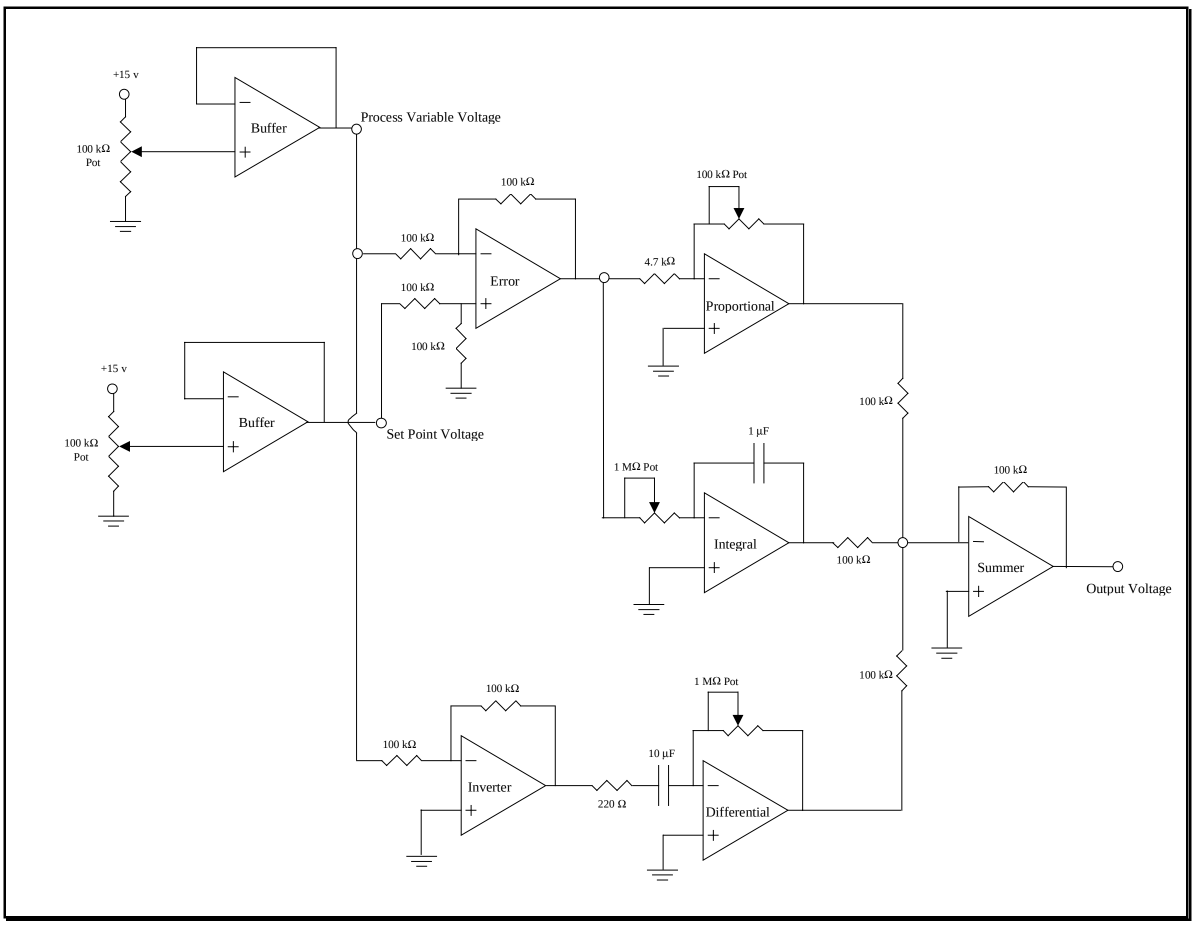
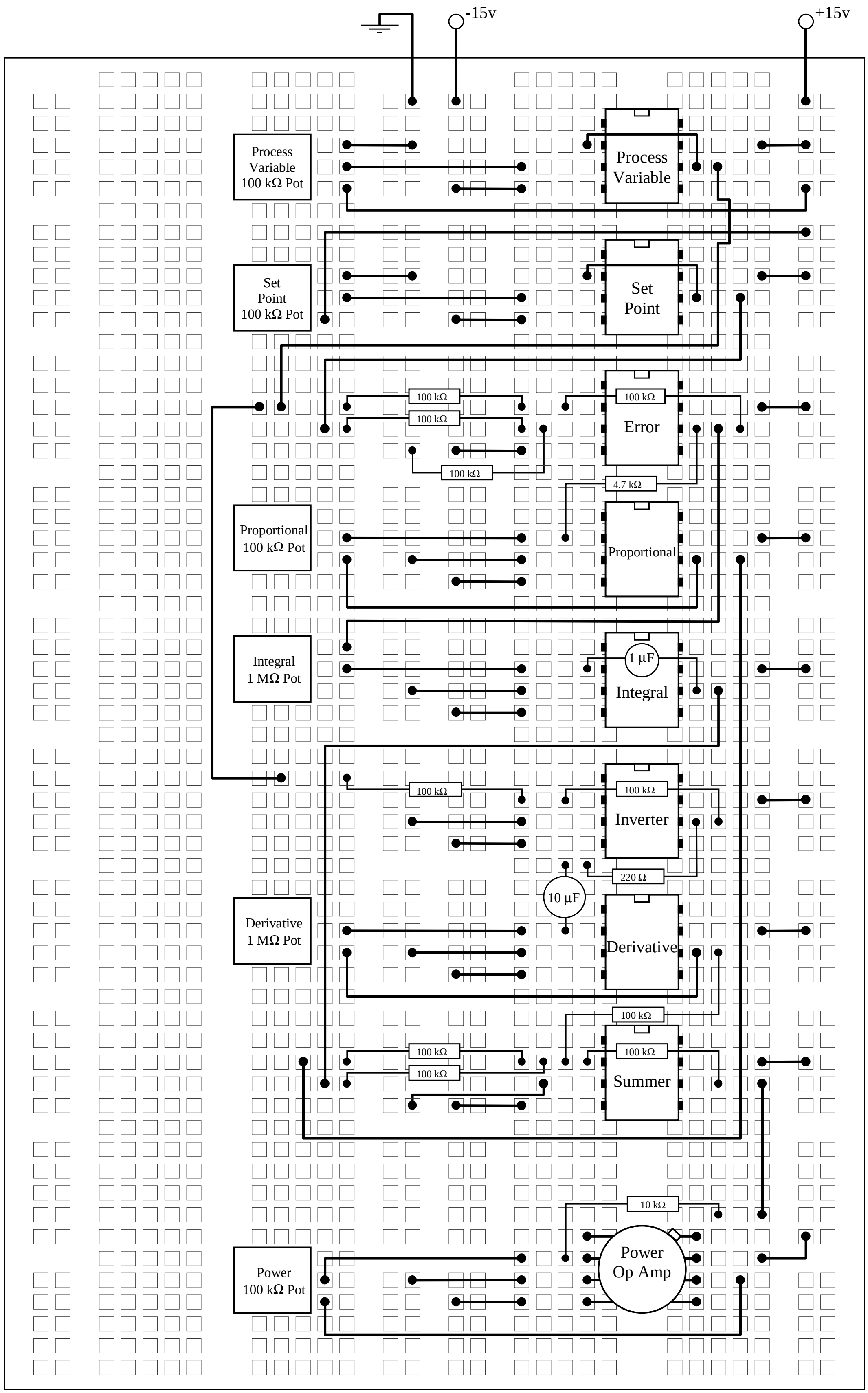
Basic Experiments in PID Control for Non-electrical Engineers



J.P. Thrower, S. Kiefer, K. Kelmer, and L. Silverberg

May 1998



-15v
+15v
Process
Variable
100 kΩ Pot
Process
Variable
Set
Point
100 kΩ Pot
Set
Point
100 kΩ
100 kΩ
100 kΩ
100 kΩ
Error
4.7 kΩ
Proportional
100 kΩ Pot
Proportional
Integral
1 MΩ Pot
1 µF
Integral
100 kΩ
100 kΩ
Inverter
220 Ω
10 µF
Derivative
1 MΩ Pot
Derivative
100 kΩ
100 kΩ
100 kΩ
100 kΩ
Summer
10 kΩ
Power
100 kΩ Pot
Power
Op Amp

Contents

Introduction

As you read this little book and follow its directions you will be guided to an understanding of proportional-integral-derivative (PID) control. However, the purpose of this manual, and the value of learning PID control goes beyond PID control itself.

Assuming that you are not an electrical engineering student, it is possible that you lack a basic understanding of electrical engineering. You may ask yourself, what is it that electrical engineers do? Of course, a lack of understanding of basic principles in electrical engineering limits your ability to understand most mechanical devices--since most are really "electromechanical." This lack of understanding limits your imagination and prevents you from dreaming up and designing new electromechanical devices to solve whatever problem you face--whether the device is to be sold on the open market, integrated into a manufacturing process, or simply used in a laboratory.

So, what are these basics that electrical engineers follow? The answer is really quite simple. Also, you might be surprised to find out that these basics have been followed for over a century and that they haven't changed. Certainly, the devices themselves have changed--they've become smaller, more reliable and more efficient - but not the basic principles.

The basic principles followed in electrical engineering give the electrical engineer the ability to manipulate electrical currents and voltages.

The electrical currents and voltages (signals) travel in *circuits* to energize *actuators* and *sensors*. Actuators are components that *produce* physical quantities (forces, displacements, heat, temperature, acoustic radiation, electrical radiation, etc..) when *subjected* to an appropriate electrical signal. On the other hand, sensors are devices that *produce* an appropriate electrical signal when *subjected* to a physical quantity. Indeed, sensors and actuators are used for opposite purposes. The manipulation of these actuators and sensors is through circuit design - of which there are two types: *analog* design and *digital* design.

Analog design deals with electrical signals that are *continuous* (in time). Since actuators and sensors are physical devices, and since physical quantities change continuously (in time), you'll find that analog circuits are used to manipulate actuators and sensors.

Digital design, on the other hand, deals with electrical signals that are *discontinuous* (in time). Since the process of making logical decisions requires asking whether certain statements are true or not, and since this is a discontinuous process, we find digital design used largely in the decision making parts of the device. In fact, if you look at a circuit board, you'll find the analog components situated close to the actuators and sensors, and the digital components, if any, located further away from the actuators and sensors.

The PID control system that you are about to make is a beautiful application of analog design. However, you'll need some basic principles of analog design before you build the PID control system. (Incidentally, learning about analog design will be more valuable to you in the long run than learning about PID control).

The *first layer of principles* in analog design addresses the components of the simple circuit and the analysis of the simple circuit. We'll deal with three basic components: the resistor, the capacitor, and the operational amplifier. The analysis of the simple circuit will be performed by looking at voltages around closed paths and by looking at the currents entering the nodes of the circuit. Building on these first principles, the *second layer of principles* is associated with basic operations that can be performed with simple circuits. We'll specifically look at buffering, and how to add, multiply, differentiate, and integrate signals. Building on these principles, we come to the *third layer of principles.* The third layer of principles is associated with the manipulation of simple circuits (operations) into *complex circuits.* Examples of complex circuits are filters, controllers, estimators, and identifiers. We'll restrict our attention here to the PID controller.

This little book covers the first and second principles, and then PID control.

Getting Ready

The readers of this little book are mechanical and aerospace engineering students, both undergraduates and graduates, and others who wish to learn about basic analog design and PID control. The belief followed here is that the best way to learn this material is not by watching a video or through classroom instruction, but by a hands-on experience that progress at your own pace.

So, to get ready, find a couple of hours of solitary time, and read over this little book before you do anything else. In fact, read it over several times! When you're ready, familiarize yourself with the components that you'll use. You will then be ready to build your PID control system.

If you take your time (and grab a Coke), you'll find the experience quite enjoyable.

1. Analog Components

The Resistor

Perhaps the most basic component in an electric circuit is the resistor. Resistors dissipate energy (which is converted into heat). A drawing of a typical resistor and the symbol that we use to represent a resistor in circuit diagrams are shown in Fig. 1.

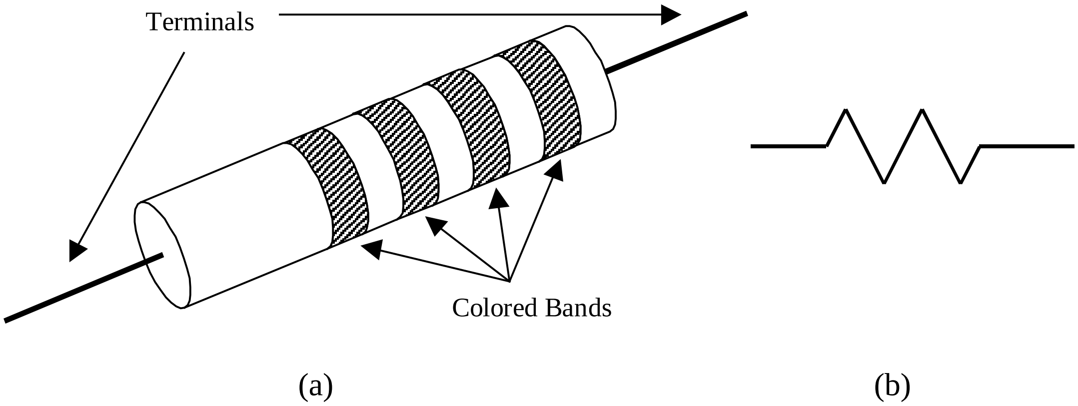


Figure 1: The resistor and the resistor symbol

The four colored bands around the resistor indicate the value and accuracy of the resistor's resistance R measured in ohms (Ω). To determine the value of resistance, orient the resistor with the gold or silver band to the right. The first two bands on the left determine the numerical value of the resistance. The third band is a base ten multiplication factor, and the fourth band gives the accuracy of the resistance value. The fourth band is either gold or silver. The values are interpreted using the following table.

Color	**Digit**	**Color**	**Digit**
Black	0	Green	5
Brown	1	Blue	6
Red	2	Violet	7
Orange	3	Gray	8
Yellow	4	White	9

Color	**Tolerance**
Gold	5%
Silver	10%
No Fourth Band	20%

For example:

Yellow	Violet	Black	Gold	=	$47 \times 10^0\ \Omega$	± 5%	= 47 ± 2.35 Ω
Yellow	Violet	Red	Gold	=	$47 \times 10^2\ \Omega$	± 5%	= 4700 ± 235 Ω
Brown	Black	Yellow	Silver	=	$10 \times 10^4\ \Omega$	± 10%	= 100k ± 10 kΩ

Quite frequently, the resistance in a circuit needs to be changed. A resistor that has a variable resistance is called a rheostat, a potentiometer, or simply a “pot”. The pot’s resistance is changed by turning a knob or screw slot in the pot. A typical pot has three terminals. The resistance between the outside terminals is the maximum (total) resistance of the pot (e.g. 10KΩ). The resistance between the middle terminal and either of the outside terminals varies as the knob is turned. Sketches of typical pots, the pot symbol, and the resistance between the terminals are shown in Fig. 2.

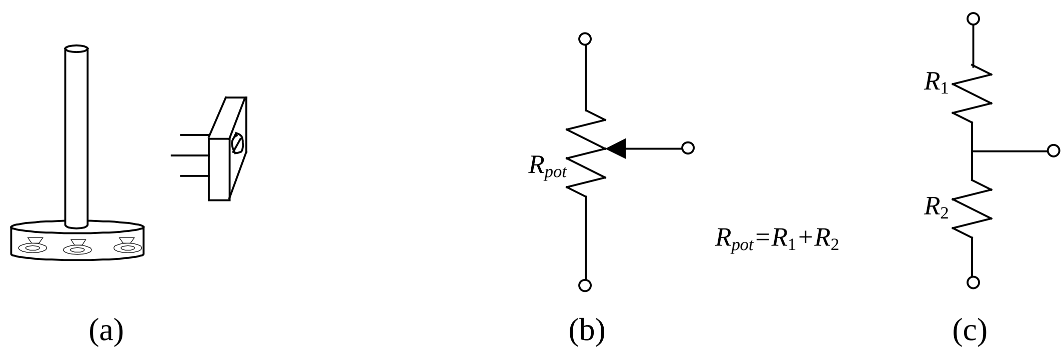


Figure 2: The potentiometer a) sketches, b) symbol, and c) equivalent circuit

In some pots, R_2 increases in linear proportion to the amount the knob is turned. This type of pot is called a *Linear* Pot. In other pots the log of R_2 increases in proportion to the amount the knob is turned. This type of pot is called an *Audio Taper* Pot. We will confine ourselves to linear pots.

When analyzing circuits we'll sometimes look at *voltage differences* and other times we'll look at *point voltages* at the nodes in a circuit. The voltage difference v across a resistor is proportional to the current i passing through it. This relationship is called Ohm's law

$$v = iR$$

where v is measured in volts (V), i is measured in Amperes (A), and R is measured in ohms (Ω).

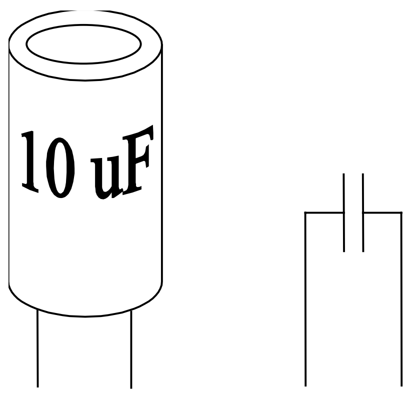
The Capacitor

The second component in a simple circuit is the capacitor. Capacitors are used to store electrical charge q. If you apply a voltage v across the two leads of a capacitor, a specific amount of charge will accumulate in the capacitor. The amount of charge is proportional to the voltage. The proportionality constant is called the *capacitance C*. In other words, $q = Cv$, where q is measured in Coulombs, C is measured in Farads, and again v is measured in volts (V).

Similar to resistors, there is a relationship between voltage and current across a capacitor. Since current is the time derivative of charge ($i = dq/dt$), it follows that

$$i = C\frac{dv}{dt}.$$

Some capacitors have polarity, which refers to the fact that the voltage difference across them can only be applied in one direction for them to function correctly. These are called electrolytic capacitors. We will not use electrolytic capacitors. Instead, we'll confine ourselves to bipolar capacitors, for which the voltage difference can be applied in either direction. A physical drawing and the capacitor symbol are shown in Fig. 3.



(a) (b)

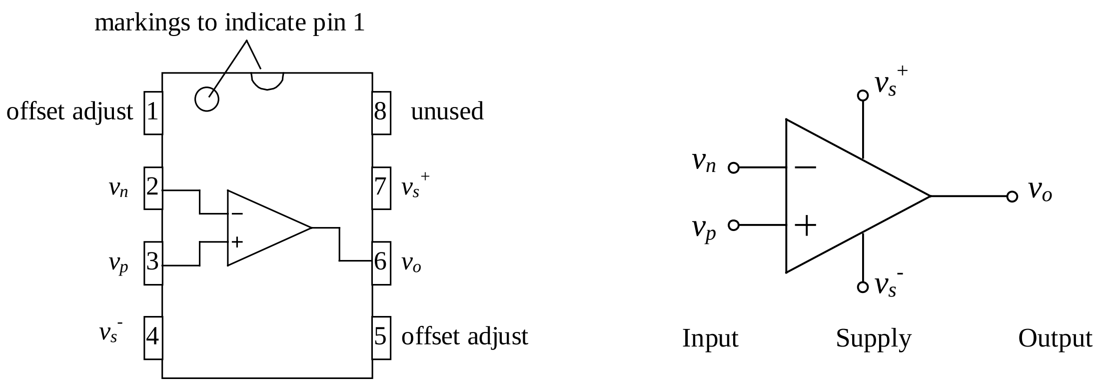
Figure 3: The bipolar capacitor and the capacitor symbol.

The Operational Amplifier

As the word amplifier suggests, the function of an operational amplifier (op amp) is to amplify a voltage. However, the operational amplifier does much more than that. It also functions as a buffer and as a cascader—which are two functions that enable simple circuits to be assembled into complex circuits to create higher level functions which are called *operations*—hence the name *operational amplifier*.

Op amps have five terminals that are important. The voltage that is amplified is the difference between the voltage at the '+' terminal v_p and the voltage at the '-' terminal v_n, as shown in Fig. 4. The amplified voltage is the output voltage v_o.

Unlike the resistor and capacitor, which are both "passive" (unpowered) devices, the op amp is an "active" device. Indeed, the op amp needs a voltage supply for the amplification. The v_s^+ and the v_s^- terminals are the positive and negative supply voltages, respectively. The op amp schematic and the chip that we'll use are shown in Fig. 4.



(a) The 741 op amp on a chip (b) The op amp symbol

Figure 4: The op amp schematic and chip configuration.

The way in which op amps buffer and cascade, and the methods in which they can be used to carry out operations (multiplication, addition, differentiation, integration, etc.) will be discussed later when they are analyzed.

Powering The Breadboard

A breadboard is a tool that enables you to assemble a circuit without soldering. The breadboard allows you to connect components together to form a circuit by plugging component terminals and jumper wires into holes. For simplicity, certain rows and columns of holes are already electrically connected, as shown in Fig. 5.

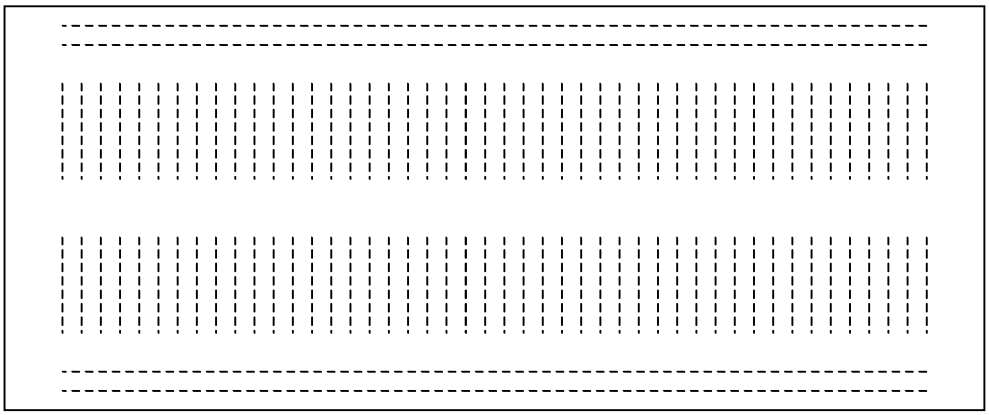


Figure 5: The connection diagram of a breadboard (dashed lines connect rows and columns of holes).

The breadboard can be powered one of two ways. If you are working at home, the first step is to connect two 15 volt DC converters (which can be purchased at Radio Shack) as shown below in Fig. 6.

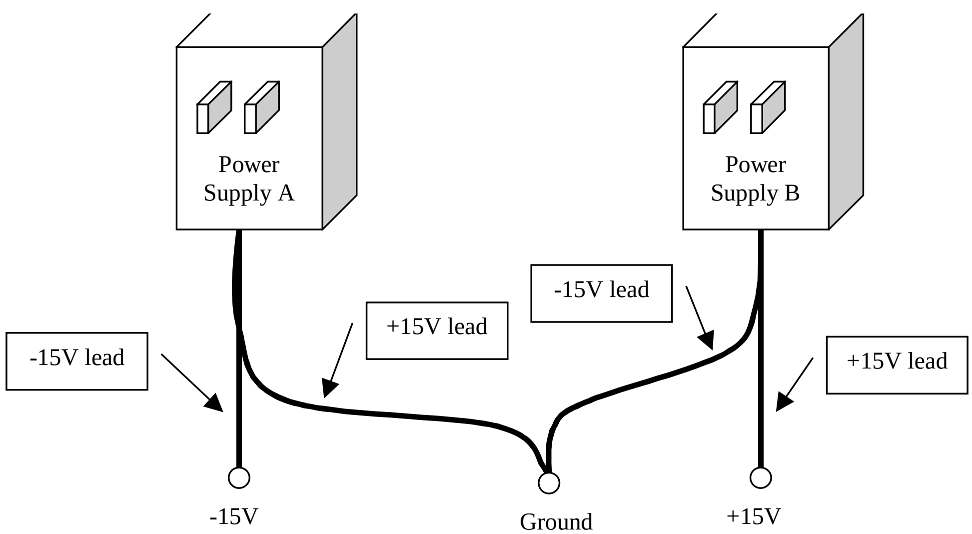


Figure 6: Configuration for home power supplies

The other possibility is to power the board using the power supply provided to you in a lab. Once you have available voltage (either from the DC converters or from a lab power supply), connect the terminals to the screw connectors, as shown in Fig. 7. Then use jumper wires to power the vertical rows of the breadboard, as shown. The vertical columns are connected the entire length of the board, so you will now be able to get +15 volts, 0 volts, and -15 volts anywhere along the respective columns (also see Fig. 5).

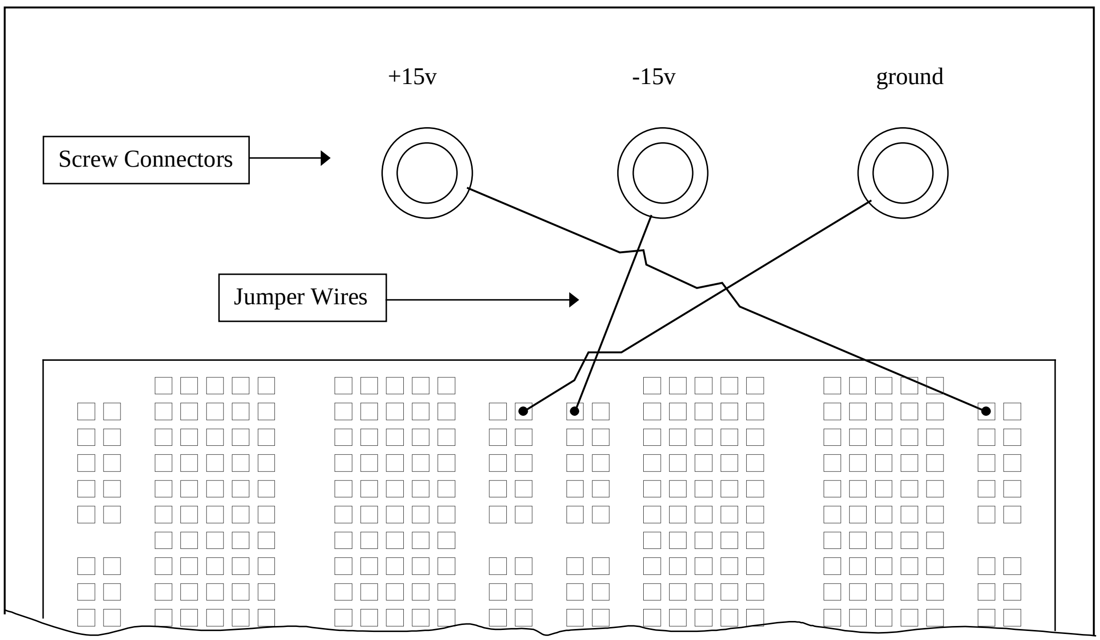


Figure 7: Powering the breadboard and row block connection

WARNING: Do not leave the power connected to the breadboard while you are building a circuit. Always disconnect the power when installing new components or you may burn them out.

2. The Analysis of Simple Circuits

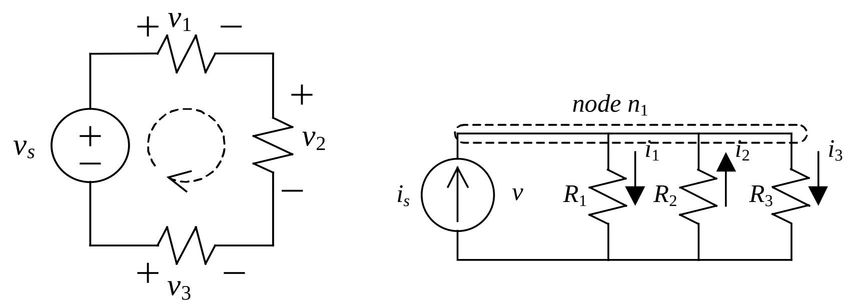
Kirchhoff's Voltage Law (KVL) and Kirchhoff's Current Law (KCL)

There are two laws that define basic circuit theory. Both laws are now described along with a step by step procedure for analyzing circuits.

KVL simply says that the sum of the voltage drops around a closed loop is zero. (Voltage is potential energy per unit charge of electrons). KVL is based on conservation of potential energy. The convention of summing the voltages begins at any point in the circuit, continuing in a loop until you return to the same point. If you hit the "+" side of a component before you reach the "-" side, then you add that voltage. If you hit the "-" side first then you subtract that voltage.

KCL simply says that the sum of the currents entering a node equals zero. This is based on conservation of charge (or electrons). A node is any place where two or more components join. By convention, the current entering a node is added, while the current leaving a node is subtracted. Also, recognize that any two nodes connected only by wire (with nothing else between them) can be regarded as the same node.

Fig. 8 gives a simple example of KVL and a simple example of KCL:



(a) KVL: $-v_s + v_1 + v_2 - v_3 = 0$ (b) KCL at node n_1: $i_s - i_1 + i_2 - i_3 = 0$

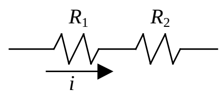
Figure 8: KVL and KCL in a circuit.

In essence there are three analysis steps to determine the circuit variables. These are given below:

Step 1: List the *v*/*i* relationship for each component (e.g. $v=iR$ and $i=Cdv/dt$).
Step 2: List the KVL and KCL equations.
Step 3: Solve the KVL and KCL equations.

Series Equivalent Resistance and Voltage Division

Now that we can analyze circuits in general, we can turn our attention to specific circuits of interest to us. The first circuit that we will analyze can be used to divide voltages. To accomplish this, it is instructive to look at series equivalent resistance. Two resistors are in *series* if they share the same current.



We can find the *equivalent resistance* of the two resistors by finding what resistance R_{eq} in circuit 2 yields the same voltage v_s and current i as is in circuit 1. Figure 9 shows the analysis.

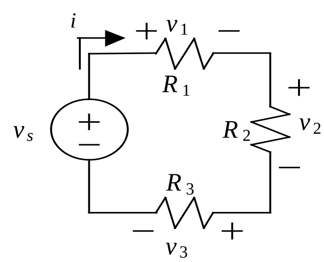
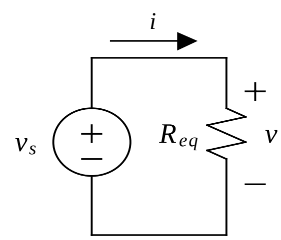
	Circuit 1	Circuit 2
		
Step1: *v*/*i* relationships	$v_1 = iR_1,\ v_2 = iR_2,\ \ v_3 = iR_3$	$v = iR_{eq}$
Step 2: Using KVL	$-v_s + v_1 + v_2 + v_3 = 0$	$-v_s + v = 0$
Step 3: Solving equations	$-v_s + iR_1 + iR_2 + iR_3 = 0$ $v_s = i(R_1 + R_2 + R_3)$	$v_s = iR_{eq}$

Figure 9: Analysis of circuits having equivalent resistance

The two circuits shown in Fig. 9 are equivalent for any voltage and current when

$$\boxed{R_{eq} = R_1 + R_2 + R_3}$$

Using Ohms Law (step 1), notice that across any resistor $\quad i = \dfrac{v_1}{R_1} = \dfrac{v_2}{R_2} = \dfrac{v_3}{R_3}$

We can compactly write the above equation as $i = \dfrac{v_n}{R_n}$, where n could be 1, 2, or 3.

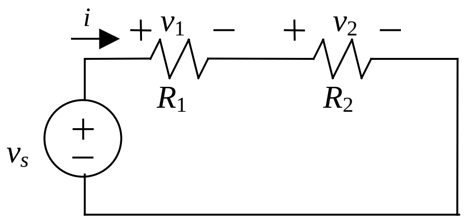
Since it is also true that $i = \dfrac{v_s}{(R_1 + R_2 + R_3)}$, we can see that $\dfrac{v_n}{R_n} = \dfrac{v_s}{(R_1 + R_2 + R_3)}$, or that

$$\boxed{v_n = \frac{R_n}{(R_1 + R_2 + R_3)} v_s}.$$

This is known as *voltage division*.

Experiment 1: Measuring Currents and Voltages

The first step in this experiment is to wire up the following schematic on a breadboard.



v_s = 15 volts
R_1 = 4.7 kΩ
R_2 = 10 kΩ

We first wish to predict the values of v_1, v_2, and i. Toward this end, v_1 and v_2 are predicted from the previously derived equation. We set

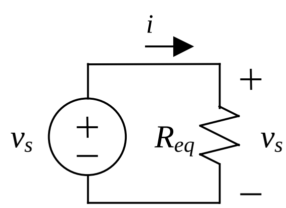
$$v_1 = \frac{R_1}{R_1 + R_2} v_s = 4.80 \text{ V}, \qquad v_2 = \frac{R_2}{R_1 + R_2} v_s = 10.20 \text{ V}$$

Next we reduce the circuit to predict the current i.
This yields

$$R_{eq} = R_1 + R_2 = 14.7 \text{ k}\Omega,$$

so

$$i = \frac{v_s}{R_{eq}} = 1.02 \text{ mA}$$

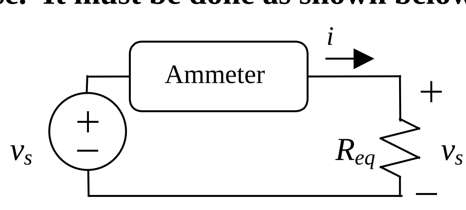


NOTE: If you are having trouble, refer to Figure9a.

The second step in the experiment consists of measuring the voltages across R_1 and R_2 and the currents through the resistors and to compare these measured values with your predictions.

We measure voltages by placing the multimeter in DC voltmeter mode (sometimes symbolized by **V**⎓). Since voltage goes *across* a component, place the two voltmeter leads **on each side** of the resistor. The voltmeter will then measure the voltage between the red lead and the black lead.

We measure current by placing the multimeter in DC ammeter mode (sometimes symbolized by **A**⎓). Place the leads **in series with** the resistor. To do this we must physically disconnect a resistor lead and insert the ammeter into the circuit. Alligator clips are the best tool. **WARNING: If the multimeter is connected across the component (like when measuring voltage), it will blow a fuse. It must be done as shown below.**



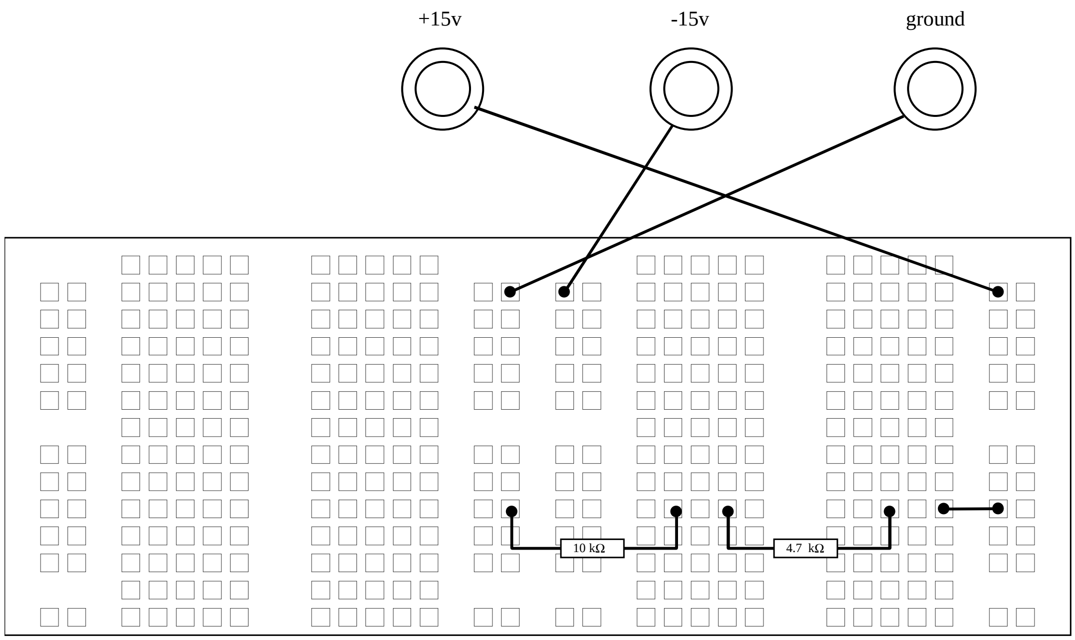
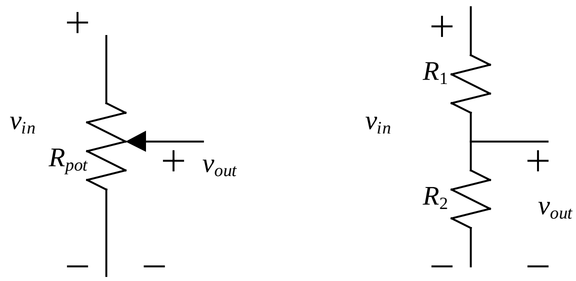


Figure 9a: Position Diagram for Experiment 1

Experiment 2: Using A Potentiometer

As you read earlier, a pot is a variable resistor. It can be modeled as two resistors, whose sum equals the maximum (total) resistance of the pot.



$$R_{pot} = R_1 + R_2$$

By voltage division

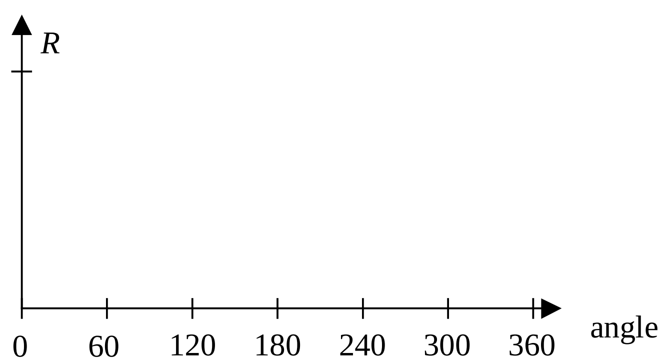
$$v_{out} = \frac{R_2}{R_1 + R_2} v_{in} = \frac{R_2}{R_{pot}} v_{in} .$$

Since v_{in} and R_{pot} are constants, this equation really says that v_{out} is in direct proportion to the position of the potentiometer.

In this experiment, we are given a pot and we first wish to measure the resistance between all 3 terminals. Judging by the magnitudes of the resistances, which 2 terminals are the fixed terminals and which terminal is variable?

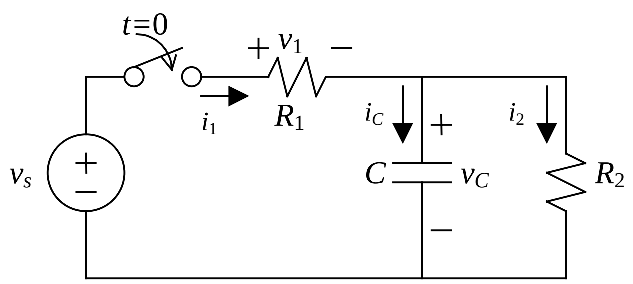
Next, take measurements with the pot turned all the way clockwise, all the way counterclockwise, and four or five measurements in between. Plot the values of the resistance as a function of the angle. Determine whether your pot is a linear pot or an audio taper pot.

angle	*R*
0	
60	
120	
180	
240	
300	
360	



The RC Circuit

The next circuit that we'll analyze is called the RC circuit. We'll see what happens when a resistor and capacitor are placed together in a circuit. The RC circuit and the analysis of the circuit are shown in Figure 10.



v_s = 15 V
R_1 = 300 kΩ
R_2 = 300 kΩ
C = 100 μF

Step 1: v/i relationships: $v_1 = i_1 R_1$, $v_C = i_2 R_2$, $i_C = C\dfrac{dv_C}{dt}$

Step 2: Using KVL and KCL: $-v_s + v_1 + v_C = 0$, $i_1 - i_C - i_2 = 0$

Step 3: Solving equations:

Since $i_1 = \dfrac{v_1}{R_1} = \dfrac{v_s - v_C}{R_1}$ then $\dfrac{v_s - v_C}{R_1} - C\dfrac{dv_C}{dt} - \dfrac{v_C}{R_2} = 0$ so the differential equation that describes the voltage is

$$R_1 R_2 C \frac{dv_C}{dt} + (R_1 + R_2) v_C = R_2 v_s .$$

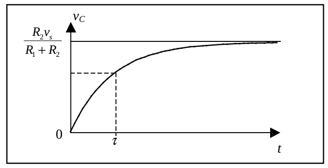
Figure 10: Analysis of an RC circuit

The analysis of the RC circuit performed in Fig. 10 leads us to a linear first order differential equation describing the voltage across the capacitor, v_C. The general solution of this equation is

$$v_C(t) = Ae^{-\frac{R_1+R_2}{R_1 R_2 C}t} + \frac{R_2 v_s}{R_1 + R_2},$$

where A depends on the initial conditions. You can verify that this expression for v_C solves the differential equation by plugging it back into the differential equation. If the switch is initially open and it is closed at $t = 0$, then the initial condition of the circuit is $v_C(0) = 0$. Substituting this into the solution for $v_C(t)$ yields $A = -\dfrac{R_2 v_s}{R_1 + R_2}$. The solution is then

$$v_C(t) = \frac{R_2 v_s}{R_1 + R_2}\left(1 - e^{-\frac{t}{\tau}}\right)$$

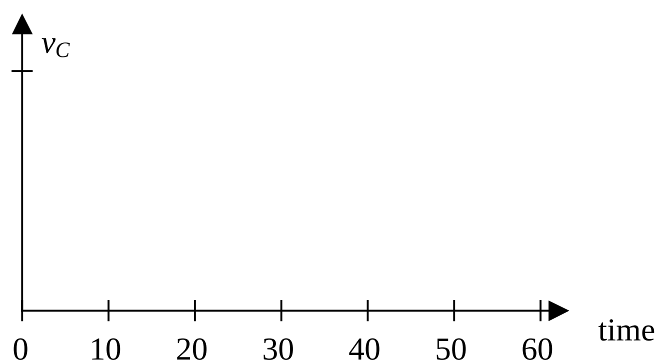


where $\tau = \frac{R_1 R_2 C}{R_1 + R_2}$ is called the *time constant* of the circuit. This equation shows us that resistors and capacitors, when placed together in a circuit, act together as a "filter"—slowing down the transition between the "open state" of $v_C(t)$ and the "closed state" of $v_C(t)$. Indeed, the amount of time to transition between these states is characterized by the time constant τ.

Experiment 3: Measuring the Accumulation of Voltage Across a Capacitor.

First, build the RC circuit that was described above but leave the supply voltage disconnected (that is, leave the switch open). Next, get ready to measure the voltage across the capacitor. Connect the voltmeter across the capacitor in preparation for taking voltage measurements. The instant you connect the supply voltage to the circuit (by closing the switch) you will need to begin recording voltages, continuing to do so every 10 seconds. Record the voltage readings in the table below. When you're done, plot the results on the graph. Compare your measured voltages with the predictions of $v_C(t)$.

t (sec.)	v_C (V)
0	
10	
20	
30	
40	
50	
60	



Operational Amplifiers

In order to appreciate how to use op amps, it's important to appreciate how they are used to create analog circuits. Specifically, as mentioned earlier in this little book, op amps fulfill three needs. First, they enable voltages to be *amplified*. Secondly, they can act as *buffers*, which means that they can isolate the input voltage from the output voltage. Third, the output voltages are independent of the output load, which means that the output voltages remain the same regardless of the resistance (load) that the output is connected to. This enables op amp circuits to be cascaded with other components. This property of buffering and cascading enables op amps to be used with other analog components to create more complicated circuits which can perform various operations—this is why they are called *operational* amplifiers.

Let's now take a closer look at the internal workings of op amps and analyze them.

Derivation of Op Amp Assumptions

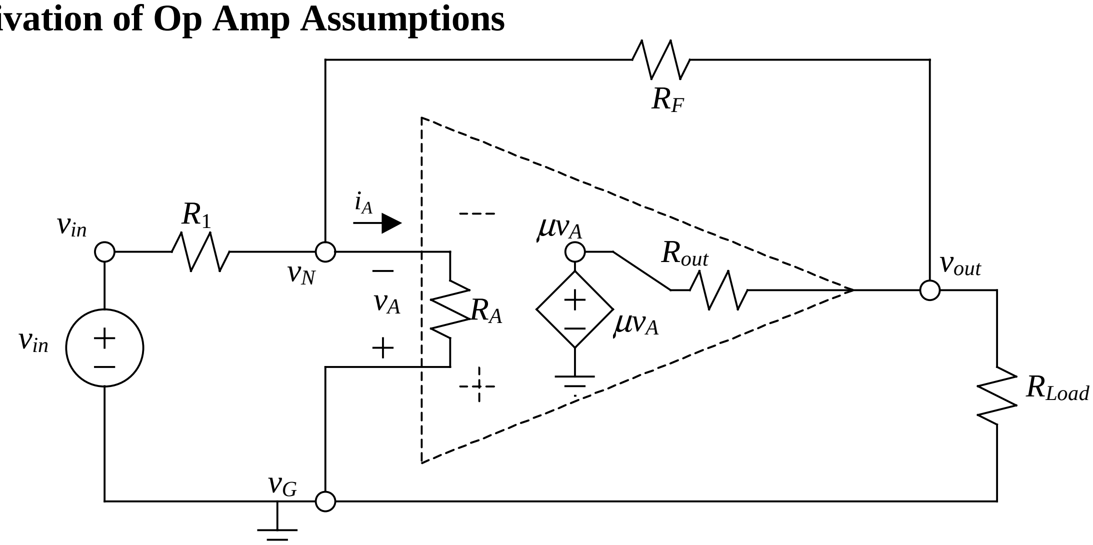


Figure 11: An op amp in negative feedback mode

In Fig. 11, we see an op amp with a resistor R_F connecting the output to the negative input of the op amp. The op amp's internal values are designed to be:

$$R_A \to \infty,\ R_{out} \to 0,\ \mu \to \infty,$$

and v_A is defined with the same polarity as the "+" and "–" on the op amp. In Fig. 11, the voltage of the ground node is zero, so $v_G = 0$ (without loss of generality).

Using KCL we get

At node v_N:

$$\frac{v_{in} - v_N}{R_1} + \frac{0 - v_N}{R_A} + \frac{v_{out} - v_N}{R_F} = 0 \qquad (1)$$

At node v_{out}:

$$\frac{\mu v_A - v_{out}}{R_{out}} + \frac{0 - v_{out}}{R_{Load}} + \frac{v_N - v_{out}}{R_F} = 0 \qquad (2)$$

Notice in Fig. 11 that we can express v_A in terms of other voltages

$$v_A = v_G - v_N\,. \qquad (3)$$

From Eqs. (2) and (3), and since $v_G = 0$ we get

$$v_A = \frac{v_{out}\left(R_{Load}R_F + R_{out}R_F + R_{out}R_{Load}\right)}{R_{out}R_{Load} - \mu R_{Load}R_F}$$

Since $R_{out} \to 0$, this equation reduces to

$$v_A = \frac{v_{out}\left(R_{Load}R_F\right)}{-\mu R_{Load}R_F} = \frac{v_{out}}{-\mu}$$

But since $\mu \to \infty$, $R_A \to \infty$, and $i_A = \frac{v_A}{R_A}$, we now obtain the *two op amp assumptions*

$$\boxed{v_A \approx 0} \qquad \boxed{i_A \approx 0}.$$

These are the two assumptions that we make without having to analyze the entire circuit each time.

Saturation of Op Amps

When the negative feedback resistor R_F in Fig. 11 is removed, the output voltage can no longer influence the voltages at the inputs. In other words, the input becomes independent of the output, and thus v_A can no longer be held to zero. The equation for v_{out} is then

$$v_{out} = \mu v_A\,.$$

Remember that as μ approaches infinity and v_A can not be held to zero, the result is an unbounded output voltage, i.e., v_{out} approaches infinity, at least in theory. In practice, v_{out} saturates.

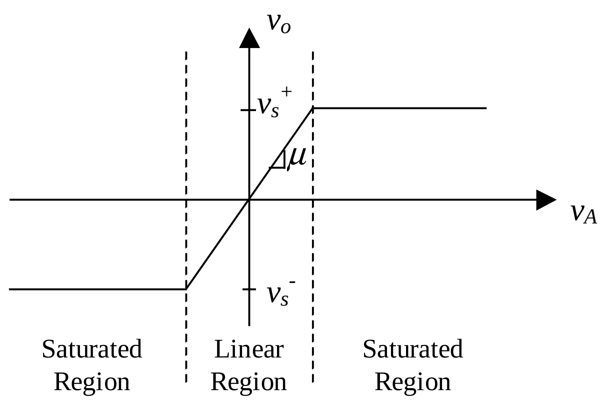


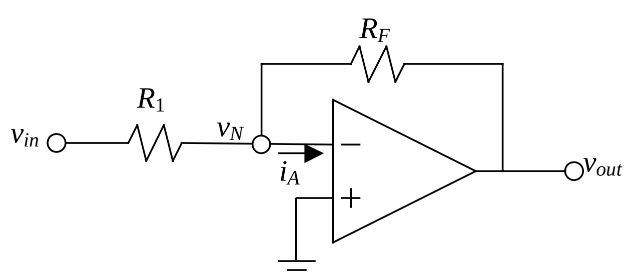
Figure 12: The two operating regions of the op amp

In Fig.12 we see that the op amp has two regions of operation: the saturated region (when there is no negative feedback) and the linear region (when there is negative feedback). To use the op amp for mathematical operations, it is necessary to use the op amp in the linear region. Since as μ approaches infinity, the linear region is very narrow, and is easily passed by when v_A isn't very close to zero. The upper and lower limits on the saturation occur because the op amp cannot supply any higher voltage than it actually receives via v_s^+ and v_s^-.

Indeed, we can see that without negative feedback, which stabilizes the op amp at $v_A = 0$, we have a circuit that performs a threshold function, which is governed by the equation

$$v_{out} = \begin{cases} v_{cc}^+ & if \;\; v_A < 0 \\ v_{cc}^- & if \;\; v_A > 0 \end{cases}.$$

Signal Gain and Signal Inverting



From KCL we have $\frac{v_{in} - v_N}{R_1} + \frac{v_{out} - v_N}{R_F} - i_A = 0$. Since the voltage across the op amp input terminals is zero, then $v_N = 0$ and $i_A = 0$, so

$$\boxed{v_{out} = -\frac{R_F}{R_1} v_{in}}.$$

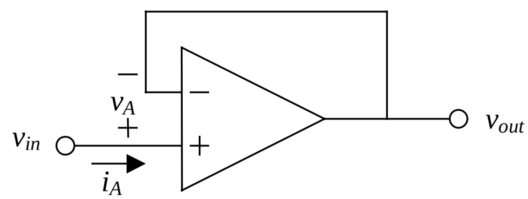
This allows us to amplify the input voltage by some gain $-R_F/R_1$.

A signal is inverted by setting the gain equal to –1. If we let $R_1 = R_F$ above then we get

$$v_{out} = -v_{in}.$$

This inverts the sign of the input signal.

Signal Buffering

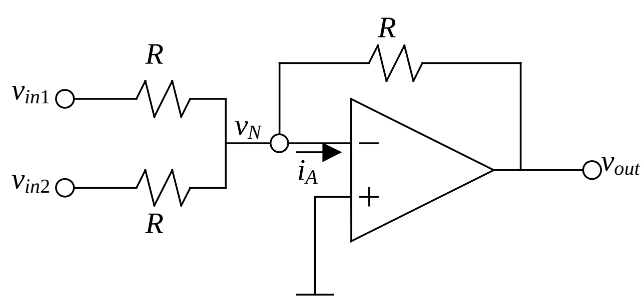


We showed earlier that an op amp with negative feedback has $v_A = 0$ and $i_A = 0$. Since we can see from KVL that $v_{out} + v_A = v_{in}$, this means that

$$\boxed{v_{out} = v_{in}}.$$

You may wonder about the usefulness of a circuit whose output mirrors the input, but realize that the input current is zero. This means you can measure v_{in} without changing its value. This is why it is called a buffering circuit.

Signal Addition

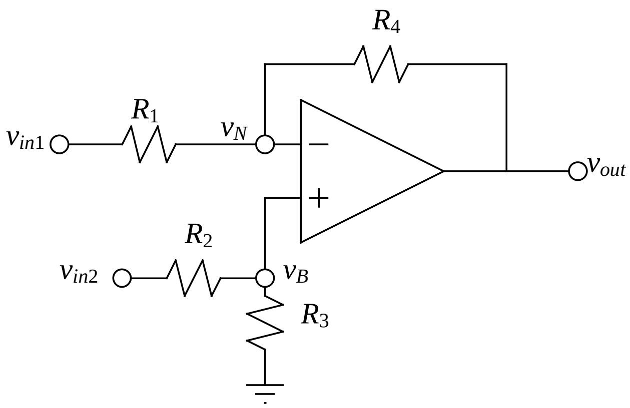


From KCL we have $\frac{v_{in1} - v_N}{R} + \frac{v_{in2} - v_N}{R} + \frac{v_{out} - v_N}{R} - i_A = 0$. Since the voltage across the op amp input terminals is zero, then $v_N = 0$ and $i_A = 0$, so

$$\boxed{v_{out} = -(v_{in1} + v_{in2})}.$$

This allows us to sum several signals.

Signal Subtraction



For the op amp, we see that $v_N = v_B$, and the current entering the op amp inputs is zero. From KCL we get

at node v_N: $$\frac{v_{in1} - v_N}{R_1} + \frac{v_{out} - v_N}{R_4} = 0,$$

at node v_B: $$\frac{v_{in2} - v_B}{R_2} + \frac{0 - v_B}{R_3} = 0.$$

Since $v_A = v_B$, then

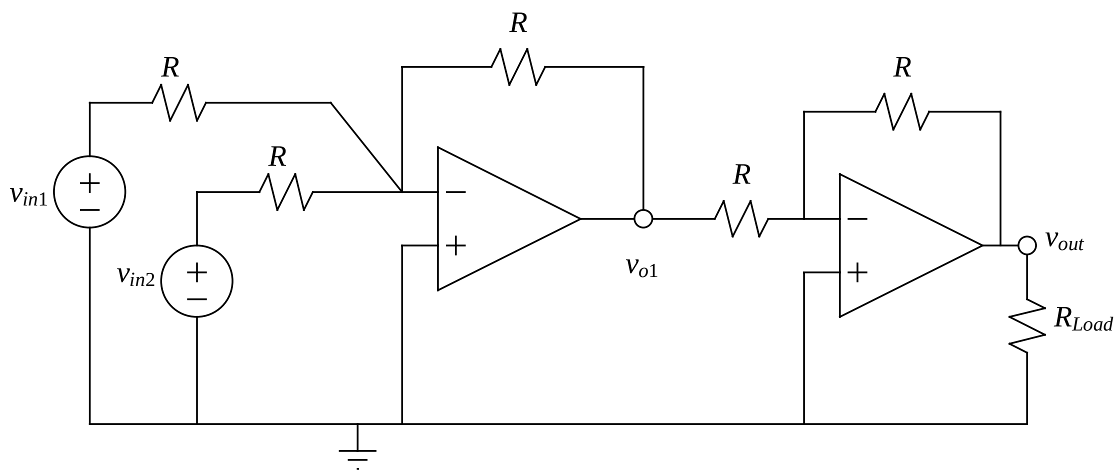
$$v_{out} = \frac{(R_1 + R_4)R_3}{(R_2 + R_3)R_1} v_{in2} - \frac{R_4}{R_1} v_{in1}$$

If we let $R = R_1 = R_2 = R_3 = R_4$, then

$$\boxed{v_{out} = v_{in2} - v_{in1}}$$

Signal Cascading

The op amp maintains a voltage on the output regardless of the current draw by the load. This is a very important property because it means that the load doesn't affect the voltage. If you look at the equation for voltage output, v_{out}, you will notice that the load resistance is not a factor. This allows several op amps to be cascaded together but analyzed separately instead of requiring the whole circuit to be analyzed together.



As an example, the first op amp is a summing circuit, the second op amp is an inverter.

From the previous examples

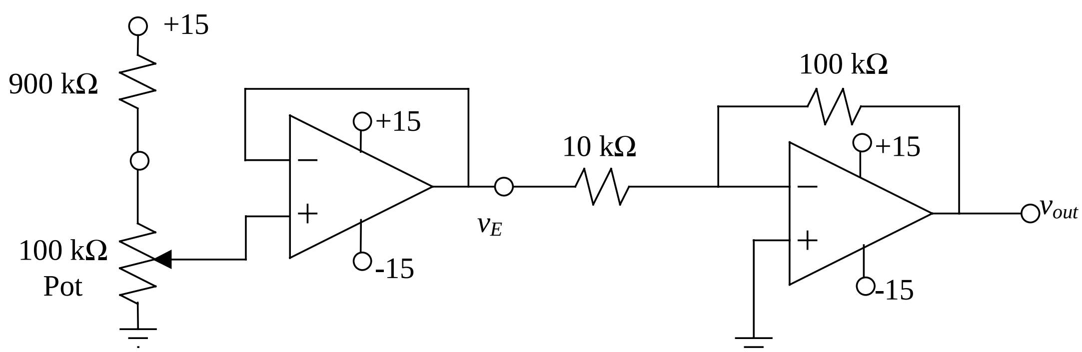
$$v_{o1} = -(v_{in1} + v_{in2}), \text{ and } v_{out} = -v_{o1},$$

so

$$v_{out} = v_{in1} + v_{in2}.$$

Experiment 4: Measuring Op Amp Gain

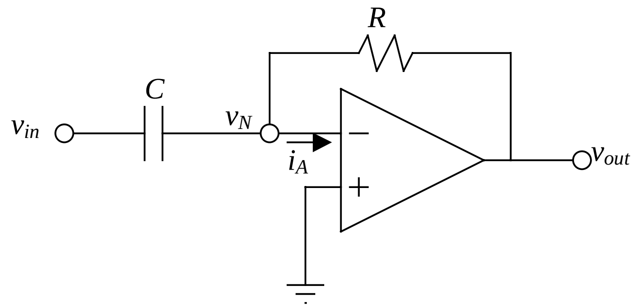
Build this circuit. Remember that you need to provide supply voltage to the op amps.



1) As you turn the pot, measure v_E. It should vary between 0 and 1.5 V.

2) Measure v_{out}. It should vary between 0 and 15 V.

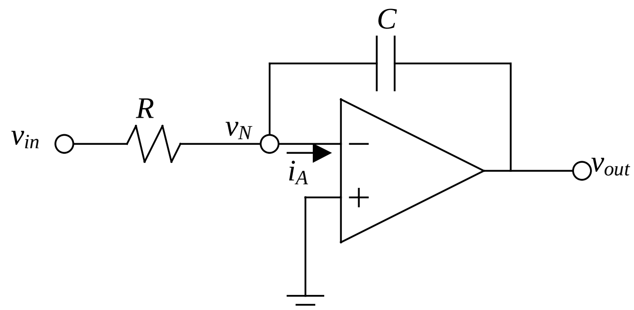
The Derivative Operation



We have an op amp above with negative feedback, so $v_N = 0$, and $i_A = 0$. KCL at node v_N gives $C\frac{d}{dt}(v_{in} - v_N) + \frac{v_{out} - v_N}{R} - i_A = 0$, so

$$\boxed{v_{out} = -RC\frac{dv_{in}}{dt}}$$

The Integral Operation



Again we have an op amp above with negative feedback, so $v_N = 0$, and $i_A = 0$. KCL at node v_N gives $\frac{v_{in} - v_N}{R} + C\frac{d}{dt}(v_{out} - v_N) - i_A = 0$, so

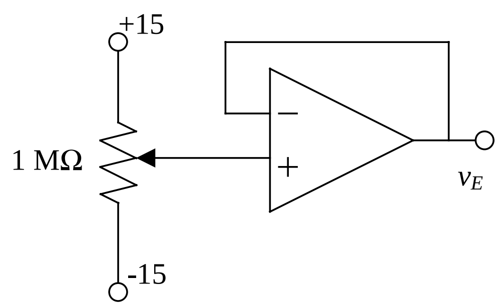
$$\boxed{v_{out}(t) = -\frac{1}{RC}\int_{-\infty}^{t} v_{in}(\tau)d\tau}$$

It is interesting to notice that both the integral and the derivative use a resistor and a capacitor, but in opposite positions. Can you visualize why this would be the case?

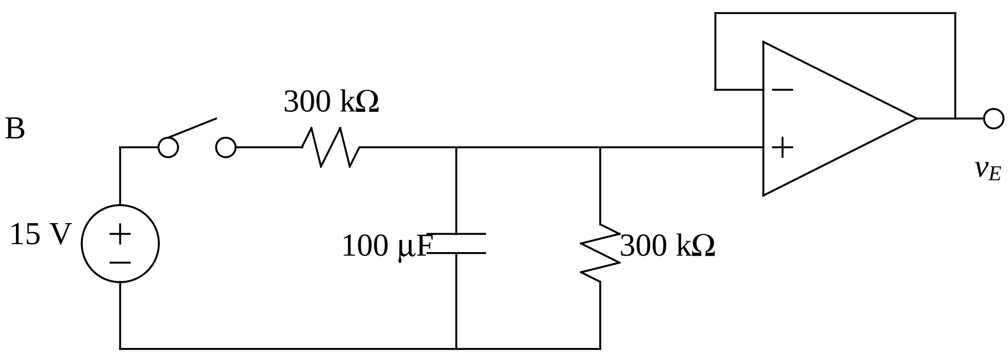
Experiment 5: Measuring Derivatives and Integrals

Build the following two circuits to be used as inputs. They will be used for part one and two of this experiment, so leave them connected after completing part one. Input circuit A is simply a voltage divider, so the output voltage, v_E, will vary between –15 and +15 volts depending on the position of the pot knob. Input circuit B is the RC circuit from experiment 3. It will be used as a time delay input voltage, so you can observe the derivative and integral elements. Remember to provide the op amps the 15 volt supply voltage (which is no longer being explicitly shown).

Input Circuit A

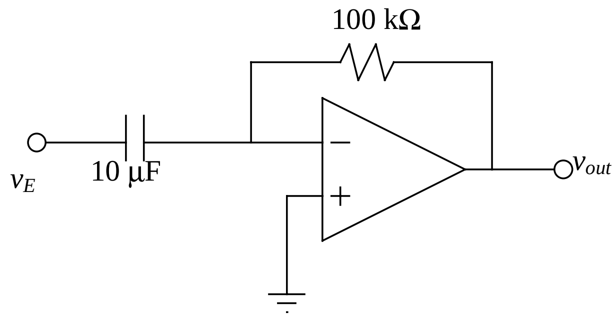


Input Circuit B



Experiment 5, *Part* 1: The Derivative

Build this circuit



1) a) Connect input circuit A to the input at v_E.

   b) Measure v_{out}. Notice that the faster you turn the pot, the higher the magnitude of v_{out}.

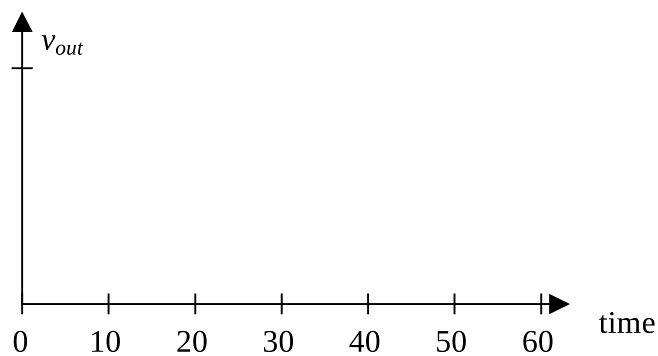
2) a) Connect input circuit B to the input at v_E. Notice this input circuit will produce a slowly changing input voltage, v_{E}. As v_{E} is changing you will see an output voltage from the derivative. When v_{E} stops changing the derivative will go back to zero.

   b) Measure the voltage at v_E. Leave the switch open until it is zero.

   c) Connect the voltmeter to v_{out}. As you close the switch at $t = 0$, measure v_{out} for the next 60 seconds and graph the results.

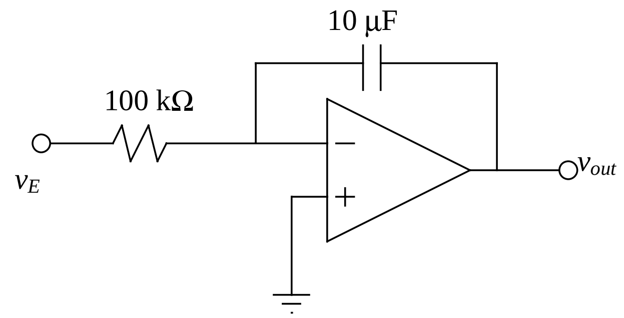
   d) What is the initial value at $t = 0$? What is the time constant of the circuit? How does this compare to the results in Experiment no. 3.

t (sec.)	v_{out} (V)
0	
10	
20	
30	
40	
50	
60	



Experiment 5, *Part* 2: The Integral

Build this circuit



1) a) Connect input circuit A to the input at v_E.

   b) Measure v_{out}. Notice that the value of v_{out} increases or decreases at a rate that is in proportion to the position of the pot. However, v_{out} saturates at the maximum voltage supplied to the op amp. Try to make v_{out} equal to zero and hold it there.

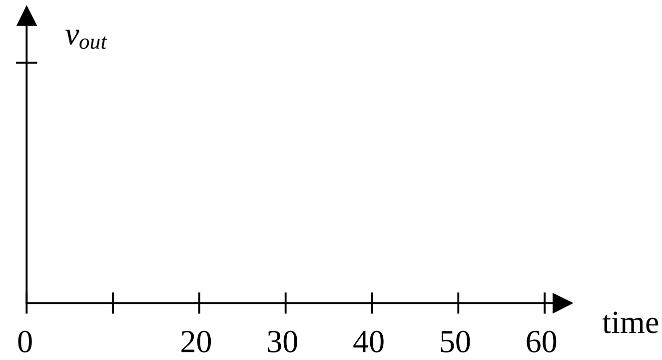
2) a) Connect input circuit B to the input at v_E. Again, this input circuit will slowly increase its output voltage. You will be able to observe the integral circuit adding small voltages as time progresses until it reaches saturation.

   b) Measure the voltage at v_E. Leave the switch open until it is zero.

   c) Connect the voltmeter to v_{out}. As you close the switch at $t = 0$, measure v_{out} for the next 60 seconds and graph the results.

   d) What is the initial value of v_{out} at $t = 0$? What is the time constant of the circuit? How does this compare to the results in Experiment no. 3.

t (sec.)	v_{out} (V)
0	
10	
20	
30	
40	
50	
60	



3. Proportional, Integral, Derivative (PID) Control Theory

Terminology

Before proceeding with PID control theory, there are some terms we need to define. The *set point* is the position were you *want* your system to be. In mechanical systems you are talking about the position of a gear or another mechanism. The *process variable* is the position where the system currently is. The difference between the set point and process variable is your *error*. You would like for your controller to force the error to zero.

We have more terms to describe how the PID control system reduces the error. The *settling time* is how long it takes the error to reach its final value. The *overshoot* is the peak value of the error. Finally, the *steady state error* is the value where the error settles. Figure 13 shows the error of a system versus time as the control effort is being applied.

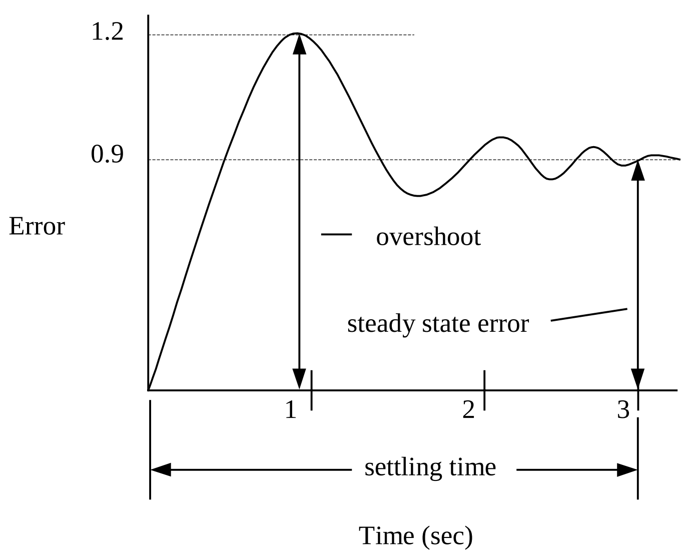


Figure 13: A system settling after a control effort is applied

Control Theory

Many mechanical systems, for the purpose of control, can be broken down into single degree-of-freedom systems. Therefore, a good way to describe linear control theory is within the context of single degree-of-freedom systems. Consider a single degree-of-freedom system shown in time, represented by the linear differential equation

$$m\ddot{x}(t)+c\dot{x}(t)+kx(t)=f_c(t)+f_d(t) \quad (1)$$

where $x(t)$ is the displacement, $f_c(t)$ is the control force, $f_d(t)$ is the disturbance force, and *m* denotes mass, *c* denotes damping, and *k* denotes stiffness. The PID control law has the general form

$$f_c(t)=-gx(t)-h\dot{x}(t)-i\int x(t)dt \quad (2)$$

where *g*, *h*, and *i* are called *control gains*. In equation (2), a control force that is composed of three parts is applied. The first part, $-gx(t)$, provides an "artificial spring" force. The second part, $-h\dot{x}(t)$, provides an "artificial damper" force. The last term, $-i\int x(t)dt$, produces a force that opposes an accumulation of x(t) over time. Letting $f_d(t)=0$, the general solution to equation (1) is

$$x(t)=Ae^{-\gamma t}+e^{-\alpha t}\left(B\cos(\beta t)+C\sin(\beta t)\right) \quad (3)$$

where γ is the *steady-state damping rate*, α is the *vibration damping rate*, and β is the *closed-loop frequency of oscillation*. The constants *A*, *B*, and *C* are determined by the initial conditions.

You can verify, by substitution, that equation (3) satisfies the differential equation (1) and (2). In the process of doing this, you will find that the control gains *g*, *h*, and *i* and the *performance parameters* α, β, and γ are related by

$$\begin{aligned} g &= m\left(2\alpha\beta+\alpha^2+\beta^2\right)-k \\ h &= 3\alpha m-c \\ i &= \gamma m\left(\alpha^2+\beta^2\right) \end{aligned} \quad (4)$$

Equations (4) are algebraic relationships between the control gains (*g*, *h*, *i*) that you apply, and the parameters (α, β, γ) that dictate the performance that you achieve. Thus, equations (4) can be used to determine control gains on the basis of a performance that you want to achieve (described by α, β, and γ). In this little book, we won't do this explicitly, however, referring to equations (3) and (4) we can see that the control gain *g* is primarily responsible for controlling peak overshoot (stiffness), that *h* is primarily responsible for controlling settling time (damping) and the *i* is responsible for steady state error.

After building the your own PID controller, you will be able to observe the characteristics of each control element using a horizontal pendulum. You will be able to displace the pendulum and watch how the PID controller returns the system to the set point depending on the gains (*g*, *h*, *i*) that you set. You will be able to make your system stiffer, change the settling time, and eliminate steady state errors.

4. Building the Complete PID Controller

Introduction

By this time you should have completed the simple circuit experiments, and you are ready to start building the complete PID Controller. We will take the complete circuit diagram shown in Fig. 14 and build it on a breadboard one piece at a time.

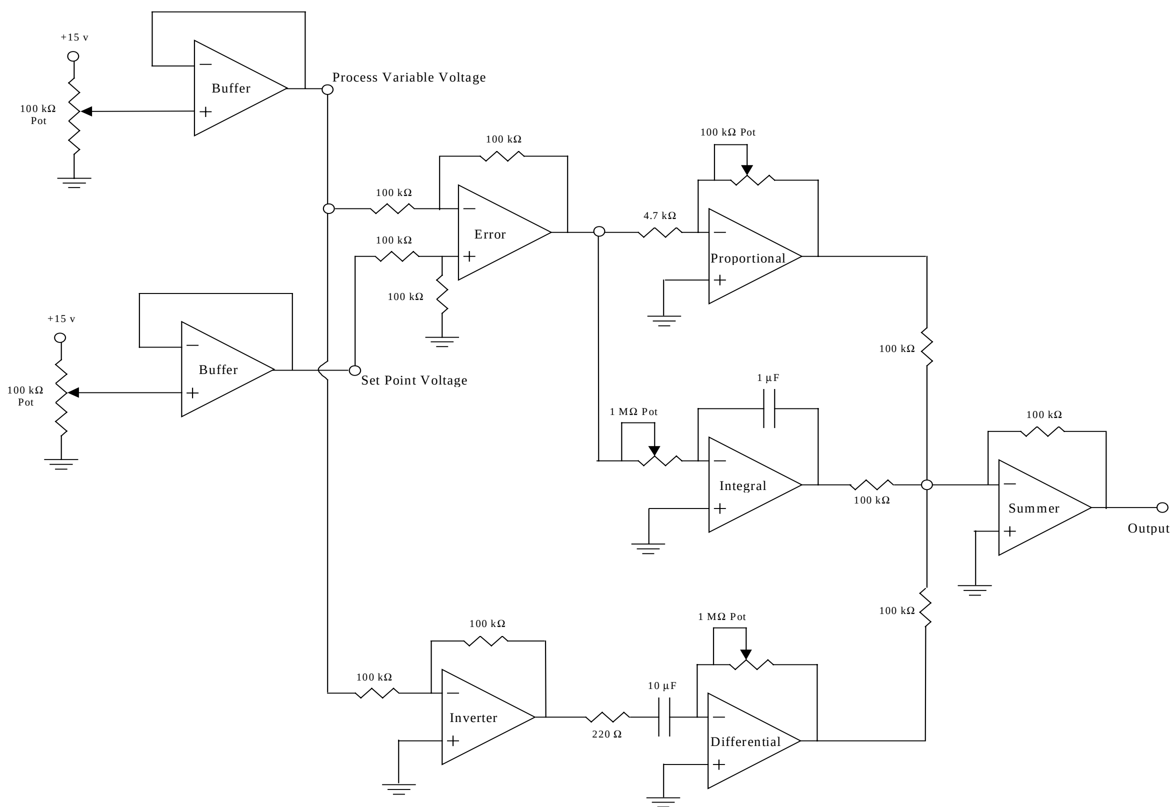


Figure 14: Complete circuit diagram for PID controller

Setting Up the Breadboard

You should already have voltage available to the board. Now, install eight op amps across two sets of horizontal rows and connect jumper wires for supply voltage as shown in Fig. 15. We will use these op amps to construct the entire PID Control Circuit.

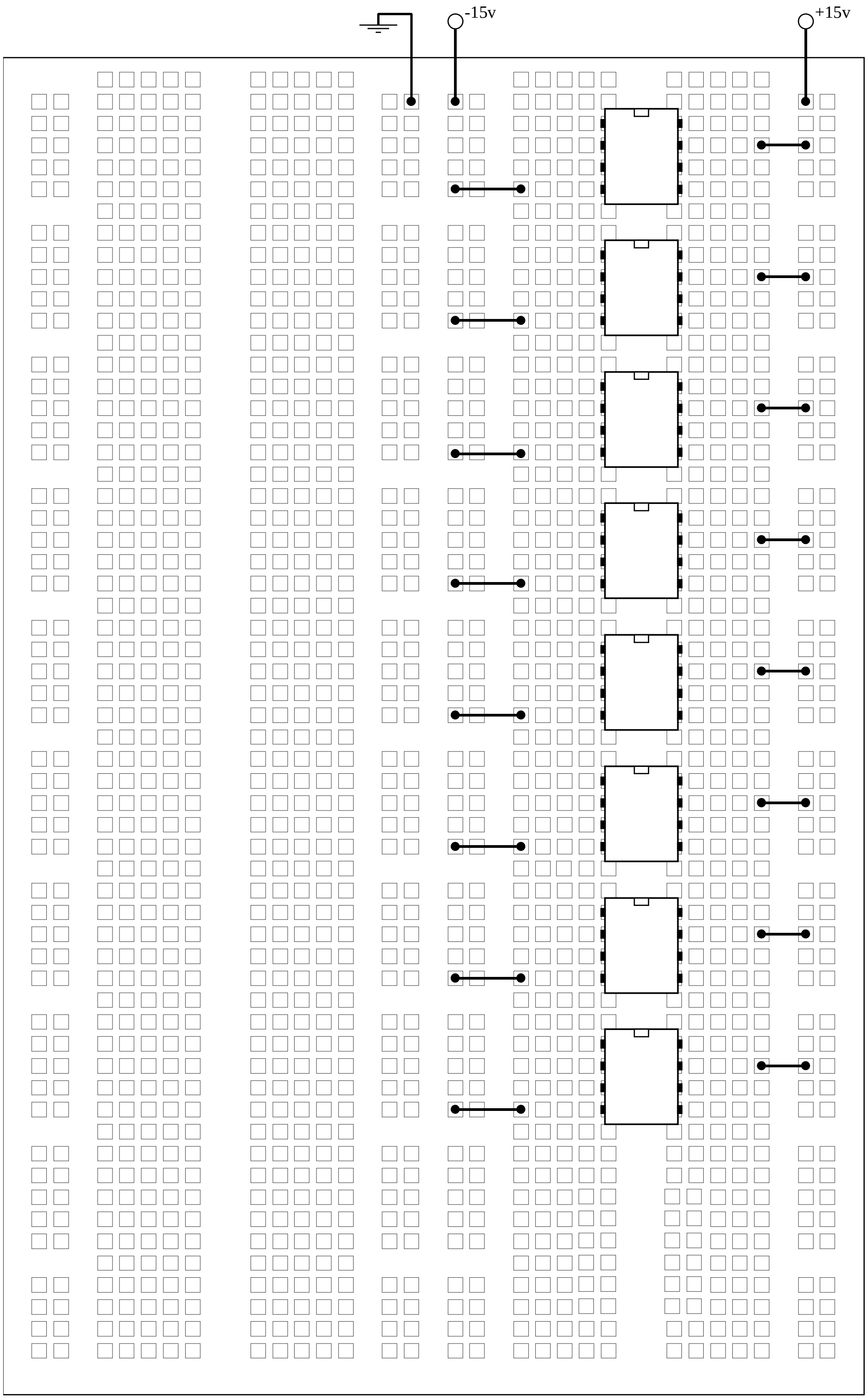


Figure 15: Placement of op amps and jumper wires

Set Point and Process Variable

In your PID controller you will start out by setting up two pots as voltage dividers; one to represent the set point and the other to represent the process variable. The difference between the voltage outputs will be the error. Therefore, when both pots are set to the same resistance the two voltages will be the same, and there will be no error. You will also put buffer op amps in front of the pots so that changing their resistances does not affect the voltages throughout the rest of the controller.

The circuit diagram for the set point and process variable are shown in Figs. 16a-b. You now follow a step by step procedure for placing the components on the breadboard (see Fig. 16b).

1. Place two 100kΩ pots in the bank of horizontal rows to the left of the op amps such that the "top" of each pot is on the right side. Then, connect the pots as a voltage dividers (between 0 and +15 volts) as shown in Figs. 16a-b. This will allow the pots to control the voltage output of the set point and process variable.
2. Now, use jumper wires to set up each of the first two op amps as buffers (also shown in Figs. 16a-b).

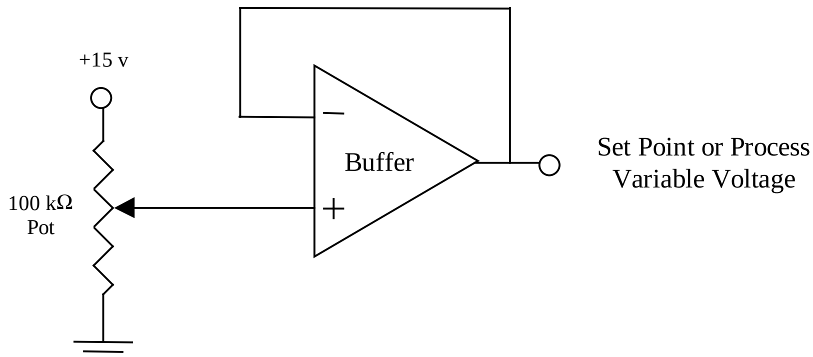


Figure 16a: Circuit diagram for set point and process variable

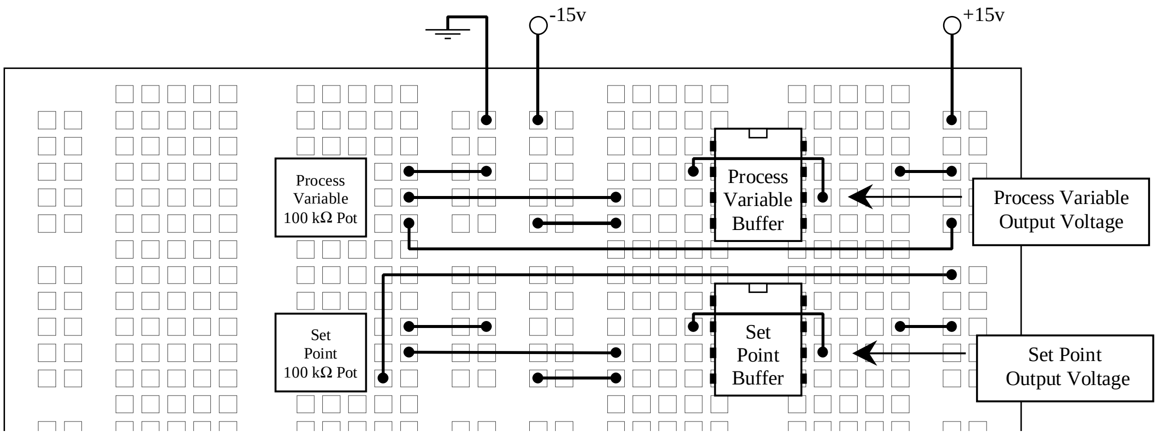


Figure 16b: Component positions for process variable and set point

3. Let's now do a quick test to insure things are functioning properly. After powering up the breadboard, set your multi-meter to DC voltage and place the black lead to ground and the red lead to the output connectors of each of the first two op amps (see Fig. 16c). As the adjustment knob on each pot is turned counterclockwise, the voltage should go to zero. As the adjustment knob is turned clockwise, the voltage should go to +15 volts. If the voltage changes in the opposite direction, switch the two outside leads. If your voltage isn't changing correctly, try changing the op amps and pots one at a time.

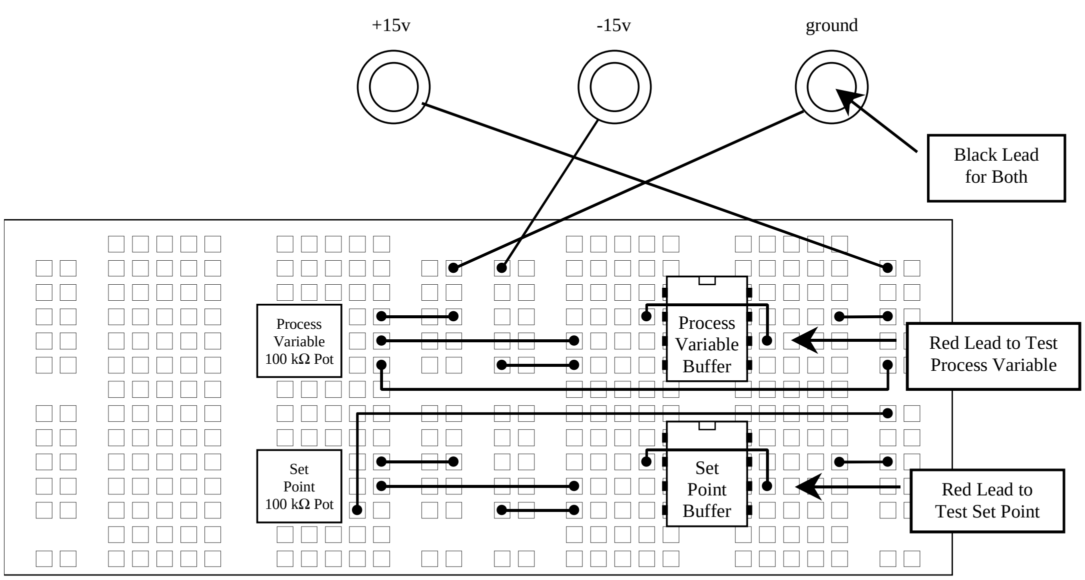


Figure 16c: Positions for testing set point and process variable

Error Comparison

Now that you have output voltages from the set point and process variable, you need to examine the difference between them (our error). For this you will use the third op amp. It will be connected in a unit gain configuration so that its output will be the exact value of the difference between the set point and process variable (remember section called signal subtraction).

4. Insert jumper wires and 100 kΩ resistors according to the circuit diagram shown in Fig. 17a. The actual component positions are shown in Fig. 17b.

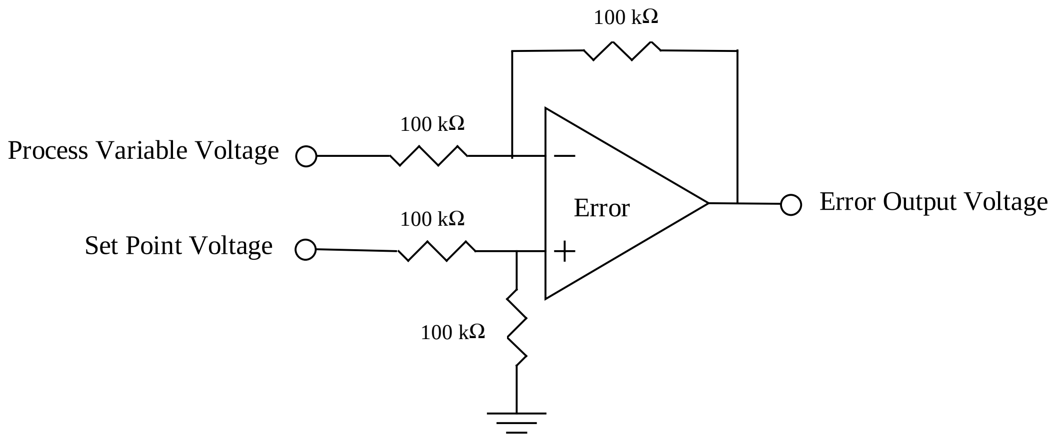


Figure 17a : Circuit diagram for error op amp

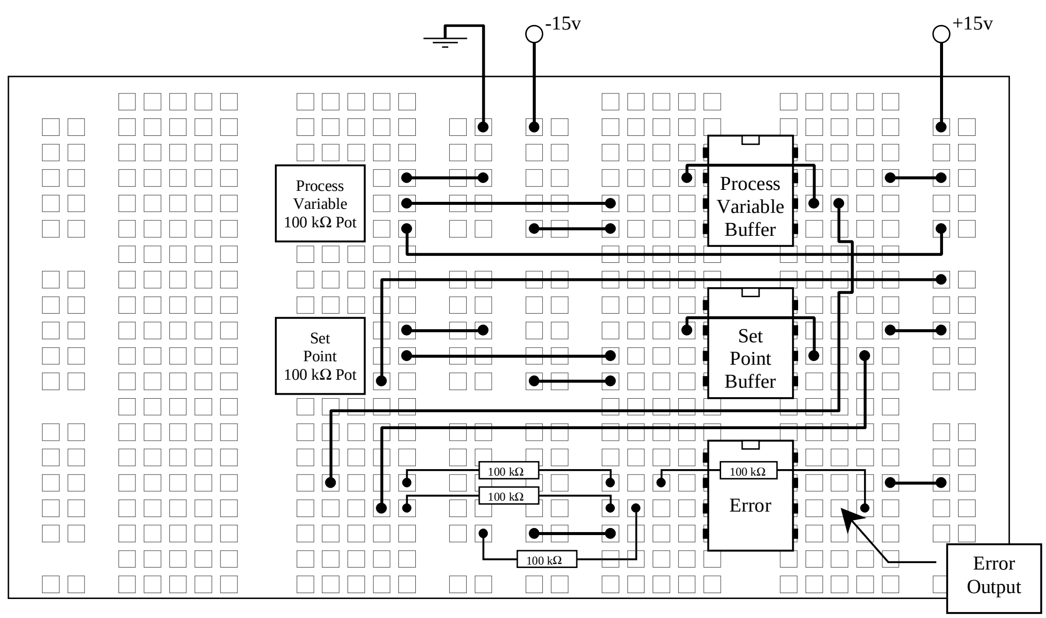


Figure 17b: Component positions for error op amp

5. You will now test to insure that the output of the error op amp is functioning properly. Turn the knob of the process variable pot approximately half way so the output of the pot is approximately 7 volts (see Fig. 17b). Now, turn the set point pot all the way to the left. The output of the error op amp (Fig. 17b) should be the same value as the output of the process variable with the opposite sign (approximately negative 7 volts). As the set point pot knob is turned to the right, the output voltage of the error op amp should approach zero as the knob approaches half way. If you continue to turn the knob all the way to the right, the output voltage should approach approximately positive 7 volts. Do not continue until the error op amp is functioning properly.

Proportional Controller

While setting up the proportional control, you will use a pot in the feedback loop of the op amp to achieve an adjustable gain. You will use a 100 kΩ pot and 4.7 kΩ input resistors. Remember from experiment 4, this means the maximum gain of the op amp (and the proportional controller) is 21.2 when the pot resistance is at its maximum. However, you are limited to a supply voltage of ±15 volts. Suppose the error voltage is 1 volt and the gain is at the maximum, the op amp cannot output 21.2 volts. It is limited to 15 volts. When the pot resistance is zero, the gain of the op amp is also zero.

6. Connect the proportional op amp as shown in Figs. 18a-b. The pot should be connected using the bottom and center leads. This way the resistance will increase as the knob is turned to the right. Please note this time there are 4.7 kΩ resistors being used along with a 100 kΩ resistor.

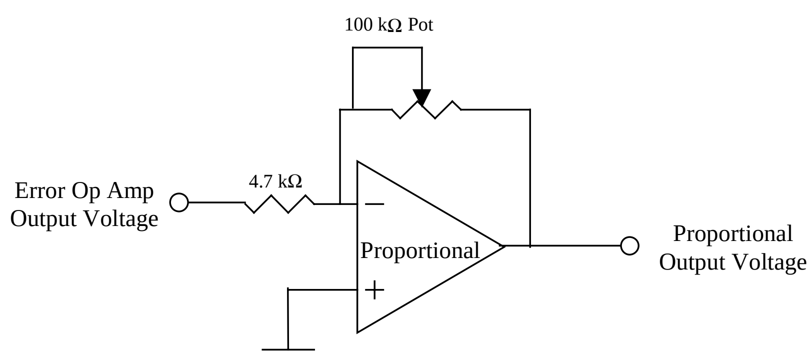


Figure 18a: Proportional controller circuit diagram

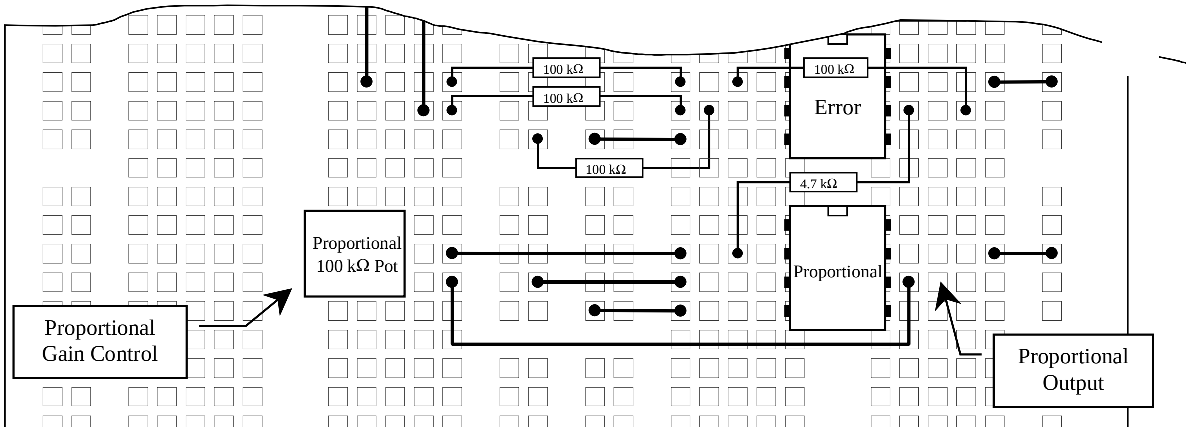


Figure 18b: Proportional controller component positions

7. Testing the proportional controller is relatively simple. Adjust the process variable and set point pots so that the output of the error op amp is approximately one volt. Then, turn the proportional pot all the way to the left (this sets the gain to zero). The output of the proportional op amp should now be zero. As you turn the proportional pot to the right, the output of the proportional op amp should increase reaching a maximum of 15 volts. If the output of the proportional op amp decreases as you turn to the right, switch the wires going into the top and bottom terminal. We will make sure all gains increase as the pots are turned to the right to avoid confusion when testing the controller.

Integral Controller

You now assemble the integral element of the controller using the fifth op amp (like you did in experiment 5). Unlike the proportional controller, increasing the resistance of the pot in the integral element will decrease the gain, so we will connect the pot with the resistance value decreasing as you turn clockwise. Also notice that to completely shut off the integral controller you would need infinite resistance. Since this is not possible, a small effect from the integral controller will always be present.

8. Connect the integral op amp as shown in Figs. 19a-b. The pot should be connected using the top and center leads. This way the resistance will decrease as the knob is turned to the right.

Notice that you are using a 10 kΩ resistor to connect to ground and a 1 μF capacitor in the feedback loop.

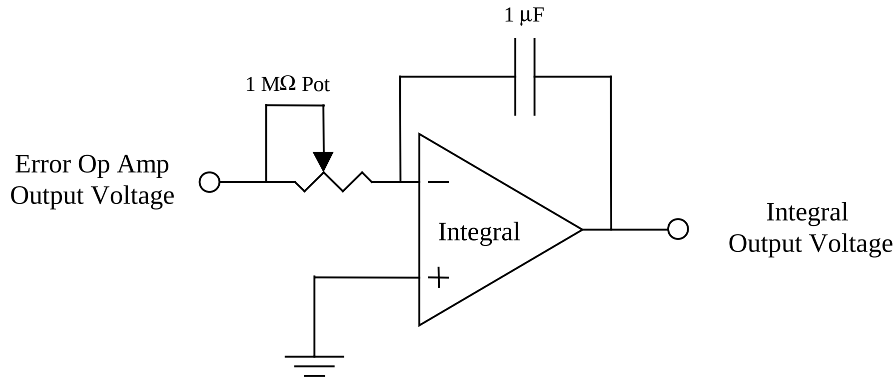


Figure 19a: Circuit diagram of integral controller

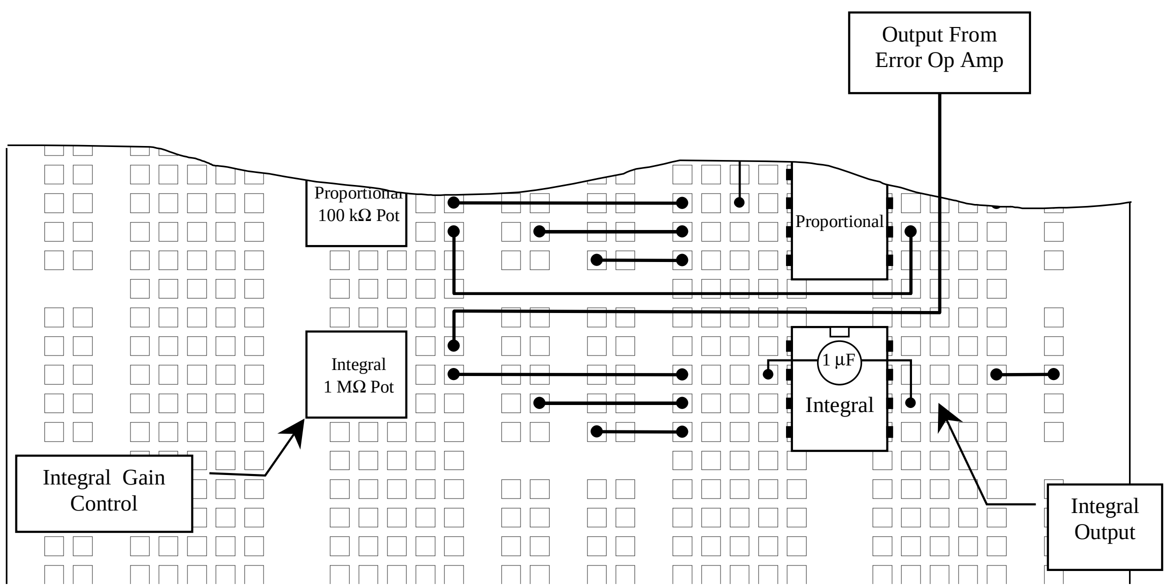


Figure 19b: Component location for integral controller

9. Now, you test the integral controller. First, turn the integral pot all the way to the left. Next, adjust the set point and process variable pots so that the error output is between 0.1 and 0.5 volts. Then, move your voltmeter lead to the output of the integral. You should be able to observe the output voltage slowly increasing until it reaches saturation (around 14 volts). You should also be able to adjust the integral pot to speed up the rate at which the voltage is increasing.

Derivative Controller

Before connecting the derivative op amp, you will go through an inverting op amp. All control efforts have the opposite sign because they oppose the motion of the structure. The inputs into the proportional and integral op amps were already switched by the error op amp. You must now switch the input into the derivative op amp as well.

10. Wire the inverter and derivative controller as shown in Figs. 20a-b.

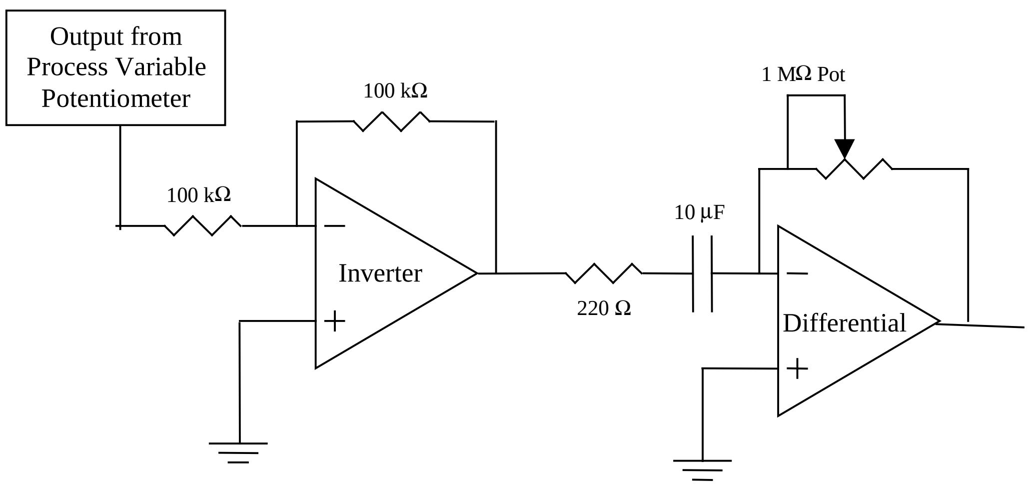


Figure 20a: Circuit diagram for the inverter and the derivative controller

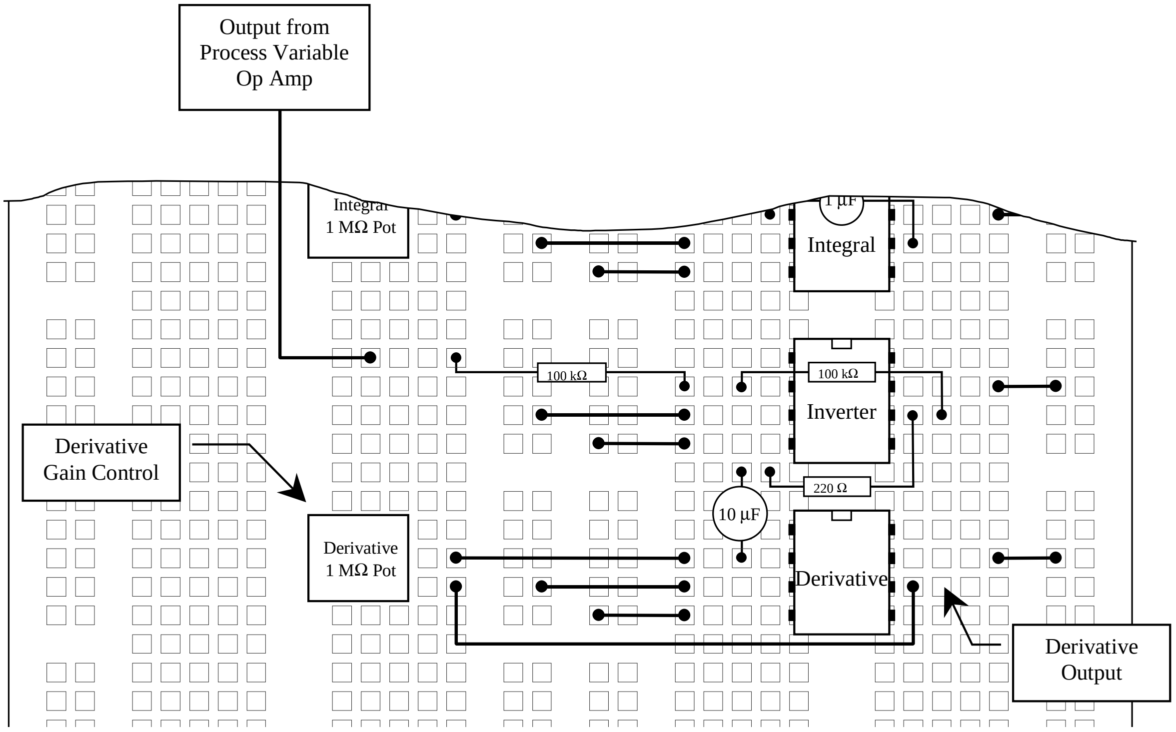


Figure 20b: Component positions for buffer and derivative controller

11. At this point you will test the inverter and the derivative. First, check the voltage at the output of the process variable op amp, and compare this to the output voltage of the inverting op amp. The output of the inverting op amp should be the same magnitude, but opposite sign.

12. Since the derivative acts according to how the process variable is changing, you will have to change the value of the process variable in order to change the derivative. Attach your multimeter lead to the output of the derivative op amp. Now, turn the process variable pot and observe what the value of the derivative op amp does. As you quickly turn the pot, the output value of the derivative should approach ±15 volts, and then quickly fall off when movement stops.

Adding the Control Efforts

You have now completed placing all of the elements of the controller on the breadboard, and you need to add together the control efforts of each element together to get one output signal. This will be done with the last op amp—the summer.

13. Connect the summer as shown in Figs. 21a-b.

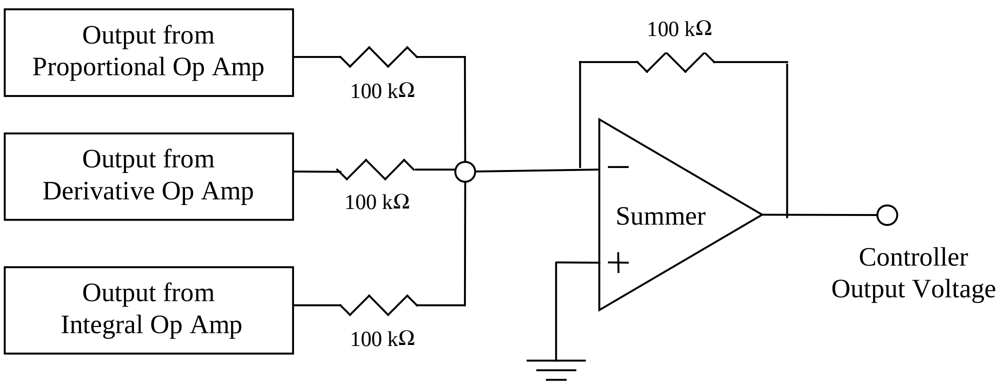


Figure 21a: Circuit diagram for summer

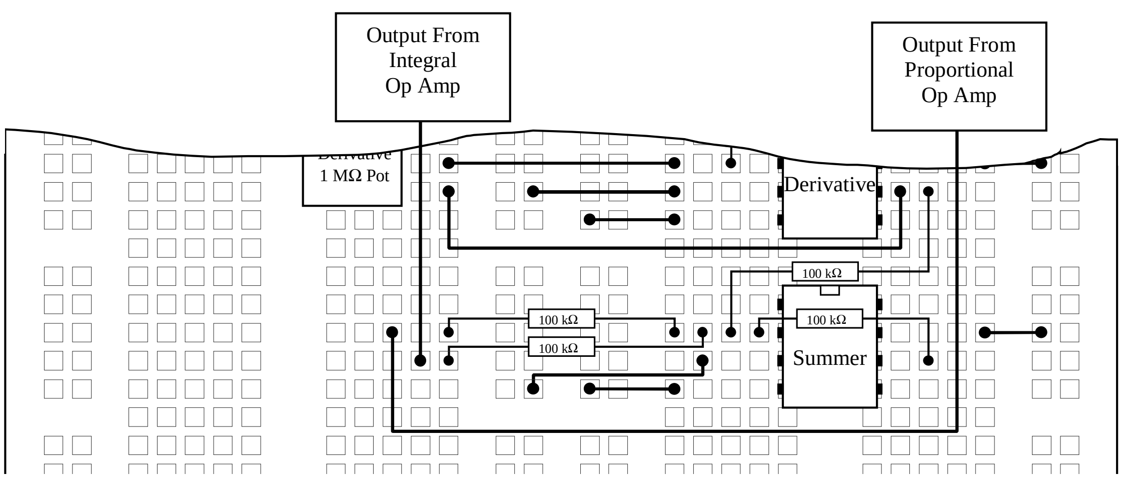


Figure 21b: Component positions for summer

14. You will now complete the controller by checking the summer. Move the set point and process variable pots so that the output of the error op amp is about 0.5 volts. Now, check the output of the proportional and integral op amps. The summer output should be the addition of the two signals.

Connecting to a Physical System

Now that your controller is complete, you can hook it up to a motor and have it control a physical system. However, the op amps you have been using are not designed to output large currents. You will need to increase the current output of your controller using a *power* op amp and a pot.

The pins of the power op amp are the same as the 741 op amp you have been using, but the appearance is round instead of rectangular. There is a small square tab over pin eight to show pin locations. This is shown in Fig. 22a.

Because power op amps handle relatively high currents, they also generate heat. To dissipate the heat, you will need to place a cooling fin (or heat sink) on top of the op amp to provide a steady stream of air over it.

12. Connect the power op amp and pot as shown in Figs. 22a-b. The easiest way to connect the power op amp is to bend the pins into the same position as they were in the 741 op amp. The pot should be connected using the bottom two terminals, so that the resistance is increasing as the knob is turned to the right. It is a common mistake to forget to give the power op amp supply voltage (because you did that in the beginning for the rest of the op amps).

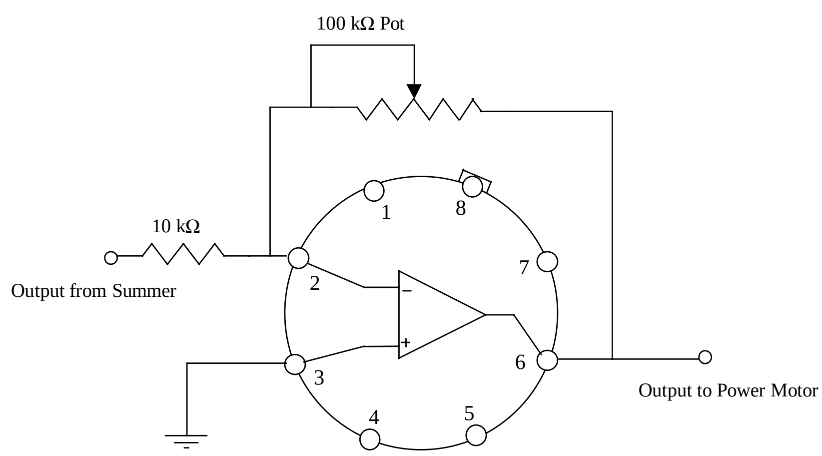


Figure 22a: Circuit diagram of power amplifier

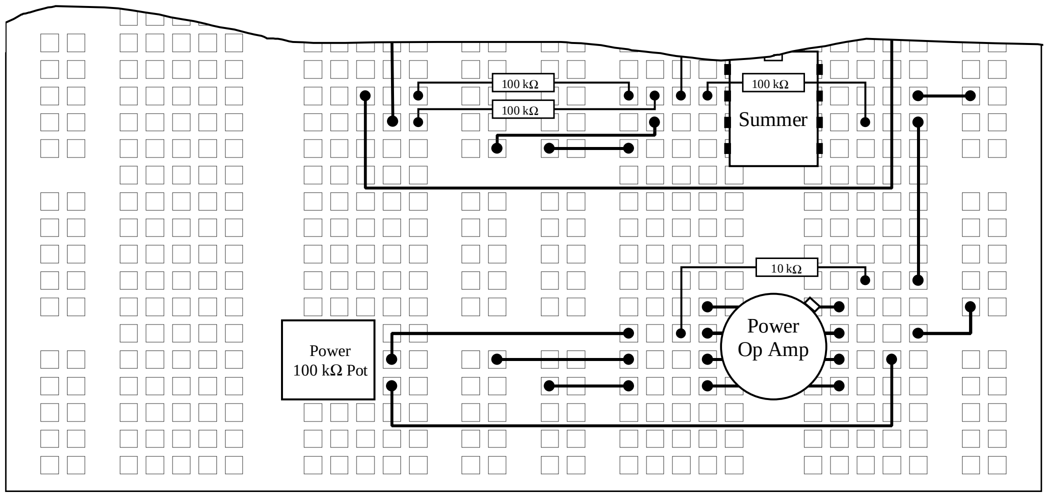


Figure 22b: Component position for power op amp

13. After installing the power op amp, place the heat sink (black, metal fin) over it. Then connect the positive lead (usually red) of the cooling fan to +15 volts, and the other lead (usually black) to ground. The fan will now run whenever you power the controller, so be careful not to have anything interfering with the blades (especially your fingers). Position the fan so it provides a steady stream of air over the power op amp.
14. You are now ready to connect a horizontal pendulum to your controller (see Fig. 23). Replace the process variable pot (the first pot) with the ten turn pot that is geared to the motor on your pendulum. Physically remove the process variable pot from the board, and insert the wires from the ten turn pot into the locations of the pins you removed. The orange wire is connected to the center terminal, the red wire (from the top terminal of the pot) to +15 volts, and the black wire (from the bottom terminal of the pot) to ground. Now, you will test to insure that the pot is functioning correctly the same way you did before. The voltage at the output of the process variable op amp should increase as the ten turn pot is turned clockwise. If this is not happening, try switching the two outside wires.

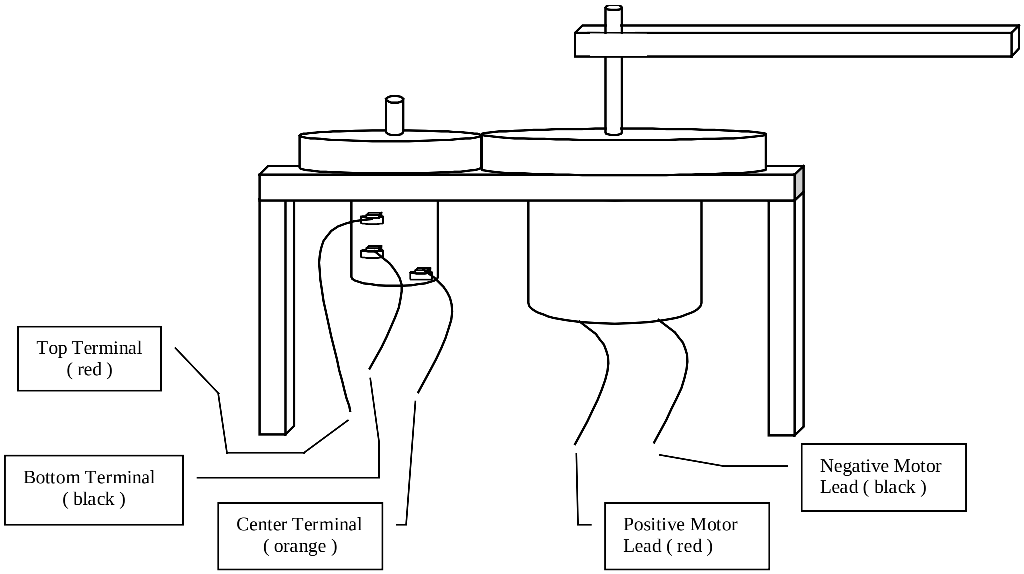


Figure 23: Horizontal Pendulum Set-up

15. Finally, connect the positive motor lead (red wire) to the output from the power op amp, and the other motor lead (black wire) to ground.
16. You are now ready to test the controller. First, connect your multimeter to the output of the error op amp. Second, turn all pots as far counterclockwise as possible. Then turn the set point pot ½ turn back. Third, turn on the power. Fourth, move the pendulum until the error is small (about ±0.5 volts). Then, turn the proportional pot about ¼ turn clockwise (the locations of the various pots can be found in figure 23). Now, slowly turn the pot that controls the power op amp clockwise until the system starts to move. As it moves watch the error. If the error is increasing, switch the red and black lead from the motor. If this does not correct the problem, check your circuit with figure 23 on the next page.
17. You can now observe the proportional control element. Without changing the proportional pot, displace the pendulum 90° and release it. Then, turn the proportional pot another ¼ turn clockwise and again displace the pendulum 90° and release it. Notice the response gets quicker as the gain (proportional pot) is increased. Also, notice the higher frequency of the response. This, of course, is because the proportional control stiffens the system. Finally, give the pot one more ¼ turn (it should now be at ¾).
18. Next, you can observe the derivative control element. With the proportional pot at ¾, displace the pendulum 90° and observe the oscillation as the pendulum returns to the set

point. Then, turn the derivative pot ¼ turn clockwise, and again displace the pendulum. Notice the response is slower, and has very few oscillations (if any). This, of course, is because the derivative element acts as a damper.

19. Finally, you can observe the integral control element. Leaving the proportional pot at ¾ turn and the derivative at ¼ turn, look at the error display on the multimeter. (it is likely to be close to, but not exactly zero). Now, increase the integral pot to about ¾ turn. This should slowly drive the error term to zero.

The component positions for the entire controller are given on the next page in Fig. 24. If you are having problems, you can use it to trouble shoot your work.

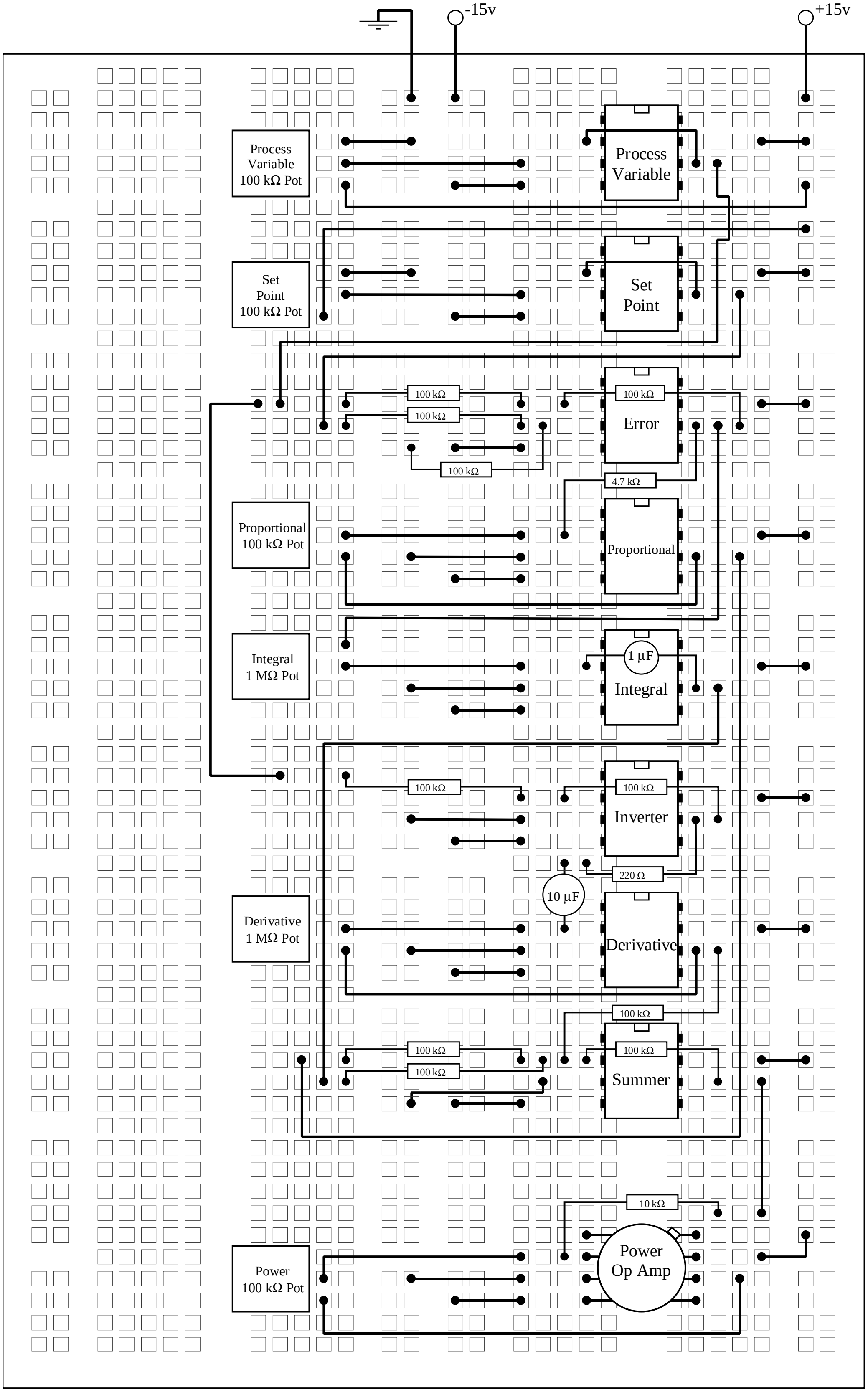


Figure 23: The complete controller

PID Parts List

Components

Jameco
1355 Shoreway RD.
Belmont, CA 94002-4100
Phone: 1-800-831-4242

Part Disc	Order No	Quantity (1 unit)	Unit Price	Total	Unit Price (50 units)	Total No (50 units)	Total Cost (50 units)
741 Op Amp	24539	8	0.35	2.8	0.22	400	70.4
100 kW Pot	43027	4	0.89	3.56	0.59	200	94.4
1 MW Pot	42981	2	0.89	1.78	0.59	100	47.2
100 kW Resistor	29997	1 pkg	0.89/pkg	0.89	1.69/pkg(100)	5	8.45
10 kW Resistor	29911	1 pkg	0.89/pkg	0.89	1.69/pkg(100)	1	1.69
4.7 kW Resistor	31026	1 pkg	0.89/pkg	0.89	1.69/pkg(100)	1	1.69
220 W Resistor	30470	1 pkg	0.89/pkg	0.89	1.69/pkg(100)	1	1.69
DC Cooling Fan	75361	1	7.95	7.95	7.95	50	397.5
Wire Jumper Kit	19289	1	9.95	9.95	8.95	50	447.5
Proto Board	20773	1	16.95	16.95	14.95	50	747.5

Digikey
701 Brooks Ave South
PO Box 677
Thief River Falls, MN 56701

Part Disc	Order No	Quantity (1 unit)	Unit Price	Total	Unit Price (50 units)	Total No (50 units)	Total Cost (50 units)
Power Op Amp	LM759CH-ND	1	6.15	6.15	2.94	50	147
Heat Sink	HS217-ND	1	0.3	0.3	0.274	50	13.7
1 mF Capacitor	P1132-ND	1	0.28	0.28	0.239	50	11.95
10 mF Capacitor	P1136-ND	1	0.62	0.62	0.534	50	26.7
Total Component Cost				**53.9**			**2017.37**

Set-up Equipment

Jameco							
Digital Multimeter	119212	1	19.95	19.95	17.95	6	107.7
6 Piece Gear Set	131801	1	3.95	3.95	3.95	6	23.7
DC Motor	112521	1	1.29	1.29	1.29	6	7.74
Digikey							
10 turn 100 kW Pot	73JA104-ND	1	21.11	21.11	21.11	6	126.66
Erector Set (approximate)				50			100
Power Supply (approximate)				30			500
Total Set-up Cost				**126.3**			**865.8**
Total Cost				**180.2**			**2883.17**