terminal.

Figure 1-9 (right): A potentiometer with total resistance R used as a variable voltage divider. As the wiper position is adjusted from bottom to top, v_{out} varies from 0 to v_{in}.

Often we will need a voltage divider with a gain (v_{out}/v_{in}) which is easy to adjust. The *potentiometer* is a circuit element designed for just this job! A potentiometer (or, sometimes, rheostat or variable resistor) is a resistor made from a relatively long, partially-exposed resistive element. An electrical contact (wiper) may be moved along the exposed resistive element from one end to the other, varying the ratio of the resistance between the wiper and one end of the resistor to the total resistance (see Figure 1-8).

Figure 1-9 shows how a potentiometer may be used as a variable voltage divider. In this circuit the output voltage may be varied from 0 all the way up to the input voltage. This is how a volume control is implemented in many audio devices, with v_{in} representing an audio signal (the "wiper position" in a modern device is usually controlled using a digital logic circuit). Other useful potentiometer circuits are shown in Figure 1-10; examples will come up throughout the course.

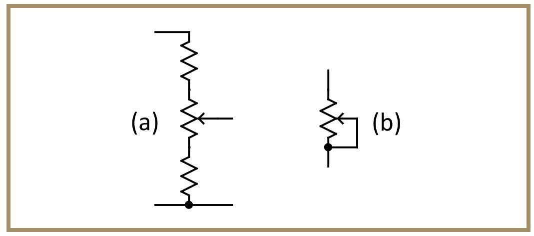


Figure 1-10: Other potentiometer circuits.

(a) adding a resistor in series at either end so that v_{out} varies over a smaller range for finer control

(b) using a potentiometer as a variable resistor

Networks, ports, gain

In Figure 1-7 (on page 1-8) the right-hand voltage divider sub-circuit (portion of a larger circuit) is our first example of a *two-port network*, a common general construction useful for analyzing circuits. Other important examples of two-port networks include amplifiers and filters. Consider Figure 1-11, which explicitly shows how we identify our voltage divider as a network with a single *input port* and a single *output port*. Each "port" of a network comprises two terminals which are meant to be connected into some larger, surrounding circuit. This surrounding circuit will, in general, inject signals (voltages and currents) at our network's input ports, and will it respond to signals emitted from the network's output ports.

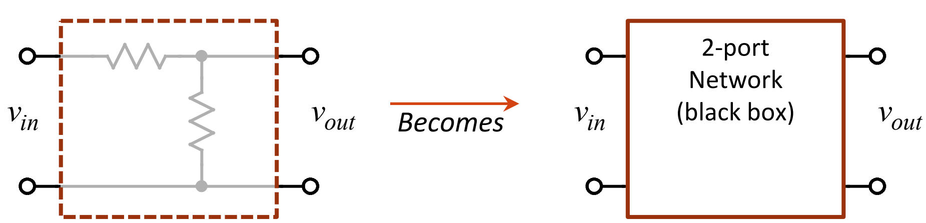


Figure 1-11: The voltage divider is a form of *two-port network*. It has a pair of input terminals, the *input port*, and two output terminals: the *output port*. In this example, the network's transfer function (or gain function) is the ratio v_{out}/v_{in}, as already mentioned in Figure 1-7, page 1-8.

This network concept is useful because we can often describe its behavior (as far as the external, surrounding circuit is concerned) with just a few equations or parameters and otherwise ignore its detailed internal construction. The network becomes a "black box" with inputs and outputs whose relationships are known, but we don't have to bother with the messy details of what's inside the box. This is exactly how we are going to handle operational amplifiers in the next section!

> The most important parameter we will use to describe a "generic" two-port network is its *gain*, or, more generally, its *transfer function*, which describes the functional relationship between its output and its input.

If the network is *linear* (as is the voltage divider) the gain function becomes a simple, fixed ratio independent of the size of the input. For example, the gain of the voltage divider (v_{out}/v_{in}) depends only on the resistors' values and is therefore independent of the magnitude of v_{in} (see Figure 1-7). Other important network parameters (discussed in a later section) are its *input impedance* and *output impedance*.

One final comment about the depiction of a network and its ports: often, one terminal of a port will be directly connected to *ground* (the 0-Volt reference) somewhere inside the network's circuitry. In this case it is common to show only one terminal for that port, the ground connection being understood as the port's other terminal, as shown in Figure 1-12. Since ground is the 0-Volt reference, the voltage of the one explicitly-depicted terminal is also the voltage present at the port (right-hand circuit in Figure 1-12). If neither terminal is actually connected internally to ground, as in Figure 1-11, then the voltage at the port is the difference in the voltages of its two terminals.

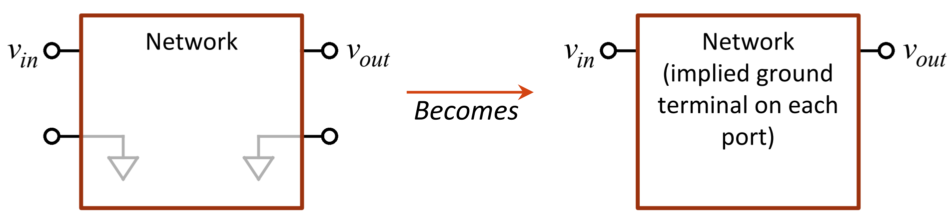


Figure 1-12: A network whose ports each use *ground* as one terminal. In this case the ground terminal is often not shown explicitly, so only a single terminal is shown for such a port. Since ground is our circuit's 0-Volt reference, the voltage of the single terminal is sufficient to determine the voltage of the port.

THE OPERATIONAL AMPLIFIER

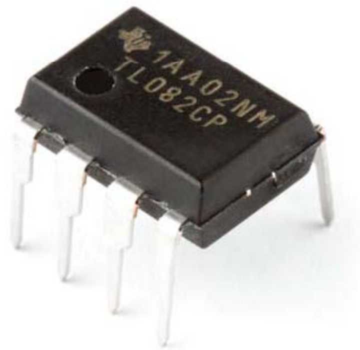


Figure 1-13: A photo of an IC operational amplifier, the Texas Instruments Inc.'s TL082 device. This integrated circuit is the type you'll use for your circuits in this experiment; it actually contains two independent op-amps in the package shown (called an 8-pin *DIP*, for "dual-inline package"). The spacing of the connector pins in one row is 0.10 inch (2.54 mm); the two rows are 0.30 inch apart. (Photo courtesy Texas Instruments Inc., ©2012)

The ideal op-amp

The most important single element we'll use for our analog circuit designs is the *operational amplifier* (op-amp). Modern operational amplifiers are examples of analog *integrated circuits* (ICs), wherein an entire network of dozens (or even hundreds) of transistors, resistors, and even capacitors is created on a single small silicon wafer. The wafer is then mounted inside a (usually) plastic package with several external metal pins used to make electrical connections to the wafer's circuitry (Figure 1-13).

Modern IC op-amps are the culmination of decades of improvements and innovations by hundreds of electrical engineers at dozens of companies; they have outstanding linearity, gain, bandwidth, and noise performance (these terms will mean more to you as the course goes on). Because of this spectacular performance available for our designs, we will first learn how to design amplifier circuits using an *ideal operational amplifier*, which is, naturally, an idealization of an actual op-amp's behavior. As you'll discover when you get to the lab, your real op-amps will perform so well that your results will approach very closely to this ideal!

You should envision our ideal operational amplifier to operate like the "cartoon" diagram in Figure 1-14; right now we will consider the op-amp to be a 4-port network. There is an input port whose two terminals are labeled +*Input* and −*Input* (with voltages of v_+ and v_-, respectively). You should think of these two terminals as being connected to a *perfect voltmeter* inside the op-amp; this voltmeter measures the voltage difference between the two terminals as indicated in Figure 1-14. By "perfect" we mean that the voltmeter is sensitive to any tiny difference in the two terminals' voltages, and that *it does not draw any current at all*

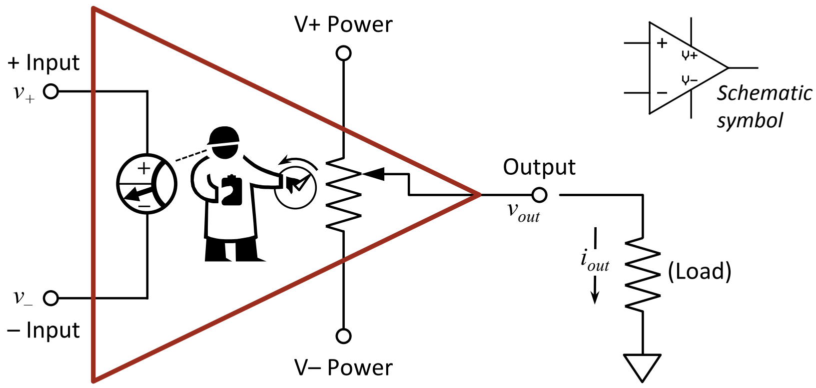


Figure 1-14: Cartoon illustrating the ideal op-amp's behavior. A "voltmeter" monitors the potential difference between the + and – inputs; if the two input voltages match, then the output voltage remains unchanged. If there is a voltage difference between the two inputs, then the op-amp *very quickly* changes the output voltage v_{out}: increasing v_{out} if $v_+ > v_-$, decreasing it if $v_+ < v_-$. The output stops changing only when the two input voltages again match (or when the "output potentiometer" has been adjusted all the way to one of its limits: a power supply terminal voltage). Since the source of the output comes from the op-amp's two power supply terminals, electrical power required by the output load comes from the op-amp's power supply, *not the inputs*. Output current supplied by the op-amp is returned to the power supply through the load, as shown (via the ground attached to other terminal of the load).

from whatever circuit is connected to an input terminal — each input has *infinite input impedance*.

The ideal op-amp also has two power supply ports: the *V+ Power* and the *V− Power* terminals and their associated, *implicit* ground terminals (implicit ground terminals like those in Figure 1-12 on page 1-10). The *V+ Power* and *V− Power* terminals will always be connected to a DC power supply for our circuits; the lab's circuit design trainer has *+12V and −12V power with a common ground connection* as shown way back in Figure 1-3 on page 1-4. You should think that inside the op-amp these two power terminals are connected to either end of a potentiometer as shown in Figure 1-14; the potentiometer's wiper is connected to the op-amp's *Output* terminal (the *Output* port also has an implicit ground terminal).

Now think of some little "technician" ensconced inside our ideal op-amp whose only job is to watch the input voltmeter and move the potentiometer's wiper depending on what the meter shows. If the *+Input* and *−Input* terminals have exactly the same voltage (so the voltmeter reads 0), then the technician stops moving the wiper or leaves it where it is; *the output terminal is thus going to be at some constant voltage between those of the V+ Power and the V− Power terminals*. If there is a voltage difference shown by the input voltmeter, then the technician starts moving the potentiometer wiper *very rapidly* — toward *V+ Power* if $v_+ > v_-$, or toward *V− Power* if $v_+ < v_-$ (see Figure 1-14). For the ideal op-amp, *the output voltage will change infinitely quickly as long as the two input terminals are at different voltages*. And that's it! That's all that the ideal op-amp is supposed to do!

THE IDEAL OPERATIONAL AMPLIFIER'S CHARACTERISTICS AND BEHAVIOR

- The two input terminals (*+Input* and *–Input*) draw 0 current from the external circuit (they each have infinite input impedance).
- The Output voltage is constant whenever $v_+ = v_-$.
- The Output voltage *increases* infinitely quickly whenever $v_+ > v_-$.
- The Output voltage *decreases* infinitely quickly whenever $v_+ < v_-$.
- The power for the Output comes from the *V+ Power* and *V– Power* terminals.

The voltage follower and negative feedback

So what can we do with such a thing as our ideal op-amp? First consider a very simple circuit known as the *voltage follower* (Figure 1-15). It is seemingly trivial — the op-amp's output is connected back to its *–Input* terminal (so that v_- will always equal v_{out}), and some sort of *input signal* is connected to the op-amp's *+Input*, so $v_+ = v_{in}$.

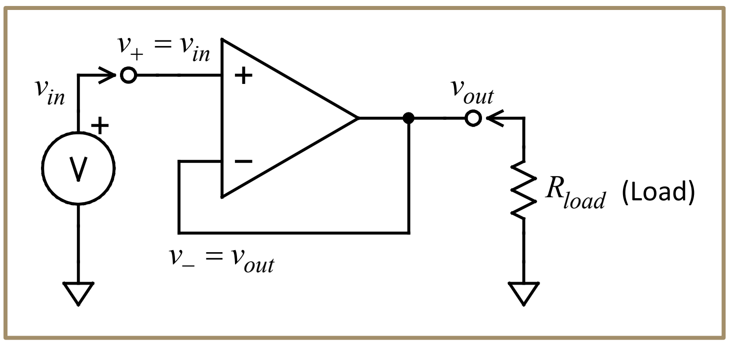


Figure 1-15: The *voltage follower* amplifier circuit. Note that the op-amp's *–Input* terminal is connected directly to its *Output* terminal, whereas the input voltage source is connected directly to the *+Input* terminal (the op-amp's two power supply terminals are not shown, but they still must be connected to a power supply!). If $v_{out} = v_{in}$, then $v_+ = v_-$, and the op-amp maintains the output voltage, v_{out}. If the input voltage v_{in} changes, then momentarily $v_+ \neq v_-$, and the op-amp rapidly changes v_{out} *in the same direction as the change in* v_{in} until the condition $v_{out} = v_{in}$, is restored. Thus the amplifier always keeps $v_{out} = v_{in}$, so the circuit's voltage gain $G = 1$.

If $v_{in}(t)$ is actually constant, then clearly an equilibrium condition for the op-amp would be $v_{out} = v_{in}$, because then $v_- = v_+$, and v_{out} would remain constant. But what if $v_{in}(t)$ changes or there is some perturbation in v_{out} so that, momentarily at least, $v_{out} \neq v_{in}$? Because we have connected v_{out} to the $-Input$ terminal (not $+Input$!), then if, for example, $v_{in} > v_{out}$, we would also have $v_+ > v_-$, so the op-amp would quickly increase v_{out} until the condition $v_{out} = v_{in}$ is restored. Similarly, the op-amp would correct the opposite condition, $v_{out} > v_{in}$.

Thus it would always be the case that the voltage follower circuit (Figure 1-15) will maintain $v_{out}(t) = v_{in}(t)$, so the voltage gain of this simple amplifier is $G = v_{out}/v_{in} = 1$.

Make sure you study Figure 1-15 in light of the ideal op-amp behavior (box on page 1-13) until you have convinced yourself that the voltage follower will always maintain $v_{out}(t) = v_{in}(t)$ (unless v_{in} exceeds the limits set by the op-amp's power supply voltages, which determine the maximum range of v_{out}).

The voltage follower circuit has a stable, linear relationship between v_{out} and v_{in} because the op-amp's output voltage is connected back to its *–Input* terminal. This arrangement is an example of *Negative Feedback*, which is the secret to the versatility of the op-amp.

How could an amplifier with a gain of 1 add any value to a system? Actually, you will find this circuit to be very useful and will possibly include one or more voltage followers in your final project design. The reason it is so useful is because the ideal voltage follower amplifier has *infinite power gain*: Since the input source is connected only to the op-amp's +*Input* terminal, which draws no current, the power required from the input source is $P_{in} = v_{in} i_{in} = v_{in} \cdot 0 = 0$, whereas the power delivered to the load attached to the amplifier's output is $P_{out} = v_{out} i_{out} = v_{out}^2 / R_{load} > 0$: the power gain $P_{out}/P_{in} \to \infty$. The current drawn by the amplifier output's load is supplied by the op-amp's power supplies, *so no power is required from the input source*.

The noninverting op-amp amplifier

Now that you understand how the voltage follower works, let's design a more general and flexible negative feedback setup using our ideal op-amp. Consider the circuit in Figure 1-16, where we now use a voltage divider consisting of resistors R_f and R_i to feed back only a fraction of v_{out} to the −*Input* terminal (assume that a source voltage and a load resistor are again connected to our amplifier like those in the voltage follower circuit, Figure 1-15). The

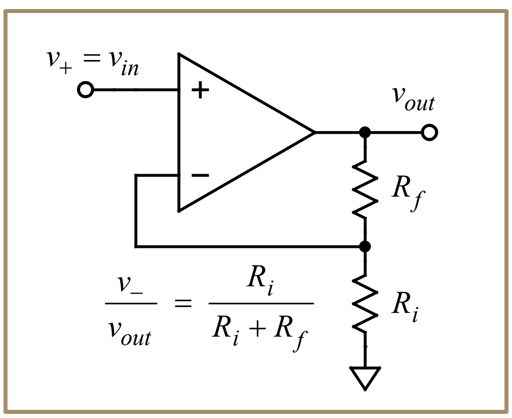


Figure 1-16: The general *noninverting amplifier* circuit. Now the op-amp's *–Input* terminal is connected to the output via a simple voltage divider circuit, so only a fraction of v_{out} is used as the *negative feedback* signal. As with the voltage follower (Figure 1-15), the input voltage source is connected directly to the *+Input* terminal. When the equilibrium condition $v_+ = v_-$ obtains, v_{out} will be larger than v_{in} by the factor $G = (R_i + R_f)/R_i = 1+(R_f/R_i)$, which is thus the *gain* of this amplifier.

op-amp's output voltage, v_{out}, will be stable when $v_{-} = v_{+}$, as before. We still have $v_{+} = v_{in}$, but now we must use the voltage divider equation (see Figure 1-7 on page 1-8) to determine v_{-} from v_{out}; the resulting relation is shown in Figure 1-16. So at equilibrium v_{-} is smaller than v_{out} by a ratio determined by the voltage divider, and, since $v_{-} = v_{+} = v_{in}$, we see that v_{in} must be smaller than v_{out} by this same ratio. Thus we now have an amplifier with a voltage gain G of whatever we want it to be (although $G \geq 1$): we just choose an appropriate pair of values for the resistors R_f and R_i (see equation 1.5) The amplifier is referred to as *noninverting* because v_{out} has the same sign as v_{in}.

Ideal, noninverting amplifier gain

1.5

$$\boxed{G = \frac{v_{out}}{v_{in}} = \frac{R_i + R_f}{R_i} = 1 + \frac{R_f}{R_i}}$$

Note that some current from the op-amp's output must flow to ground through the noninverting amplifier's voltage divider. *This current demand upon the output will add to the current required by the amplifier's load.* In this case the current through our feedback network will be $I_f = v_{out} \big/ (R_f + R_i)$, since the two resistors are in series (remember, 0 current flows to the ideal op-amp's $-Input$ terminal); using larger values for these resistors will reduce the required feedback current.

The inverting op-amp amplifier

Let's start with the noninverting amplifier of Figure 1-16, but instead of connecting the input signal to the op-amp's *+Input* terminal, let's connect it to the bottom end of the feedback voltage divider; we then ground the *+Input* terminal. The result is the *inverting amplifier* circuit of Figure 1-17 (note that we've rearranged the locations of the components and

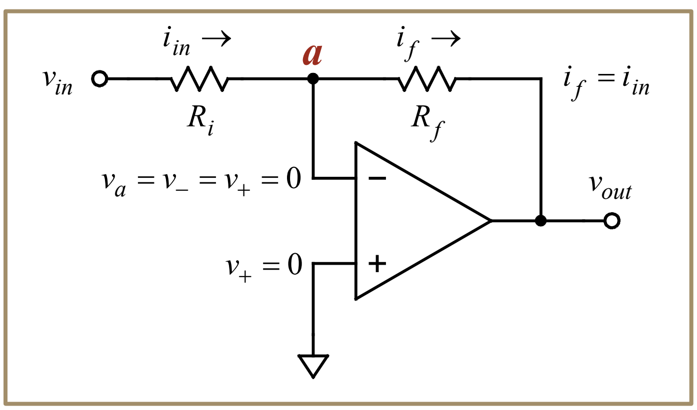


Figure 1-17: The general *inverting amplifier* circuit. The input signal is attached to one end of the negative feedback voltage divider, and the *+Input* terminal is connected to ground. At equilibrium $v_+ = v_-$, and the point a becomes a *virtual ground*, since its voltage will be 0 as well. Since no current flows into the op-amp's *–Input*, the current from the input source (i_{in}) must also flow through R_f to the op-amp's output terminal. The voltage across R_i is just v_{in}; the voltage across R_f must be (R_f / R_i) times larger, because the currents are the same. Thus the gain $G = -(R_f / R_i)$.

flipped the op-amp symbol so that the −*Input* terminal is above the +*Input*). You should closely compare the two circuits to convince yourself that the only change has been to swap the ground and source input connections. The inverting amplifier is a little harder to analyze, but there are a couple of clever shortcuts you can use to quickly derive the gain. These "tricks" are quite useful when analyzing op-amp circuits, so let's carefully consider them.

First, assume that the negative feedback works effectively, so that the op-amp continuously adjusts its output voltage as needed to maintain its equilibrium input condition $v_- = v_+$. This implies, as shown in the figure, that the voltage at *node* (connection) *a*, where the two resistors are joined to the −*Input*, will be 0 (ground), *even though there is no direct connection of this point to ground*. For obvious reasons, therefore, the node *a* is called a *virtual ground*. Now we know the potential across resistor R_i: $v_{in} - 0 = v_{in}$, so we immediately know that $i_{in} = v_{in} / R_i$.

Now comes the second trick: since no currents flow into an ideal op-amp's input terminals, the only place for the current i_{in} to go is to continue on through R_f, so $i_f = i_{in}$, and we now know the potential across R_f: $R_f\, i_f = 0 - v_{out} = -v_{out}$. Thus we have derived the gain for the *ideal, inverting amplifier*:

Ideal, inverting amplifier gain

1.6 $$\boxed{G = \frac{v_{out}}{v_{in}} = -\frac{R_f}{R_i}}$$

The amplifier is call *inverting* because the sign of the output voltage is the opposite of the sign of the input voltage. The magnitude of this circuit's gain may be chosen to be anything by picking values for R_f and R_i, whereas the gain of the noninverting amplifier must be at least 1 (compare equation 1.5). There is one significant drawback to the inverting amplifier circuit, however: *the current drawn from the input source is not zero*. In other words, the *input impedance* (v_{in} / i_{in}) of this amplifier is finite — in fact, it is equal to R_i. The concept of input impedance will be discussed in the next section, although we'll forego the general definition of *impedance* until Experiment 2.

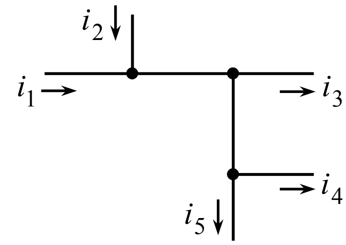
> **Important caveat about the inverting amplifier circuit**: If the source of the input signal has its own resistance (called its *output impedance*, discussed starting on page 1-42), then that resistance will be in series with the circuit input, adding to R_i and reducing the circuit's gain.

You should again convince yourself that the negative feedback from the op-amp output to the –*Input* is such that the ideal op-amp behavior will keep $v_- = v_+ = 0$ as the input voltage changes. Remember, the condition $R_f\, i_{in} = -v_{out}$ is satisfied because the op-amp adjusts v_{out} to make it so: it adjusts v_{out} until $v_- = 0$, and this will be the case only when $v_{out} = -(R_f / R_i)\, v_{in}$. When you are confident that this is how the circuit works, you will have learned what you need to know about op-amps for now.

TECHNIQUES FOR ANALYZING CIRCUITS

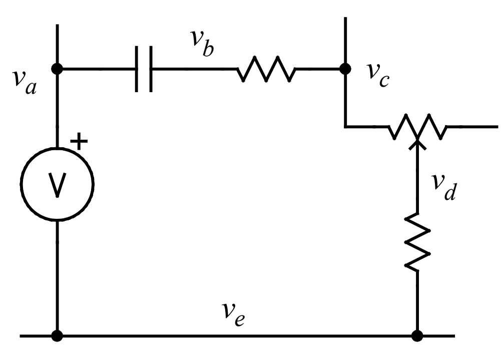
Circuit nodes and loops; Kirchhoff's laws

Now is probably the appropriate time to explicitly state the rules we've been using to determine the relationships between the voltages and currents in our circuits. We've already defined what we mean by a *lumped circuit element*, of which our resistors and op-amps are examples: a lumped element always has *zero net current* flowing into (or out of) it. It may be obvious, but let's state it anyway: the same consideration applies to the connections (called *nodes*) between the terminals of our various elements. For example, the node illustrated on the right connects 5 terminals of some assortment of elements (the placement of the "wires" and their connections at the dots are arbitrary and are chosen to make the schematic as readable as possible — all that matters is that this node makes a common connection to 5 different terminals). Our rule about currents states that, given the arbitrary way we've picked the directions for the current flows toward or away from each terminal, it must be true that $i_1 + i_2 = i_3 + i_4 + i_5$. This rule is commonly known as *Kirchhoff's Current Law*, named for the Prussian physicist Gustav Kirchhoff (1824–1887). If, when we use this rule and solve for the currents, we find that one or more of the currents has a negative value, this result just means, of course, that the actual current flow is opposite to the way we've drawn the arrow.



It also must be noted that for any given node (such as that pictured above), the voltage is the same everywhere along it: i.e., *all terminals connected together by a node are at the same voltage*. In other words, the lines connecting terminals in a schematic are not supposed to represent any sort of "physical" model of real wires with some nonzero resistance. In our real circuits, though, if the physical distance between a pair of elements is large, and the current flow between them is substantial, then the wire you use to connect them may have enough resistance to introduce a noticeable voltage drop; in this case it would be wise to include the wire itself as another element in your schematic and in your calculations.

Another rule relates the voltages across elements whose terminal connections form a closed loop in the circuit (of course, there must be *at least one closed loop* in our circuit, which is why it is called a *circuit*!). Consider a circuit fragment containing a loop like the one at right, where we've also labeled the voltage at each node of the loop. The *loop voltage rule* seems trivial when one labels the node voltages; it says that if we



pick any node and add up the voltage differences across the elements' terminals as we go around the loop, the total voltage change must be zero. In other words, if we start at, say, node *e* (at voltage v_e) and then proceed clockwise around the loop, the voltage across the signal source takes us from v_e to v_a; but if we proceed the other way, using the current-voltage relationship appropriate for each of the elements, the calculated voltages across the elements must take us from v_e to v_d to v_c to v_b and, finally, again to v_a, the same as before. This obtains because *the potentials of the nodes are all well-defined and single-valued*, which will nearly always be true as long as our circuit is small compared to the wavelength of any nearby oscillating electromagnetic field (no magnetically-induced EMFs allowed around our circuit loops!). This rule is known as *Kirchhoff's Voltage Law*. Using these current and voltage laws for our nodes and loops along with the current-voltage laws for the various elements (like Ohm's law, equation 1.1) will give equations relating the various currents in the circuit and the voltages at the circuit's nodes.

A SIMPLE EXAMPLE OF HOW TO USE THE VOLTAGE AND CURRENT RULES

Let's illustrate the use of Kirchoff's laws to solve for the unknown voltages and currents in the simple circuit shown in Figure 1-18. In this circuit a voltage source with voltage v_s drives a network of three resistors; we wish to find the values of all of the various voltages and currents in the circuit shown in the figure in terms of the specified resistor values (R_1, R_2, and R_3) and the source voltage (v_s).

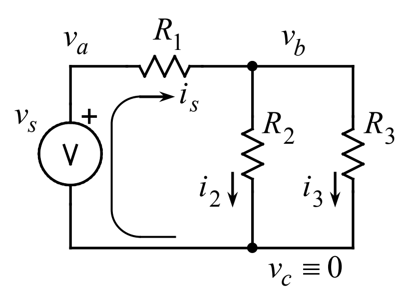


Figure 1-18: A simple example illustrating how Kirchhoff's laws are used to solve for the voltages and currents in a circuit.

The circuit in Figure 1-18 has three nodes: *node a* connects the source and R_1; *node b* connects R_1 to R_2 and R_3; and *node c* connects R_2 and R_3 back to the source. The voltages at these nodes will be designated as v_a, v_b, and v_c, respectively. We'll choose to use v_c as our voltage reference (*ground*), so *by definition* $v_c \equiv 0$, as shown in the figure. Thus we have two unknown voltages: v_a and v_b. Also shown in the figure are three unknown currents: the current supplied to the circuit by the source, i_s (which also must flow through resistor R_1, as shown); and the currents i_2 (through R_2) and i_3 (through R_3).

These five unknowns are connected to the source voltage v_s and to each other by Kirchhoff's voltage and current laws:

(1) Clearly, by going up the left side through the voltage source: $v_a = v_c + v_s = v_s$
(2) Following the current through R_1, the voltage will drop across it: $v_b = v_a - R_1 i_s$
(3) The voltage drop across R_2 takes us back to ground: $v_c = 0 = v_b - R_2 i_2$
(4) Ditto for the voltage drop across R_3: $v_c = 0 = v_b - R_3 i_3$
(5) The sum of the currents into *node b* must vanish, so: $i_s = i_2 + i_3$

Considering the sum of the currents into *node c* gives the same equation as (5), so this additional equation would be redundant.

When writing down loop equations we must be careful about the direction of a current through an element and the sign of its associated voltage difference: if current flow through a resistor is positive (in the chosen direction of the arrow), then the voltage at the arrow's tail must be greater than the voltage at its head, as shown in Figure 1-4 (on page 1-5). Note that equations (2)–(4) comply with this condition.

The opposite is true for a power source: current leaves at the terminal with the more positive voltage and enters at the other.

We have five independent equations for the five unknowns (v_a, v_b, i_s, i_2, and i_3); these equations are straightforward to solve. If we define $R_{\|}$ as the equivalent resistance of the three resistors in parallel:

$$\frac{1}{R_{\|}} \equiv \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3}$$

then a convenient way to express the solutions for the five unknowns is:

1.7

$$v_a = v_s \qquad i_s = \frac{v_s}{R_1}\left(1 - \frac{R_{\|}}{R_1}\right)$$

$$v_b = \frac{R_{\|}}{R_1} v_s \qquad i_2 = \frac{v_s}{R_2}\frac{R_{\|}}{R_1} \qquad i_3 = \frac{v_s}{R_3}\frac{R_{\|}}{R_1}$$

You should take a few minutes to show that this solution satisfies condition (5): $i_s = i_2 + i_3$.

Linear circuits and superposition

The node and loop equations used in the previous example were *linear*, so that all of the various currents and voltages were proportional to the single source signal, v_s. This is generally the case for circuits constructed from linear components like our ideal resistors. It is also true for properly-designed, ideal op-amp amplifier circuits using negative feedback (it is not the case, however, for important nonlinear systems like digital circuits).

Linear circuits are particularly straightforward to analyze, even if there are multiple, independent sources of voltages and currents embedded in them. In the case of a linear circuit excited by multiple sources, the solution for each unknown voltage or current will be given by a sum of terms, each term *proportional to only one source*:

1.8 $$y = a_1 x_1 + a_2 x_2 + \ldots = \sum_k a_k x_k$$

where y is an unknown response (a voltage or current, either at the output or across one of the circuit's components) and the sum is over the various input voltages and currents x_k. The coefficients a_k of the various terms are derived from the circuit's arrangement and its component values but *are independent of the values of the sources*.

Because of this simple, linear structure for the solution, we can often determine the various coefficients a_k quickly by a simple procedure: if all but one input were equal to zero, then each unknown voltage or current is simply proportional to the single, nonzero source – the ratio of the unknown response y to that source is then the value of the corresponding coefficient of its term in equation 1.8. By cycling through each input in turn you can then determine all of the coefficients, thus finding the general solution for the unknown voltage or current. This is the *Principle of Linear Superposition*.

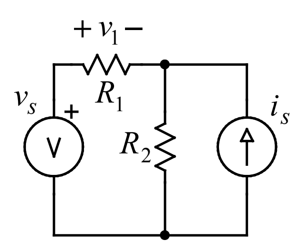
IDEAL VOLTAGE AND CURRENT SOURCES SET TO 0

Setting a *voltage source* to 0 is the same as replacing it with a *short-circuit* (wire) connecting its two terminals.

Setting a *current source* to 0 is the same as replacing it with an *open circuit* (no connection at all) between its two terminals.

Time for a few examples…

A SIMPLE EXAMPLE USING LINEAR SUPERPOSITION

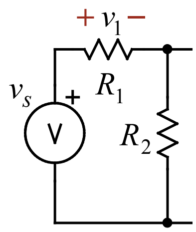


Consider the circuit shown at right, with input voltage and current sources joined by a couple of resistors. We want to solve for the voltage drop v_1 across the resistor R_1 when the two sources are v_s and i_s (with the polarities shown). According to the general expression 1.8, we can write v_1 as:

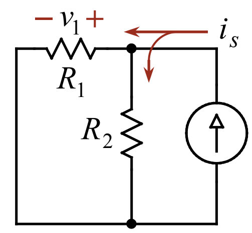
$$v_1 = a\,v_s + b\,i_s$$

for some parameters a and b which can only depend on the two resistor values. We can first calculate the value of a by setting i_s to 0 (so $v_1 = a\,v_s$), and then determining v_1 in terms of

v_s. Similarly, we get b by setting v_s to 0. Following the directions in the box on the previous page concerning the sources, the equivent circuits for the two cases are:



$i_s \Rightarrow 0 \qquad v_1 = \frac{R_1}{R_1+R_2} v_s$



$v_s \Rightarrow 0 \qquad v_1 = \frac{R_1 R_2}{R_1+R_2} i_s$

Setting i_s to 0 (left diagram above) results in a simple voltage divider with input voltage v_s, so the voltage across R_1 is given by the voltage divider equation, as shown. Setting v_s to 0 (right diagram) simply connects the two resistors in parallel, and their equivalent resistance is driven by the current i_s. The voltage across the parallel pair is then given by Ohm's law, and this voltage would also equal v_1. Note, however, that the polarity of the voltage across R_1 due to i_s is opposite to that induced by v_s, as indicated in the two diagrams. Now use linear superposition: the voltage across R_1 due to the simultaneous application of sources v_s and i_s is given by the sum of these two results, being careful of the relative polarities of their contributions. This gives us the final solution for the voltage across R_1:

$$v_1 = \frac{R_1}{R_1+R_2}\left(v_s - R_2 i_s\right)$$

THE INVERTING, SUMMING AMPLIFIER

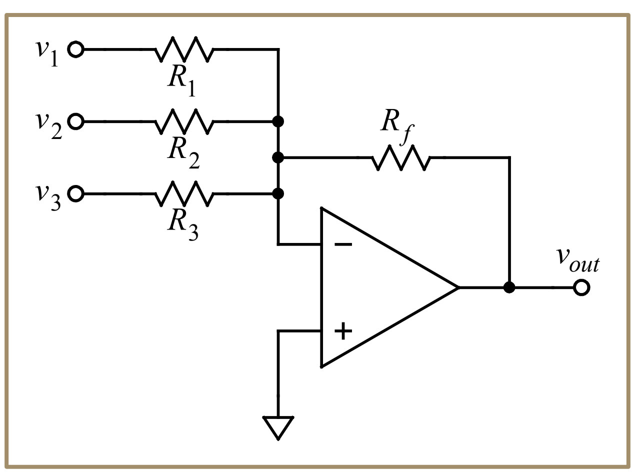


Figure 1-19: The inverting, *Summing Amplifier*. The node joining the resistors to the *–Input* is a *virtual ground*, so each input will be amplified independently of the others. The result is a weighted sum of the several inputs (inverted, of course) given in equation 1.9.

Now for a more important example of linear superposition. For this example we start with the inverting amplifier configuration, but this time with multiple input resistors and sources as shown in Figure 1-19 on page 1-21. We can solve for the circuit's response to each of its various inputs by setting all but one to zero and using superposition.

As discussed previously, the node connecting the resistors to the op-amp's *–Input* is a *virtual ground*, with 0 voltage. Consequently, for each input set to 0 the associated input resistor (R_1, etc.) will have no voltage across it, *so the current through it must vanish as well*. If no current flows through a resistor, then it could be removed without affecting the circuit! Thus, each zeroed input can have no effect on the amplifier's behavior, so the amplifier will behave as a straightforward, inverting amplifier for the one active input, with gain given by equation 1.6. By superposition, we can add such an inverting gain term for each input to give the result we seek for v_{out}, a weighted sum of the various inputs:

Ideal, inverting, summing amplifier

1.9
$$\boxed{v_{out} = -\sum_k \frac{R_f}{R_k} v_k}$$

As with our original discussion of the inverting amplifier, the current drawn from each input source will not be zero — each voltage source sees a load resistance equal to its input resistor R_k. If a source itself has nonzero output resistance, then that resistance must be added to its R_k before using equation 1.9 (see the section *Output resistance* on page 1-42).

GENERALIZING THE VOLTAGE DIVIDER

As with the previous example, the ubiquitous voltage divider often appears in circuits in a more general form: several voltages connected to a single node through resistors or other components as shown in Figure 1-20. We would like a relatively simple, easy-to-remember formula for the resulting node voltage.

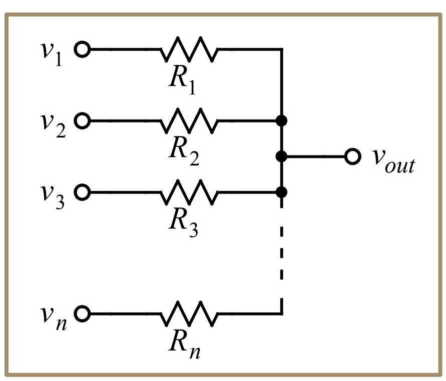


Figure 1-20: A generalization of the voltage divider circuit. Several voltage sources are connected to a single node via resistors, and we want to know the resulting voltage at that node (v_{out}).

Assume no current flows out of the terminal at v_{out}. The resulting voltage v_{out}, whose derivation we leave for the exercises, is then a weighted average of the various input voltages:

Generalized voltage divider output

1.10
$$v_{out} = \frac{\sum(v_k/R_k)}{\sum(1/R_k)}$$

The weight of each voltage source is simply the conductance ($1/R$) of its connection to the common node. If all resistors have the same value, then v_{out} will be the arithmetic mean of the input voltages.

> You must be careful when designing with this circuit, because the current it draws from each source will depend on the values of the other source voltages: $i_k = (v_k - v_{out})/R_k$, and v_{out} depends on the values of all of the inputs. It is possible that current will flow into a source even when its voltage is positive!

Keeping the above caveat in mind, we can construct a noninverting conterpart to Figure 1-19:

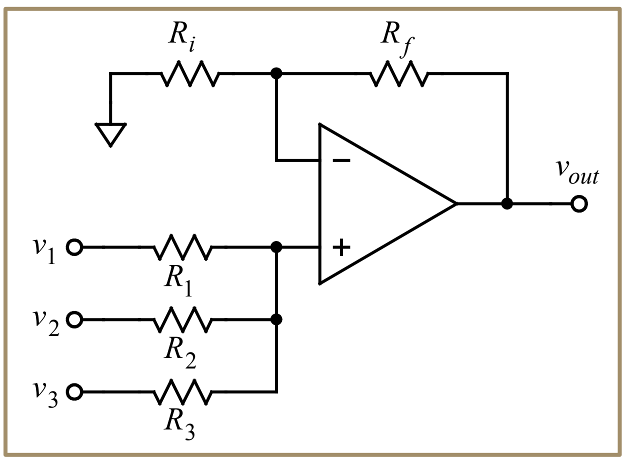


Figure 1-21: The noninverting summing amplifier. The node joining the resistors to the +*Input* has the voltage given by the formula for v_{out} in 1.10. Using a voltage follower configuration (R_i removed and $R_f = 0$) would give an output voltage given by the weighted average formula 1.10.

If all of the resistors R_k have the same value, then the voltage at the op-amp's +*Input* will be the mean of the input voltages. This will then be multiplied by the noninverting amplifier gain $(1+R_f/R_i)$ to result in the op-amp's v_{out}, that is $(1+R_f/R_i)$ times the result from 1.10.

THE DIFFERENTIAL AMPLIFIER

For our final example consider an ideal op-amp amplifier, but this time we attach two independent voltage source inputs as shown in Figure 1-22 on page 1-24. We now want to

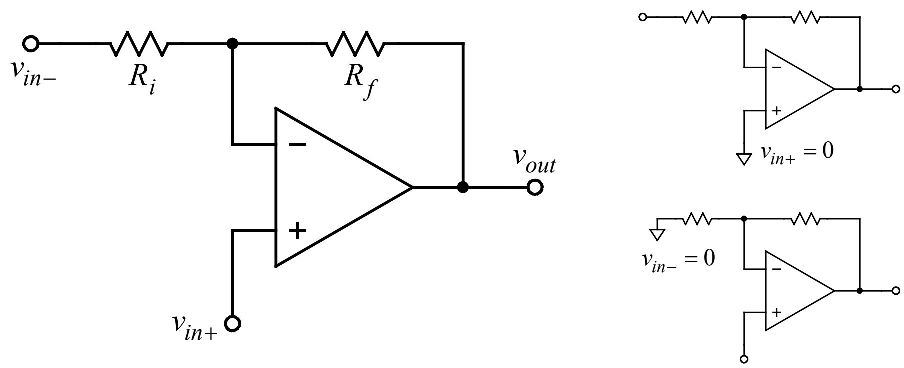


Figure 1-22: A combination of a noninverting and an inverting amplifier. If input signal $v_{in+} = 0$, then the circuit is an inverting amplifier for signal v_{in-} (top right diagram), and $v_{out} = -(R_f / R_i)\, v_{in-}$. If, instead, $v_{in-} = 0$, then the circuit is a noninverting amplifier for signal v_{in+} (bottom right diagram), and $v_{out} = (1 + R_f / R_i)\, v_{in+}$. By the *principle of linear superposition*, the response at v_{out} must be the sum of these two expressions: $v_{out} = (1 + R_f / R_i)\, v_{in+} - (R_f / R_i)\, v_{in-}$.

know what the output voltage v_{out} will be for any combination of values for the two input voltages v_{in-} and v_{in+}.

Because the circuit is linear, the solution for v_{out} will again be a sum of terms like equation 1.8. We will use the principle of linear superposition to obtain this solution by setting v_{in+} and v_{in-} to 0 in turn. As we've said before, setting a voltage source to 0 is nothing more than replacing it with a short circuit (a wire) connecting its two terminals. The input voltage sources in Figure 1-22 each have one terminal connected to ground (that's why we show only a single terminal for each source), *so setting one of them to 0 is the same as connecting that circuit input to ground.*

Thus if $v_{in+} = 0$, the circuit becomes identical to the inverting amplifier in Figure 1-17, and the inverting amplifier gain formula in equation 1.6 will describe how v_{out} depends on v_{in-}. Conversely, $v_{in-} = 0$ results in a noninverting amplifier like Figure 1-16 with gain given by equation 1.5. Linear superposition then implies that v_{out} will vary as the sum of these two expressions, so that for arbitrary v_{in+} and v_{in-} we will get:

$$v_{out} = \left(\frac{R_i + R_f}{R_i}\right) v_{in+} - \left(\frac{R_f}{R_i}\right) v_{in-} = \frac{R_f}{R_i}\left[\left(\tfrac{R_i + R_f}{R_f}\right) v_{in+} - v_{in-}\right]$$

This result is almost proportional to the difference in the two input voltages $(v_{in+} - v_{in-})$, but not quite. If we were to first scale v_{in+} by $R_f / (R_i + R_f)$, then the output would indeed be proportional to the input voltage difference, and we would have designed a *differential amplifier*. But this correction factor is just what we would get if we were to add a voltage divider with resistors R_i and R_f between v_{in+} and the op-amp's *+Input*, as shown in Figure 1-23.

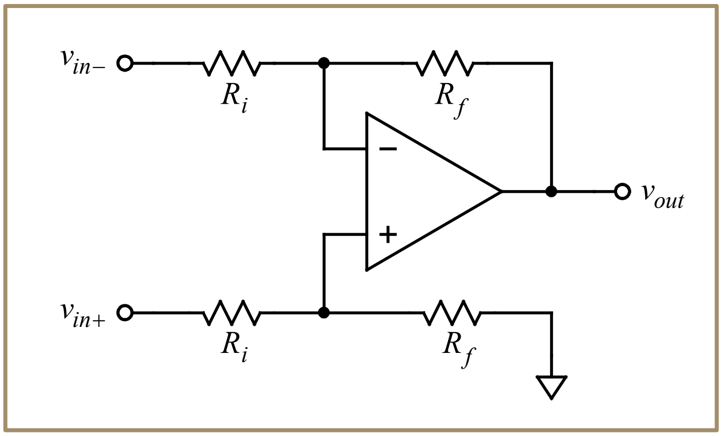


Figure 1-23: The *Differential Amplifier*. The added voltage divider on the v_{in+} input will divide it by just the right amount so that the output is proportional only to the *difference* between the two input signals, equation 1.11.

Because the ideal op-amp's *+Input* draws no current, the voltage at that input will be given using the basic voltage divider equation presented in Figure 1-7 on on page 1-8, reducing v_{in+} by the correct factor to give a purely *differential* gain, equation 1.11.

Ideal, differential amplifier

1.11
$$\boxed{v_{out} = \frac{R_f}{R_i}\left(v_{in+} - v_{in-}\right)}$$

Note one important point about the differential amplifier, however: the output will be given by (1.11) *only if the resistor values are exactly matched* so that the voltage divider on the v_{in+} input exactly compensates for the extra gain of the noninverting amplifier action. The resistors you will use have values whose tolerance is 5% (or maybe 1%), so their actual values may differ from the marked values by that percentage. In practice this implies that even though you may have $v_{in+} = v_{in-}$, you would have nonzero v_{out}. Thus inaccurate matching of the resistor values results in a nonzero *common mode gain* so that v_{out} will include a residual term proportional to the average value of the two input voltages. The ratio of the amplifier's *differential gain*, $G_{diff} = v_{out}/(v_{in+} - v_{in-})$, to its common mode gain, $G_{cm} = 2v_{out}/(v_{in+} + v_{in-})$, is known as its *Common Mode Rejection Ratio* and is an important specification when choosing or designing a differential amplifier.

You will need to use the powerful *principle of linear superposition* throughout this course to solve circuit problems and properly design your circuits.

Input resistance

We've already mentioned the terms *input impedance* and *output impedance* in our discussion of the "black box" description of a network (or sub-circuit). Now is the time to start to understand what they mean and why they are important, at least in the context of the circuits we've analyzed so far. We can't actually define *impedance* yet (that will have to wait for Experiment 2), but we can talk about *input resistance* and *output resistance*, which are closely-related concepts (in order to restrict ourselves to *resistances*, we will have to assume that our power sources output constant (DC) voltages and currents, and that we wait long enough for any transient, time-varying behavior of the circuit to die away before we take any measurements). In this section we discuss *input resistance*; *output resistance* is a slightly more advanced topic and is addressed in the section starting on page 1-42.

When we connect a power source to an input of a network or circuit, the source will, in general, apply some voltage across the two input terminals, and some current will flow.

> The *input resistance* of an input port of a network or device is *the ratio of the applied voltage and the resulting current flowing into that input*.

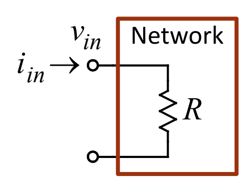
Input Resistance

1.12

$$\boxed{R_{in} \equiv \frac{v_{in}}{i_{in}}}$$

> **THE IMPORTANCE OF INPUT RESISTANCE**
>
> Input resistance determines how much power must be supplied by an input source. For example, some voltage sources are very weak and cannot provide a significant amount of current. In this case, the input resistance of the circuit must be made high enough to not load down the source.
>
> In other cases (usually involving high frequencies or short signal pulses), the circuit's input resistance must be chosen to match the *characteristic impedance* of the cable connecting the source. Otherwise, the signal will be reflected by the circuit input, wasting signal power and greatly distorting the signal shape. Typically, the input resistance in these cases should be 50Ω.

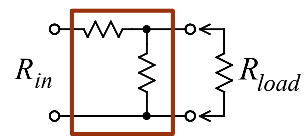
If our circuit is linear and if there is only one input power source, then R_{in} as defined in 1.12 will not vary with the magnitudes of v_{in} and i_{in}. For example, if our network consists of a single resistor R, as shown at right, then (trivially) connecting an input voltage source v_{in} to its terminals will result in current $i_{in} = v_{in}/R$, and, as expected, $R_{in} = R$.



For a less trivial example, consider the input resistance of the 3-resistor network in Figure 1-18 on page 1-18. With the input voltage to that network $v_{in} = v_s$ and resulting current $i_{in} = i_s$ (refer to the figure for the definitions of v_s and i_s), then the solution to that example, equations 1.7 on page 1-19, shows that:

$$R_{in} = \frac{v_s}{i_s} = \frac{R_1}{1 - R_{\parallel}/R_1} = R_1 + \frac{R_2 R_3}{R_2 + R_3}$$

which we could have anticipated by noting that the network simply consists of resistor R_1 in series with the parallel combination of resistors R_2 and R_3.



In general, the input resistance R_{in} of a network port will depend upon what is connected to the network's other ports, including its output ports. For example, let's examine the simple voltage divider considered as a two-port network (at right). Clearly, its input resistance R_{in} will depend on the value of the load R_{load} connected to the divider's output port, since the load resistance is in parallel with one of the voltage divider's two resistors. In fact, this network (including the output load resistor R_{load}) is the same as that in Figure 1-18, with R_{load} assuming the role of that example's R_3. This particular situation (input resistance changing if the load changes) can be avoided by adding a voltage follower as in the third example in Figure 1-24 below: the op-amp *isolates* the input from the output load.

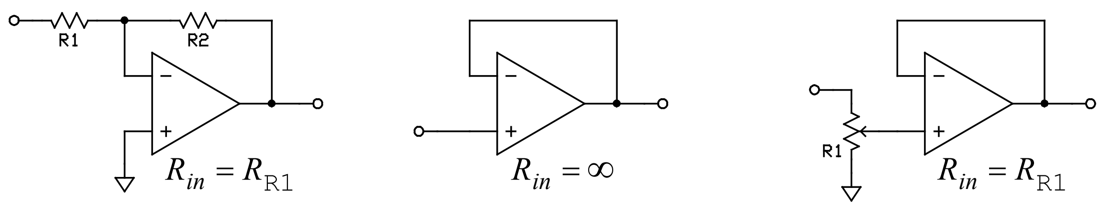


Figure 1-24: Amplifier circuit examples illustrating the *Input Resistance* concept. Inverting amplifier: since the *–Input* node of the op-amp is a virtual ground, the input resistance of the amplifier is just the value of R1, the input resistor. Voltage follower: since the op-amp inputs draw no current, the input resistance is infinite. Potentiometer as a variable voltage divider connected to a voltage follower: again, no current flows into the *+Input*, so the input resistance is given by the total potentiometer resistance R1. In each of these cases, the amplifier isolates the output load from its input, so the input impedance is unaffected by the current required by the load connected to its output.

PRELAB EXERCISES

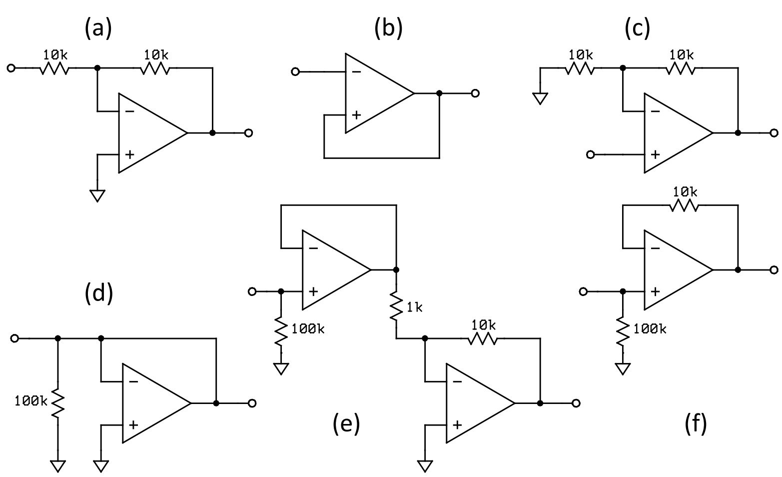
1. The lab power supplies you will use for your op-amp circuits supply $+12\,\mathrm{V}$ and $-12\,\mathrm{V}$ at up to $250\,\mathrm{mA}$ each. How many total watts of power is this?

   If the resistors you use can absorb no more than $1/4\,\mathrm{W}$ without damage, then what is the *minimum* resistor value which can be connected between the $+12\,\mathrm{V}$ supply and ground without damage? between the $+12\,\mathrm{V}$ supply and the $-12\,\mathrm{V}$ supply?

2. If you take a resistor R and add another resistor with resistance $0.1\times R$ in series with it, the total resistance of the pair is obviously $1.1\times R$, a 10% increase. If instead you wish a combination which has a total effective resistance of only $0.9\times R$, you could do it by placing a second resistor in parallel with the original R. In this case, what should be the value of this second resistor?

3. For the inverting amplifier circuit shown in Figure 1-17 on page 1-15, the input source v_{in} must supply the current i_{in} flowing through input resistor R_i. As described in the text this same current must continue on through the feedback resistor R_f (as current i_f). Where does the current i_f go from there? Assume no load is attached at the op-amp output.

4. Which of the following amplifier circuits will work correctly (do something useful)? Which won't, and why not? For those that do work correctly, what is the amplifier gain (use a '−' sign for an inverting gain)? the input resistance? Look at them carefully!



5. Refer to Figure 1-20 on page 1-22 and derive the generalized voltage divider formula, equation 1.10 (repeated below).

1.10
$$\boxed{v_{out} = \frac{\sum(v_k/R_k)}{\sum(1/R_k)}}$$

Hint: a clever way to solve this problem is to convert each of the (v_k, R_k) sources to their *Norton equivalents* (see Figure 1-37 on page 1-44); this will give (current source + parallel resistance) pairs: $(i_k = v_k/R_k, R_k)$. Remember that each source has its other (implicit) terminal connected to ground (⏚), as does the output voltage node, v_{out}. Note that now all the source resistors are in parallel, and so are the source currents! The output voltage becomes the voltage across the parallel resistor combination driven by the sum of the source currents.

6. Consider the circuit below (Figure 1-25), which you will construct and evaluate during lab. What is the the circuit's gain (v_{out}/v_{in}) when the *gain adjust* potentiometer's wiper is set to the top end of its resistance element? Set to the bottom end? Centered?

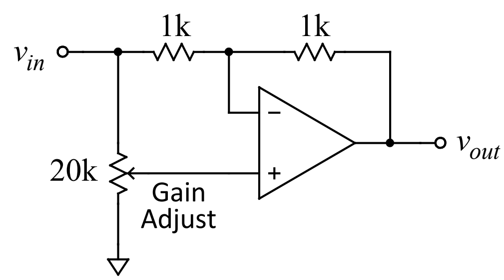


Figure 1-25: How does the gain of this circuit vary as the potentiometer is adjusted?

LAB PROCEDURE

Ask questions during the lab! Don't just sit and stare helplessly at a circuit or piece of test equipment which stubbornly refuses to cooperate!

Overview

During lab you will experiment with various op-amp circuits and evaluate their performance. You will start to become familiar with the analog electronics lab trainer, the signal generator, the oscilloscope, and a couple of the data acquisition and control programs available on the lab computer workstations. For each circuit configuration you investigate, you should use the oscilloscope to measure the input and output peak-to-peak voltages (for oscillating signals; peak-to-peak: difference between maximum and minimum values) or the mean voltages (for DC signals). For some oscillating signals you should also try square waveforms as well as sine waves.

Interesting results may be recorded by saving oscilloscope screen-capture images (see Figure 1-28 on page 1-32). Include circuit diagrams (schematics) and very brief comments regarding your findings in your lab write-up; comments may be hand-written on printed screenshots or other computer-generated graphs. Schematics (with component values and input and output ports labeled) are required for all circuits. You may refer to figures in the notes which show schematics.

Figure 1-26 shows a typical lab station setup. The more quickly you become familiar with the oscilloscope's and signal generator's controls and menus, the more you will enjoy your time in the lab and the more productive you will become. This first experiment is a good one to

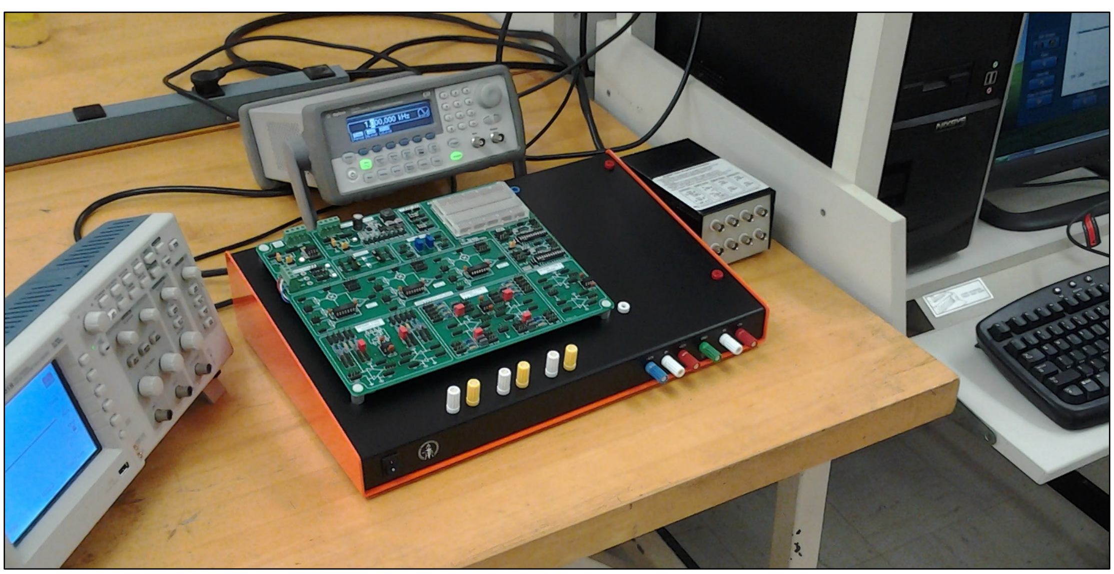


Figure 1-26: A typical lab station setup, with analog electronics trainer and breadboard, oscilloscope, signal generator, and computer with data acquisition and control software.

spend time exploring the instruments' various modes and capabilities. Ask lots of questions!

Using the analog trainer and breadboard

The analog breadboard we use is the Texas Instruments *ASLK PRO*, kindly donated by the company for our use. The manufacturer's web site supporting this system may be found at:
https://university.ti.com/en/faculty/teaching-materials-and-classroom-resources/ti-based-teaching-kits-for-analog-and-power-design/analog-system-lab-kit-pro

The manufacturer's student manual for the breadboard is found here:
http://download.mikroe.com/documents/specials/educational/aslk-pro/aslk-pro-manual-v103.pdf

> The board shows power supply connections as +10V and −10V, but the actual power supply voltages we use in the Caltech lab trainers are **+12V and −12V**.

> The board **Ground** connections are to the common power supply return as shown in Figure 1-3. *They are not connected to* **Earth Ground** *within the breadboard assembly*. A connection to earth ground is provided by the green connector on the front of the trainer chassis.

For this first experiment you will assemble various amplifier circuits by using jumper wires to connect components on the trainer. The photos in Figure 1-27 show several views of a trainer configured with a ×11 noninverting amplifier circuit and connected to a signal generator and oscilloscope. More details about how to properly use the breadboard are provided in the *detailed procedures* section starting on page 1-34.

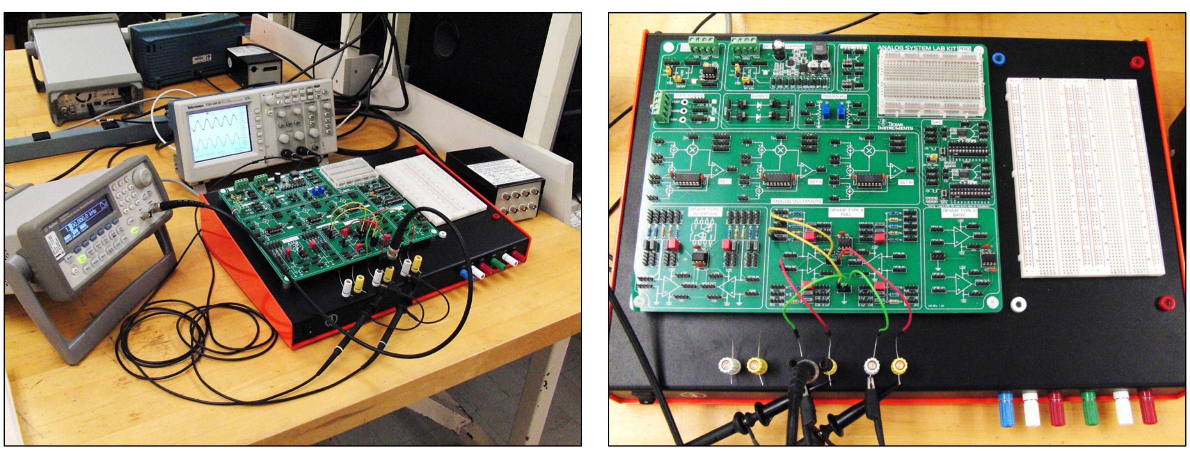


Figure 1-27: The proper way to build circuits using the breadboard's preinstalled component area.
***Left*: overall image of a setup showing the oscilloscope with its two *10× probes* and the signal generator using a *BNC cable* for its connection.**
***Right*: The trainer breadboard area in the lower-center is used for most amplifier designs. It has two op-amps with a selection of associated resistors and capacitors. In this image a ×11 noninverting amplifier is wired up in that area.**

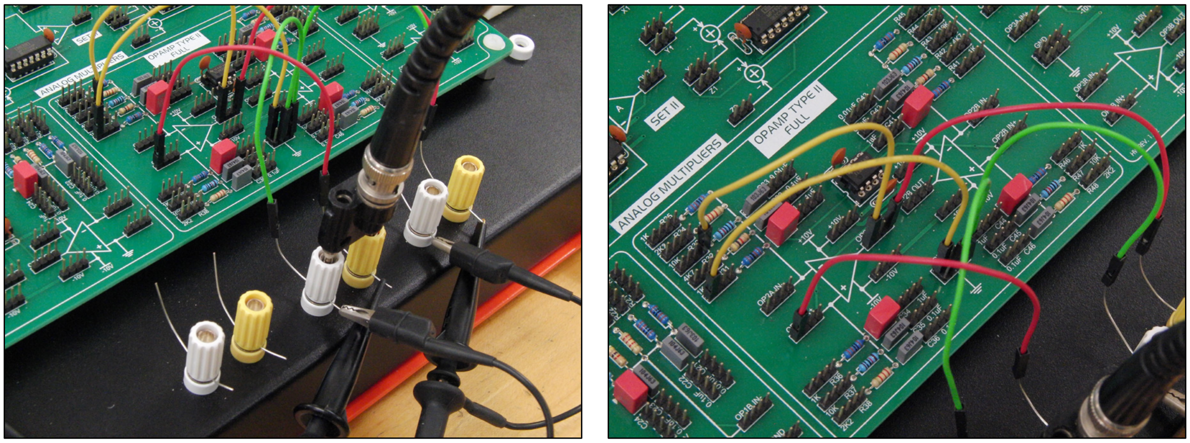


Figure 1-27 (continued).

Left: **detail showing how the *5-way binding posts* are used to connect the circuit to the signal generator and the oscilloscope probes. 22 gauge solid wire is used as a "terminal" to which a circuit jumper and/or a probe clip may be attached. BNC cables are plugged into a pair of binding posts using an adapter (Figure 1-29).**

Right: **photo showing how to construct a ×11 noninverting amplifier using jumper wires. Following the diagram in Figure 1-31 on page 1-35, the yellow wires connect a 10k resistor as R_f (to the op-amp output) and a 1k resistor as R_i (to ground). The red wires connect the amplifier input and output signals to the binding posts. Green wires connect the signal generator and oscilloscope grounds to the circuit ground, completing the circuit.**

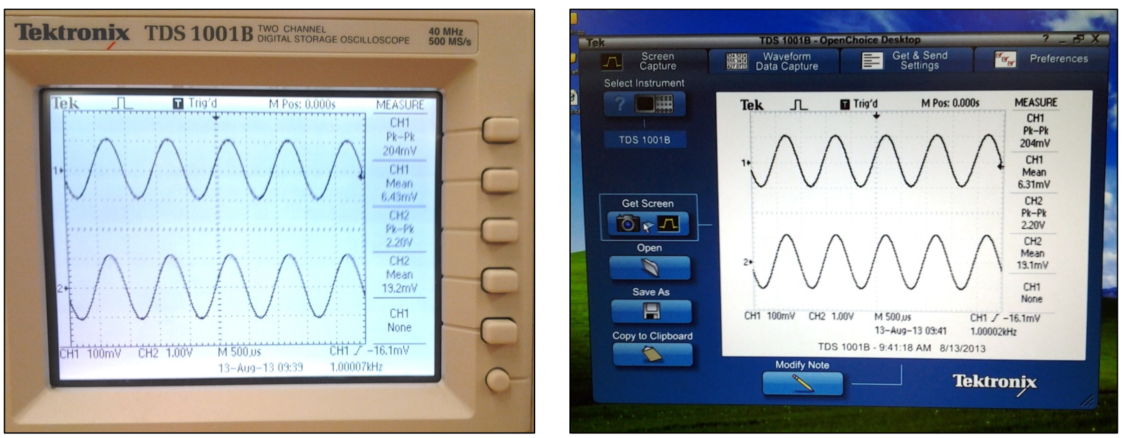


Figure 1-28: Waveform display using oscilloscope configured to measure input and output signal amplitudes and mean voltages, and a computer screen capture of similar data.

Considerations when making BNC cable connections

> ***The two conductors of the BNC cable are not equivalent***. For all the instruments in the lab the displayed voltage is that of the center pin with reference to the outer shell. In other words, *the BNC shell serves as the voltage reference for signals on the cable* (it is a *signal ground*).

Connections to external instrumentation (primarily the signal generator, oscilloscope, and the computer data acquisition system (DAQ) are made using coaxial cables with BNC connectors or, in the case of the oscilloscope, *10× probes*. The *BNC-banana interface adapter*, shown in Figure 1-29, lets you use a BNC cable with the breadboard assembly's 5-way binding posts, as shown in Figure 1-27. A BNC cable contains two conductors: an inner signal wire separated by an insulating sleeve from a surrounding braided shield. The outer metal shell of the BNC connector is connected to the cable shield, whereas the connector center pin goes to the cable's inner signal wire. The adapter's banana plug identified with the "GND" tab (as shown in the photo) connects to the BNC shell, the other banana plug (*opposite the "GND" tab*) goes to the BNC signal pin.

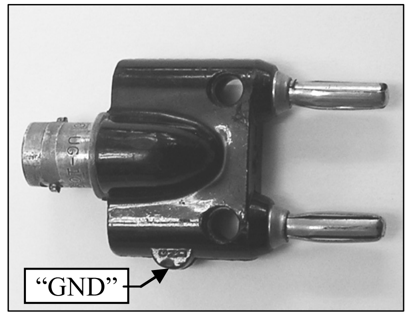


Figure 1-29: BNC adapter showing the tab identifying the connection to the BNC shell.

The DAQ analog voltage input BNC connectors are isolated from the other connectors and from *Earth Ground*, so their two connections may be made anywhere on the breadboard to make a measurement. This is not the case for the oscilloscope inputs! *Both oscilloscope input BNC shells are connected to each other and to Earth Ground*. This is also the case for the other BNC connectors on the computer DAQ interface.

The two oscilloscope inputs have BNC shells connected to **Earth Ground**. Always connect these conductors (and the ground clips on the 10× probes) to the analog breadboard **Ground** as shown in the photos of the example circuit in Figure 1-27.

CAUTION

Never apply a signal to a circuit that is not powered up. Otherwise the signal can damage the op-amps. When assembling a circuit or making changes to it, disable the signal generator output (using its OUTPUT button).

You may also want to turn off the power switch on the breadboard, especially if you are extensively rewiring a circuit.

The signal generator can supply up to 100mA to a circuit, which can cause some damage! The power supply terminals on the breadboard can source up to 250mA (at ±12V), which can make quite a spark!

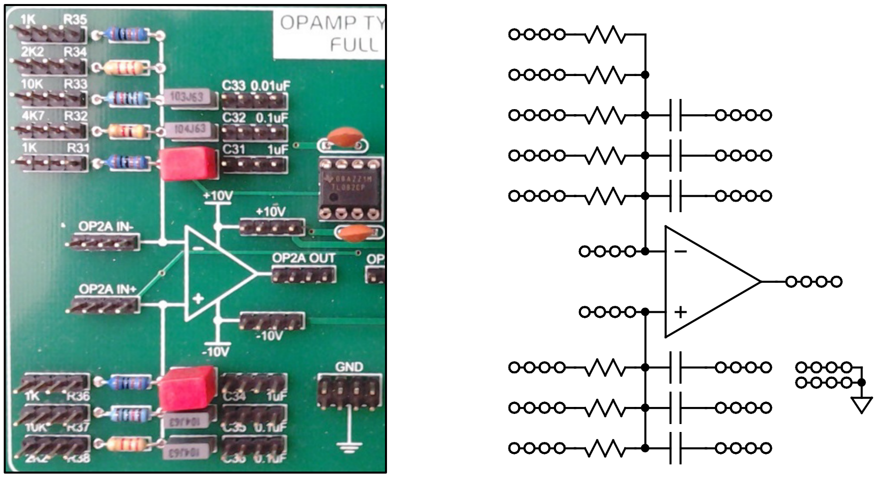


Figure 1-30: An op-amp in the trainer breadboard area with several resistors and capacitors already attached to the op-amp's + and – inputs, as shown in the equivalent schematic on the right. Terminal pins allow you to attach jumper wires to connect the free end of a component somewhere else. Pins are also provided to make direct connections to the op-amp inputs, its outputs, and to ground. The op-amp power supply terminals are already connected, so you don't need to connect to them.

Detailed procedures

Carefully examine Figure 1-30 and note that several components (a bunch of resistors and capacitors) are installed on the breadboard along with each op-amp, ***and each component already has one terminal connected to an op-amp input***. The other terminal of each component has a set of pins to which you may connect jumper wires. There are also pins providing access to the circuit ground, as shown. Only connect a jumper wire to a component you wish to use; the others will then have no effect on your circuit.

Inverting and noninverting amplifiers

Assemble a ×11 noninverting amplifier circuit, including the connections to the signal generator and oscilloscope. The left-hand diagram in Figure 1-31 (on page 1-35) shows the jumper connections required for this circuit, and it is also shown in the photos in Figure 1-27.

Investigate the behavior by starting with a 1kHz sine input with an amplitude of about 100mV. Determine the circuit gain by comparing oscilloscope measurements of the peak-to-peak input and output amplitudes (Figure 1-28). Now try different signal generator waveforms, amplitudes, and frequencies. Use the signal generator's offset function to add a constant (DC) voltage, and determine the DC gain by comparing measurements of the input and output mean voltages. Does it match the gain determined from the peak-peak voltage measurements? What happens to the output if the input amplitude is too large? Take a screen shot of this.

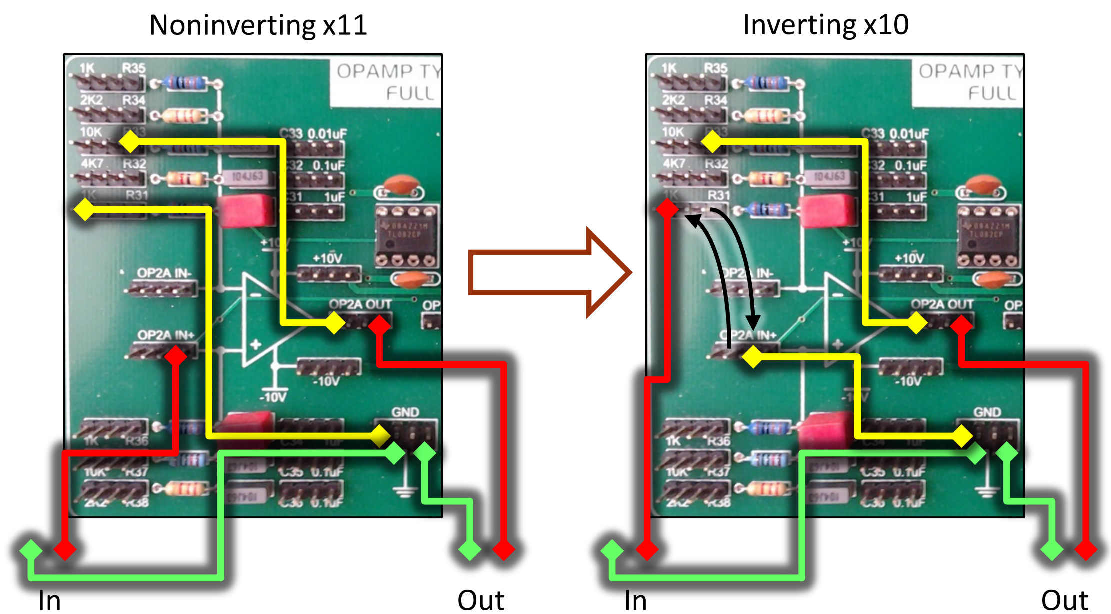


Figure 1-31: The left diagram shows the jumper connections for the first circuit you should build, a noninverting amplifier. The photos in Figure 1-27 show this circuit as well. The 10k resistor is connected to the op-amp output, making it the feedback resistor, R_f. A 1k resistor is connected to ground, making it R_i, and the input signal is connected directly to the op-amp +Input.

To convert to an inverting amplifer configuration, shown at right, simply swap the signal input and ground jumpers at their connections to the op-amp and R_i!

Next, reconfigure the input connection to the circuit to convert it into an inverting amplifier, as shown in Figure 1-31. What should be the gain for this amplifier? Confirm the operation of the amplifier in this configuration. Now try one or two different resistor combinations to get various gains, both in noninverting and inverting amplifier configurations. Fill out a table similar to this to summarize your results:

R_f	R_i	*Noninverting Gain*	*Inverting Gain*

Variable-gain amplifier

Construct the circuit presented in Figure 1-25 on page 1-29 (Prelab exercise 6). Use the 20k potentiometer included with your parts kit. The potentiometer may be installed in the breadboard area of the circuit trainer — your TA or the laboratory instructor can show you how to properly connect it. How does the circuit gain vary as you adjust the wiper position? Does it behave as you predicted in your answers to the prelab problem?

Additional circuits

Try *cascading* two amplifiers [output of one goes to the input of another, like circuit (e) in Prelab exercise 4]. Finally, if you have time, build and evaluate at least one of the circuits (your choice) from the **MORE CIRCUIT IDEAS** section.

What your experiment write-up should include

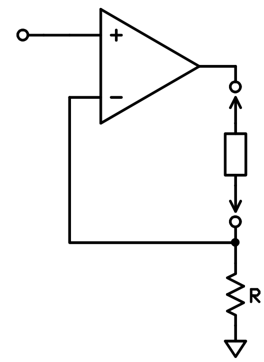
In addition to the specific requirements or questions to answer mentioned in the above procedures, your results should always include:

1. A schematic of each circuit you present results for showing all component values and how the signal generator and oscilloscope are connected to it.
2. What the expected performance of the circuit is predicted to be (gain, etc.).
3. Actual circuit performance, including interesting oscilloscope screen captures.
4. Comments about unexpected circuit behaviors and any useful "lessons learned" you ought to remember.
5. Any interesting ways you've discovered to configure the instruments to generate useful circuit inputs or make measurements.

> Don't spend a lot of time on your write-up after you leave the lab — just make it clear that you understand what you did. Spend your time outside lab studying the notes and preparing for the next experiment!

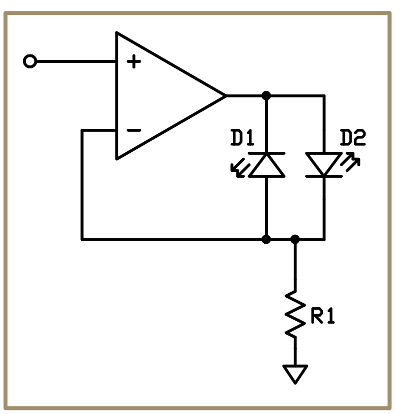
MORE CIRCUIT IDEAS

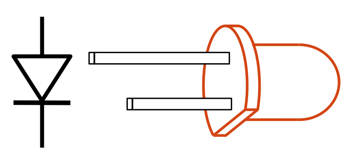
Voltage to current (transconductance) amplifier



Sometimes you would like to accurately set the current through a load by using a control voltage. Since a resistor is a "voltage to current converter" (Ohms' law, right?), we just need to apply the control voltage across a resistor and direct the resulting current through the load. A simple *transconductance amplifier*, shown at right, performs this trick. Essentially a noninverting amplifier configuration, the negative feedback ensures that the input voltage V_{in} will also appear at the *–Input* and thus across resistor R1. The current to establish this voltage drop comes from the op-amp output by feedback current passing through the load (shown connecting the output terminals). Thus the load current is given by: $I_{out} = V_{in} / R_{R1}$; the ratio of the output current to the input voltage (the amplifier gain) is $1/R_{R1}$. The gain has units of conductance (resistance^{-1}), which is called a *transconductance* because it is the ratio of a current and a voltage measured at two different places (*trans*: "transfer"). Note that no current is required from the source of the control voltage V_{in}; all load current comes from the op-amp output.

A useful application of this circuit is to illuminate a light emitting diode (LED) with a current proportional to the input voltage (see Figure 1-32 below). Since the output intensity of a LED is very nearly proportional to its current, we have a voltage to light intensity converter. LEDs require a minimum of about 1.8V to produce any appreciable light output, so using this amplifier can really simplify things (we'll learn more about diodes, including LEDs, in a later experiment). If R1 is a 1k resistor, then the LEDs will get 1mA of current for every 1V input, a useful conversion factor for many applications.



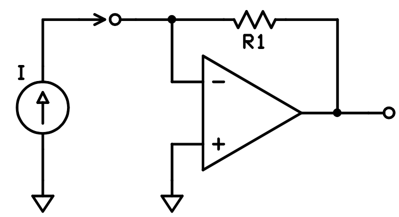


Determining diode polarity: the long lead is the anode (+); the short lead (nearest the flat section of the LED circumference) is the cathode (−).

Figure 1-32: A LED driver using a transconductance amplifier. LED *D1* is illuminated for $V_{in} < 0$, *D2* whenever $V_{in} > 0$. Resistor *R1* sets the conversion from input voltage to LED intensity.

One major drawback of this simple circuit is that the load must be an isolated, 2-terminal device (neither output terminal is a circuit ground, so the load must be able to *float*). We'll investigate transconductance amplifier circuits in a later experiment which relax this requirement.

Current to voltage (transimpedance) amplifier



Sometimes the input signal to be amplified and measured is a current rather than a voltage. One popular way to accomplish this would be to let the current flow through a small resistor of known value and then to amplify the voltage drop across it. A simple *transimpedance amplifier*, shown at right, is often a very useful circuit for accomplishing this result: simply an inverting amplifier with no input resistor! Current entering the transimpedance amplifier's input terminal (which is, of course, a virtual ground) flows on through the feedback resistor R1. The voltage drop across this resistor appears at the op-amp output, so the gain is: $V_{out}/I_{in} = -R_{\mathrm{R1}}$. The gain has units of impedance (resistance), which is called a *transimpedance* (or *transresistance*). Because the input is a virtual ground, the voltage the current source must be able to supply at the amplifier input in order to maintain its current is tiny (this is called the required voltage *compliance* of the amplifier).

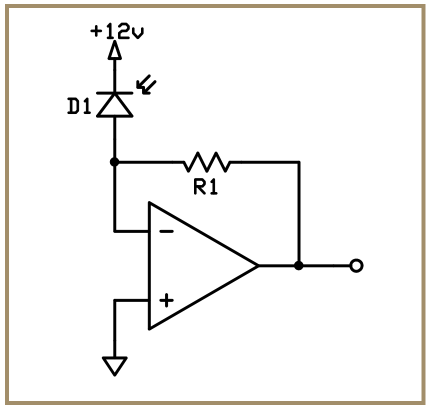
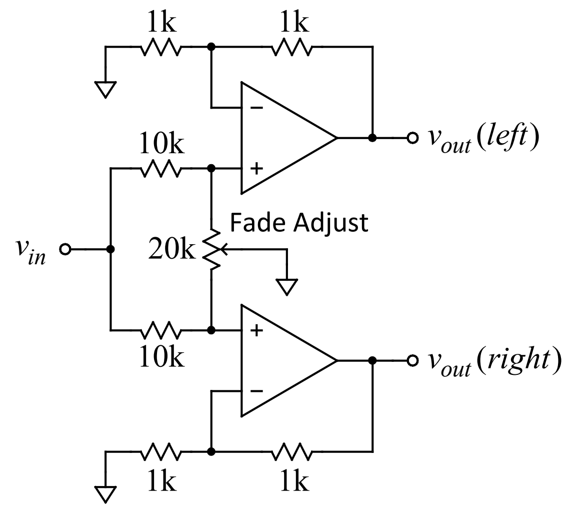


Figure 1-33: A photodiode amplifier. The output is proportional to the light intensity; *R1* sets the gain.

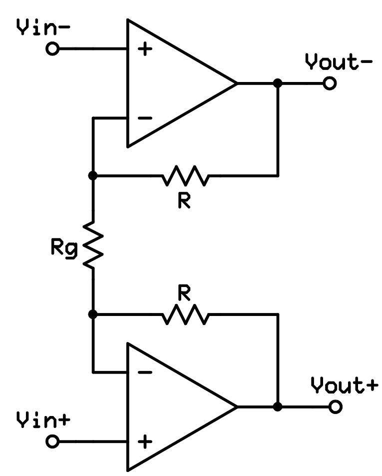
Figure 1-33 is an example of the usefulness of this amplifier. A *reverse-biased photodiode* is connected between a power supply and the amplifier input as shown. Connected as shown, only a fraction of a microamp of current will flow through the diode when it is not illuminated (the diode's *dark current*). When illuminated the diode's current will be much larger — 10's to 100's of microamps, and since its current is very nearly proportional to the level of illumination, so will be the amplifier's output voltage (the reverse of the circuit in the previous section). Select the value of resistor R1 so that the output voltage range is appropriate for the light intensities you expect to experience, and/or add another amplifier stage to the circuit's output.

Audio fader control



Consider the schematic at right, which demonstrates another interesting way to use a potentiometer. When the *fade adjust* wiper is in its center position, $v_{in}/2$ is presented to each opamp's *+Input*, which is configured as a ×2 noninverting amplifier. Thus both the left and right outputs equal v_{in}. As the wiper position is varied the input signal is distributed unequally to the two amplifiers, so that it may be smoothly panned from one output channel to the other. Note that with the resistor values shown, the total output power (which is proportional to the sum of the squares of the two output signals) stays approximately constant. If two different signals are applied to the pair of 10k resistors (rather than the single input as shown in the figure) then the circuit becomes a stereo balance control.

Instrumentation amplifier



Consider the circuit shown at right. How do we analyze its response to the two inputs? The answer is, naturally, to use linear superposition, but we're going to apply it in a clever way. Instead of setting V_{in+} or V_{in-} individually to zero, consider instead two combinations of them: their difference, $V_{diff} = V_{in+} - V_{in-}$, and their average, $V_{cm} = (V_{in+} + V_{in-})/2$. Then if we set $V_{diff} = 0$, we would have $V_{in+} = V_{in-} = V_{cm}$; if instead $V_{cm} = 0$, then $V_{in+} = -V_{in-}$. Now use linear superposition of these two new independent variables, V_{diff} and V_{cm}, to determine V_{out+} and V_{out-}, assuming that the two resistors labeled R are perfectly matched, and the resistor $R_g \neq 0$.

Consider the case $V_{diff} = 0$ first, so $V_{in+} = V_{in-}$. Assuming that the negative feedback does what it's supposed to, then the two op-amps' *–Inputs* are also equal, which means that the voltage drop across R_g vanishes, and thus so does the current through R_g. Since no current flows through it, resistor R_g could be removed from the circuit without affecting the circuit's operation! We're left with two independent voltage followers (the value of R doesn't matter), and $V_{out+} = V_{out-} = V_{cm}$. The circuit has a gain of 1 for a *common mode input*, V_{cm}: $G_{cm} = 1$.

Next set $V_{cm} = 0$, so $V_{in+} = -V_{in-}$ and ditto for the two op-amps' *–Inputs*. The resistor R_g has its two terminals with equal and opposite voltages, so *the center of this resistor is at* *ground potential*. We can split the resistor into two resistors in series, each with value $R_g/2$, and we know that their junction is at ground potential. This symmetry again lets us separate the circuit into two twins, but this time each half is a noninverting amplifier with $R_f = R$ and $R_i = R_g/2$, so each amplifier has a gain of $1+2R/R_g$. So:

$$V_{out+} - V_{out-} = \left(1+2R/R_g\right)\left(V_{in+} - V_{in-}\right) = \left(1+2R/R_g\right)V_{diff}$$

$$\boxed{G_{diff} = 1+2R/R_g}$$

We have a circuit with a high gain for a *differential input signal*, but a gain of 1 for a *common mode signal*. The differential gain, G_{diff}, can be adjusted by changing the value of only one resistor, R_g. Because each amplifier has a noninverting configuration, its *input resistance is very large*.

This clever circuit is meant to be combined with a traditional differential amplifier (Figure 1-23) to create the 3 op-amp *instrumentation amplifier*:

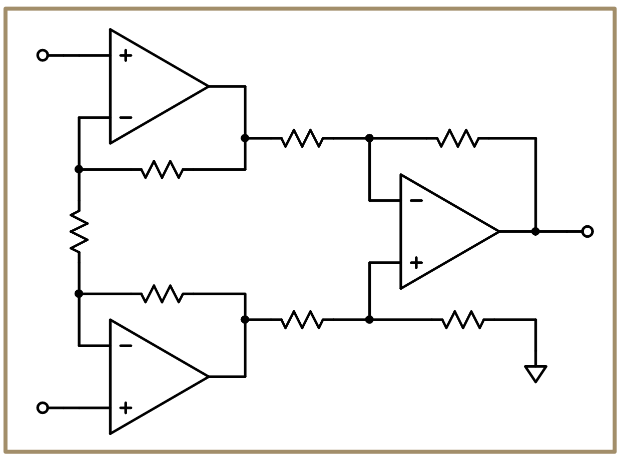


Figure 1-34: The *Instrumentation Amplifier*. This enhancement of the basic differential amplifier has a very high input impedance for both the + and – inputs, has a differential gain which can be changed by varying the value of one resistor, and has very high *common mode rejection*. It is so useful, especially for scientific applications, that many versions are available in the form of single integrated circuit devices.

ADDITIONAL INFORMATION ABOUT THE CIRCUITS

This section expands on some of the material presented earlier. It is better skipped during a first reading, but you may want to go over it after you thoroughly understand the concepts discussed in the first several sections.

Other sorts of circuit grounds

Often in commercial electronic equipment (but not always) the circuitry's 0-Volt reference point will be physically connected to the ground beneath the building using the "ground connection" in the device's 3-pin AC power-line cable; this reference point thus called *earth ground* and is said to be at *earth ground potential*.

If it is important to distinguish between a circuit's 0-Volt reference potential and earth ground (because the two may not be actually physically connected), the circuit reference point is then called *local ground* (or just *ground*), and *earth ground* would be, in general, a different reference point with a different potential. Other common reference points include *chassis ground* (the potential of the equipment's metal enclosure), *analog ground* and *digital ground* (if these different sections of a circuit do not share a common reference point), *signal ground* (the bottom terminal of an input signal source's circuit element), etc. The symbols used in this text for various grounds are shown in Figure 1-35.

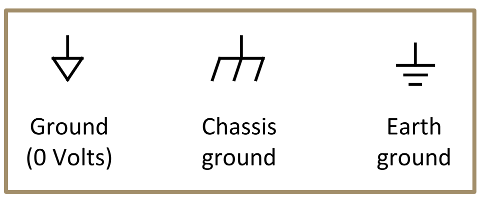


Figure 1-35: Ground symbols we might use. Usually, only will represent the 0-Volt reference point.

Incremental Input Resistance

If there are other sources providing inputs to a network, then at an input port in question it may no longer be the case that v_{in} and i_{in} are proportional; it may be, for example, that $i_{in} \neq 0$ even though $v_{in} = 0$. Consequently, we probably don't want to use equation 1.12 to calculate the port's input resistance. Consider, for example, the differential amplifier in Figure 1-23 (on page 1-25): the op-amp's *–Input* node *will not be a virtual ground* because the op-amp's *+Input* voltage depends on the source voltage v_{in+} (and the op-amp *–Input* will be at the same voltage as the *+Input*); therefore even if $v_{in-} = 0$, the current i_{in-} at that port will not vanish because of the nonzero voltage drop across the input resistor. To avoid this sort of problem with our definition of the input resistance, we instead define the circuit's *incremental* (or *dynamic*) *input resistance*, r_{in}, *as a derivative* (equation 1.13).

Incremental Input Resistance

1.13
$$\boxed{r_{in} \equiv \frac{\partial v_{in}}{\partial i_{in}}}$$

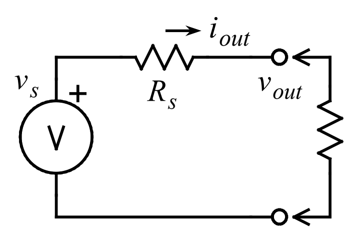
We use a *partial* derivative in (1.13), meaning that all other *independent input sources* are held constant. For linear circuits, the incremental input resistance r_{in} will not vary with the magnitudes of v_{in} and i_{in}, even in the presence of other input sources. The incremental input resistance r_{in} also will not be affected by the amplitudes of the other input sources. The concept of an incremental resistance, however, is useful even for nonlinear circuits, although in this case you should expect that its value will vary with voltage and/or current.

CALCULATING A PORT'S INCREMENTAL INPUT RESISTANCE

In a linear circuit the incremental input resistance r_{in} of any individual input port is not affected by the magnitudes of any of the other independent input sources. Thus you can calculate its value by first setting all other input sources to 0 (using the rules on page 1-20) and then determining the ratio v_{in}/i_{in} for the single input port in question. Make sure all other ports of the network (sub-circuit) are first *terminated* (connected to whatever circuits to which they will interface).

Output resistance

The *output resistance* of a power source or a network's output port characterizes how its output varies with changing load resistance. For example, consider a power source which is made up of an ideal voltage source, v_s, in series with a nonzero source resistance, R_s, driving some load resistor as shown at right. The current through the load, i_{out}, also must flow through the source resistance, and the voltage drop across this resistance is then $R_s i_{out}$. Thus, v_{out}, the output voltage, is less than v_s, the voltage it would be if the load resistance were infinite ($i_{out} = 0$).



$v_{out} = v_s - R_s i_{out}$

We define the output resistance of an output port in terms of the drop in the output voltage with increasing output current:

Output Resistance

1.14
$$\boxed{r_{out} \equiv -\frac{\partial v_{out}}{\partial i_{out}}}$$

We use a partial derivative in this expression to reflect the fact that the driving source v_s *must be held constant* as the output current is varied to determine r_{out}. An *ideal voltage source* is one whose output voltage is unaffected by the load current it must supply, so its output resistance is 0. An *ideal current source*, on the other hand, supplies a constant current regardless of the voltage required to push that current into its load, so $\Delta i_{out} \equiv 0$. Thus for a current source $r_{out} = \infty$.

Our old friend, the voltage divider, however, is not such an ideal character. Let's calculate r_{out} for the output of a voltage divider driven by an ideal voltage source at its input. We will solve this problem by using linear superposition, our powerful circuit analysis ally.

VOLTAGE DIVIDER OUTPUT RESISTANCE

The problem is to determine how the output voltage changes as we change the output current required by a load attached to a voltage divider. To do this using linear superposition, we perform a sort of "thought experiment": we replace the output load with a *current sink*, an ideal current source we can control to independently set the output current to mimic any load.

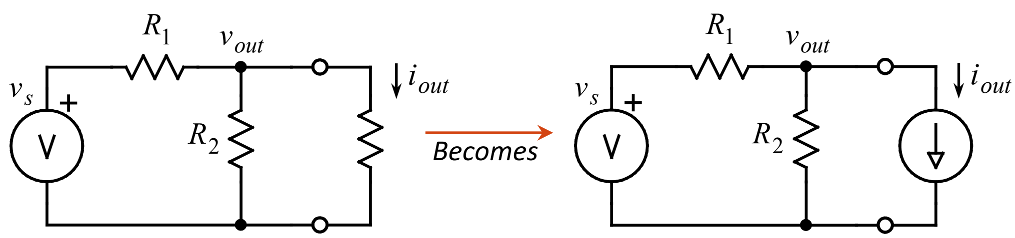


Figure 1-36: Determining the output resistance of a voltage divider driven by an ideal voltage source. Think of the load as a *current sink* (right schematic) and then set the source to 0; this puts the two resistors in parallel, and that parallel combination gives the output resistance, r_{out}.

With this change (Figure 1-36), we know that v_{out} will be a function of v_s and i_{out} with the form given by equation 1.8: $v_{out} = a\,v_s + b\,i_{out}$ for some constants a and b determined by the resistor values R_1 and R_2. This means that $r_{out} = -\partial v_{out} / \partial i_{out} = -b$, so we just need to determine the value of b. But we've already solved this problem: see the simple example of superposition presented starting on page 1-20, where we found that the coefficient b of i_{out} is given by the parallel combination of resistors R_1 and R_2:

$$r_{out} = R_1 R_2 / (R_1 + R_2)$$

Note that this calculation is operationally equivalent to replacing the embedded voltage source with a short circuit (setting it to 0) and then treating the output port as though it were an input and calculating its input resistance. This procedure is generally the correct way to calculate a linear circuit's output resistance (or, more generally, its output impedance). That this method works is yet another example of the principle of linear superposition. As with the determination of the input resistance of a network or sub-circuit, the answer will generally depend on how all its other ports are *terminated* (what sorts of circuits are attached to them).

CALCULATING A PORT'S OUTPUT RESISTANCE

To determine the output resistance of a network port, connect all other ports to whatever circuits they will interface to. Then set all independent driving voltage and current sources to 0, replacing them as described on page 1-20. Finally, treat the port in question as an input, and calculate its input resistance — this result will equal r_{out} (equation 1.14).

Thevenin and Norton models of power sources or outputs

We can model most real power sources or circuit output ports as ideal voltage or current sources combined with a finite resistance (or *impedance*, as we'll use in Experiment 2) so that their output resistance matches that of the real source, as shown in Figure 1-37. These are called *Thevenin* and *Norton* equivalent circuits (or *models*) of a real source. For example, many commercial signal generators (including the one you will use in lab) have a 50 Ohm output resistance and are best represented using a Thevenin model (voltage source in series with $R_s = 50\Omega$).

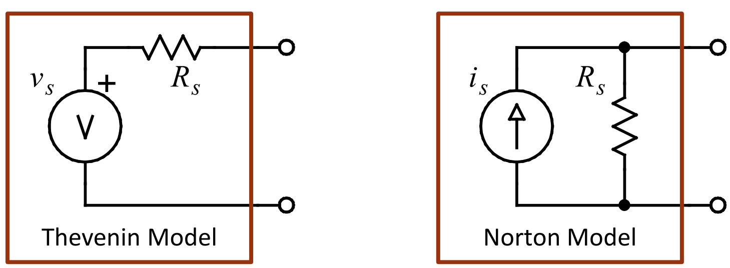
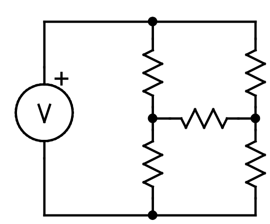


Figure 1-37: Thevenin and Norton models of power sources with finite output resistance. Which model you choose depends on your application, but if the output resistance R_s is relatively small, then the Thevenin (voltage source) model is probably the correct choice. Use the Norton (current source) model if R_s is large. The two models are completely equivalent (have the same output regardless of load) if the source resistance R_s is the same for both and the two source amplitudes are related by $v_s = R_s i_s$ (see also Prelab Exercise 5 on page 1-28). The laboratory signal generator has R_s = 50 Ohms.

A nontrivial example of circuit analysis using Kirchhoff's laws



Consider the *bridge circuit* of five resistors shown here, driven by a voltage source. We want to determine two things: the total equivalent resistance of the circuit, and the current through the center, horizontal resistor, assuming that the values of the various resistors are all different (the center resistor is called the "bridge resistor" because it "bridges" the outputs of two voltage dividers). You will quickly realize that this is a nontrivial problem; we can't use our series or parallel resistor formulas to simplify our analysis, so we will have to employ a more brute-force method.

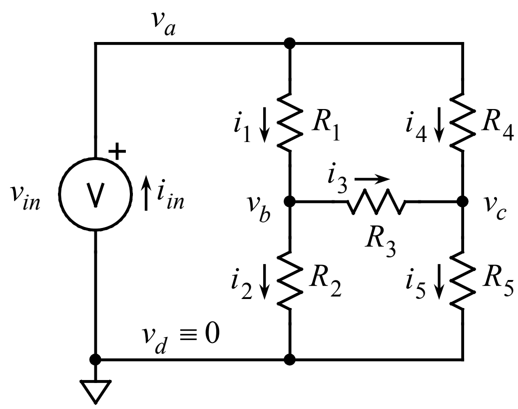


Figure 1-38: A nontrivial example to exercise the circuit voltage and current rules: a resistor "bridge" with voltages and currents identified and labeled for analysis. The bottom node is defined to be ground, so the voltage there is 0 by definition. The source is assumed to supply voltage v_{in}, and the circuit draws current i_{in} from it.

We start by labeling the currents and voltages at each node (there are 4 nodes in all). Figure 1-38 gives a possible labeling which we'll use for this example. Notice that we have already used the rule concerning the total current flow into a lumped element's terminals: since each element has two terminals, we know that the various currents may be represented by a single current flowing through each of the elements. We have also chosen the bottom node to be *ground*, so the voltage there is 0, by definition (since there are no hidden or implied power supplies or other elements connected to ground for this example, we know that there are no other connections carrying current into or away from our ground node, and our list of currents is complete).

The example problem has one independent (driving) variable, v_{in}, and five fixed parameters, the resistor values R_1 through R_5. There are three unknown voltages, v_a through v_c, and six unknown currents, i_{in} and i_1 through i_5. So we need 9 independent equations to solve for the unknowns. Start with the loop voltage rule and use Ohm's law to relate the node voltages to the resistor currents:

(1) Clearly, by going up the left side through the voltage source: $v_a = v_d + v_{in} = v_{in}$
(2) Following the current through R_1, the voltage will drop across it: $v_b = v_a - R_1 i_1$
(3) Similarly for R_4: $v_c = v_a - R_4 i_4$
(4) The voltage drop across R_3: $v_c = v_b - R_3 i_3$
(5) The voltage drop across R_2 takes us back to ground: $v_d = 0 = v_b - R_2 i_2$
(6) Ditto for R_5: $v_d = 0 = v_c - R_5 i_5$

The node current rule gives:

(7) at node *a*: $i_1 + i_4 = i_{in}$
(8) at node *b*: $i_2 + i_3 = i_1$
(9) at node *c*: $i_3 + i_4 = i_5$
(10) at node *d* (ground): $i_2 + i_5 = i_{in}$

Now we have 10 equations for our 9 unknowns, but, of course, they are not all independent. As you might expect, the problem is with the node current equations; in fact the combination of equations $(9)+(10)-(8)$ yields equation (7), so we can discard one of these four equations and the remaining nine will form a complete, independent set. This will often be the case for the node current equations of a circuit.

We can solve this linear system of equations in any of the standard ways; this is the sort of problem computers were originally designed to solve. The solution is messy, and the answers to the original problem turn out to be:

The total equivalent resistance presented to the source:

$$R \equiv \frac{v_{in}}{i_{in}} = \frac{\left(R_1 R_2 R_4 + R_1 R_2 R_5 + R_1 R_4 R_5 + R_2 R_4 R_5\right) + R_3\left(R_1 + R_2\right)\left(R_4 + R_5\right)}{\left(R_1 + R_4\right)\left(R_2 + R_5\right) + R_3\left(R_1 + R_2 + R_4 + R_5\right)}$$

The current through the bridge resistor, R3:

1.15
$$\frac{i_3}{v_{in}} = \frac{R_2 R_4 - R_1 R_5}{\left(R_1 R_2 R_4 + R_1 R_2 R_5 + R_1 R_4 R_5 + R_2 R_4 R_5\right) + R_3\left(R_1 + R_2\right)\left(R_4 + R_5\right)}$$

The bridge circuit is said to be *balanced* when the current through the bridge resistor, R_3, vanishes. We see that the requirement for balance is $R_2 R_4 = R_1 R_5$.