Laboratory Manual for Physics 305

Electronics

Physics Department

University of Northern British Columbia

November 21, 2002

Contents

List of Figures

Experiment 1

AC Circuits and Filters

1.1 Introduction

Circuits that contain capacitors and resistors that are driven by an AC voltage source are very useful and can be used to shape signals and filter certain frequencies. In this experiment we will learn how to measure AC voltages and study phase relations in the various forms of RC circuits. We will also study low and high pass filters as well as band pass filters the are formed by combing low and high pass filters. This experiment consists of the following four parts:

Part I: Measuring AC voltages in RC circuits

Part II: Low Pass filters

Part III: High Pass filters

Part IV: Bandpass filters

1.2 Equipment & Components

1. Digital multimeter (DMM)
2. Function Generator
3. Oscilloscope
4. (2) 10 $k\Omega$ Resistors
5. (1) 2.7 $M\Omega$ Resistor
6. (2) 0.01 μF Capacitors
7. (1) 1 μF Capacitor

1.3 Procedure

A DMM can measure AC voltages rather well within a certain frequency range. Each DMM has a range of frequency for which voltages can be measured with good accuracy. Check the frequency range of the DMM you are using and make sure that you use it only (unless you are told otherwise) if the frequency of the voltage you are measuring is within that range.

1.3.1 Part I: Measuring AC voltages in RC circuits

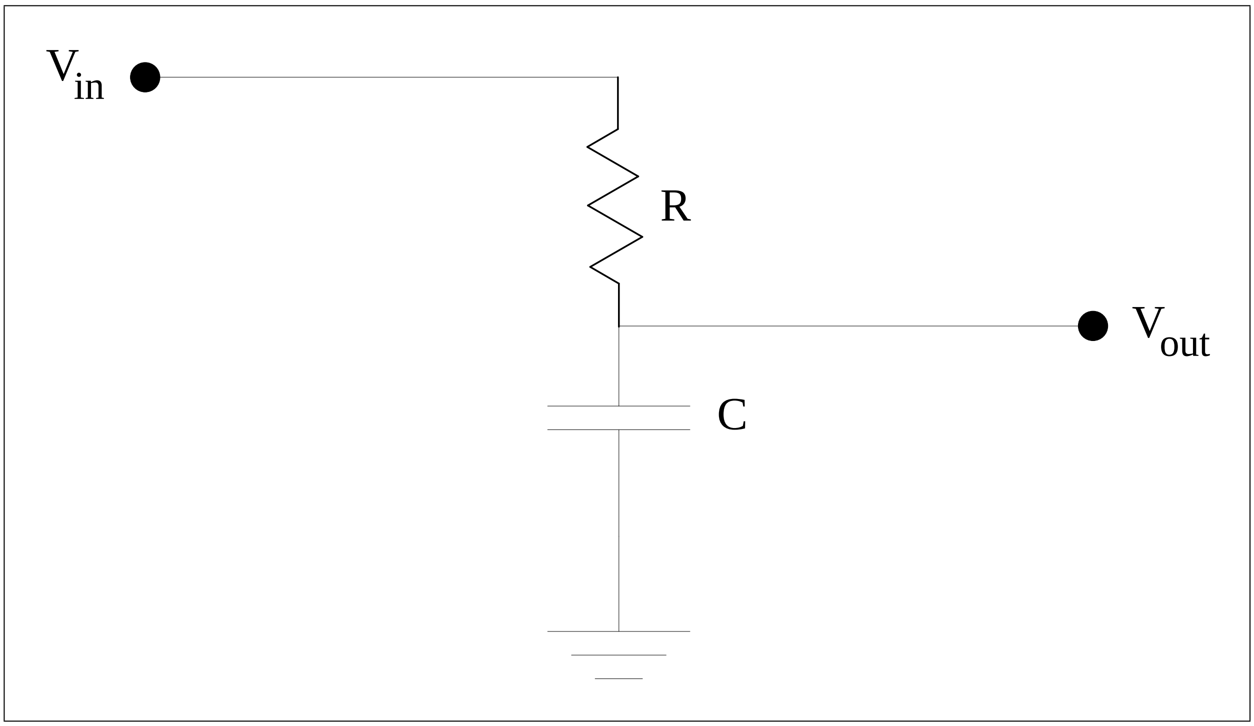

Figure 1.1: Low pass filter

Build the circuit shown in Figure (1.1). Use R = 2.7 $k\Omega$ and C = 1 μF. Set the function generator to a sine wave with a frequency of 60 Hz. Connect the function generator to an oscilloscope, adjust the voltage of the function generator to 5 V peak-peak. Connect the function generator to the circuit's input. Measure the input voltage V_{in}, the voltage across the resistor V_R and the voltage across the capacitor V_C, using a DMM in the voltmeter mode. Record your results and answer the following questions:

1. *Is V_{in} what you expected? Explain.*

2. *According to Kirchoff's Voltage Law, $V_{in} = V_R + V_C$. Does this hold true for the DMM data? Why or why not?*

Change the frequency of the function generator to 120 Hz and measure V_{in}, V_R and V_C again using the DMM as a voltmeter. Record your results and answer the following question:

3. *If your data are different for the 120 Hz signal, explain what is happening.*

1.3.2 Part II: Low Pass filters

The circuit of Figure (1.1) is a low pass filter. Use R = 10 $k\Omega$ and C = 0.01 μF. Use the oscilloscope to measure V_{out}. Vary the sine-wave frequency from 10 Hz to 1 MHz in intervals (10, 100, 1000, etc.) and measure V_{out} as a function of frequency, f. To measure the phase angle between V_{in} and V_{out} connect each voltage to one of the input channels of the oscilloscope and view both channels simultaneously. By observing the time difference t between the two signals one can find the phase angle ϕ in degrees from:

$$\phi = \frac{t \times 180}{T} = t \times f \times 180 \tag{1.1}$$

where T and f are the period and frequency of the signals respectively. Tabulate your results and plot V_{in} and V_{out} as a function of frequency f on same graph. On a separate graph plot ϕ as a function of f. Answer the following questions:

4. *calculate the break point frequency from:*

$$f_B = \frac{1}{2\pi RC} \tag{1.2}$$

   Estimate the experimental value of f_B from the graphs. Calculate the errors in the theoretical and experimental values of f_B given the accuracy of the oscilloscope is about 3%, resistor tolerance is about 5% and the capacitor tolerance is about 10%. Now compare the two values of f_B and comment on the comparison.

5. *Explain why the phase angle ϕ between V_{in} and V_{out} changes with frequency.*

6. *Why is this circuit called a low pass filter?*

   Set the frequency of the function generator to 1 kHz and adjust the DC offset to both positive and negative and measure V_{out} in each case. Does V_{out} behave as you would expect?

1.3.3 Part III: High Pass filters

A high pass filter is identical to a low pass filter except that the output voltage is taken across the resistor instead of the capacitor. One can then construct a high pass filter by switching the resistor and capacitor in the circuit of Figure (1.1) using the same components as in the low pass filter. Vary the sine-wave frequency as 10, 100, 1k, 10k, 100k and 1MHz and measure V_{out} as a function of frequency, f. Measure the phase angle ϕ between V_{in} and V_{out} as a function of frequency as described in Part II. Once again plot V_{in} and V_{out} versus f on the same graph and ϕ versus f on a separate graph. Answer the following questions:

7. *calculate the break point frequency from:*

$$f_B = \frac{1}{2\pi RC} \tag{1.3}$$

Estimate the experimental value of f_B from the graphs. Calculate the errors in the theoretical and experimental values of f_B given the accuracy of the oscilloscope is about 3%, resistor tolerance is about 5% and the capacitor tolerance is about 10%. Now compare the two values of f_B and comment on the comparison.

8. *Explain why the phase angle ϕ between V_{in} and V_{out} changes with frequency.*

9. *Why is this circuit called a low pass filter?*

Set the frequency of the function generator to 1 kHz and adjust the DC offset to both positive and negative and measure V_{out} in each case. Does V_{out} behave as you would expect?

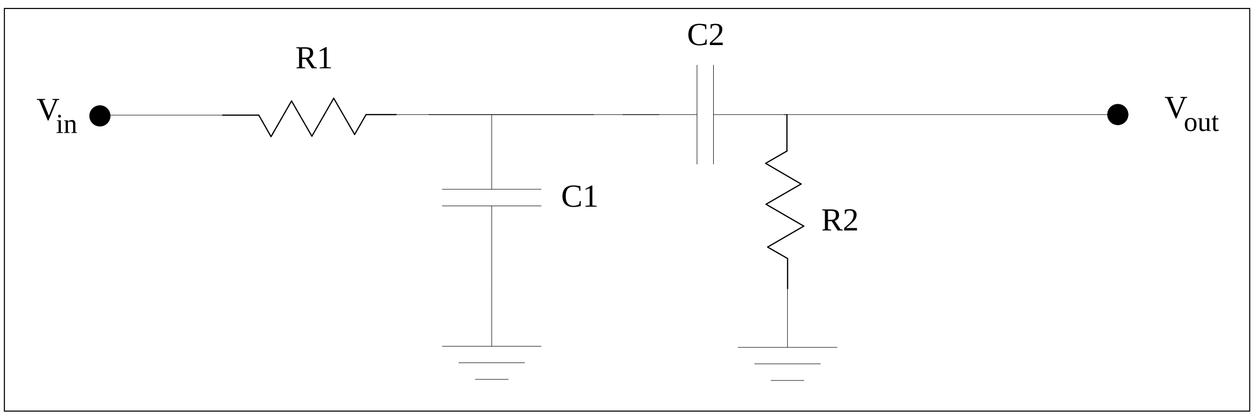

Figure 1.2: Band pass filter

1.3.4 Part IV: Bandpass filters

A high pass filter passes high frequencies almost unaffected and highly attenuates low frequencies. A low pass filter passes low frequencies almost unaffected while highly attenuated high frequencies.

In some application, one needs a filter to attenuate high and low frequencies leaving a band of frequencies in between unaffected. such a filter is called a band pass filter. it is possible to construct a band pass filter by connecting a high pass and low pass filters together as shown in Figure (1.2).

Set $R_1 = R_2 = 10k\Omega$, and $C_1 = C_2 = 0.01\ \mu F$. Follow the steps of Part II or Part III, and answer the following question:

10. *Describe quantitatively how would the V_{out} vs. frequency graph change if R_2 were decreased to 1 $k\Omega$. How would the graph change if C_1 were increased to 0.1 μF ?*

Experiment 2

RLC Circuits

2.1 Introduction

In this experiment we introduce the role an inductance plays in an RC circuit. Inductance plays an important role in high frequency circuits. They also play an important role in resonance circuits, band pass filters and notch filters. In addition, stray inductance and capacitances often occur in circuits and causes the signal to go through unwanted oscillations or “ringing”. In many circuits stray capacitances and inductances degrade signals and disrupt the performance of the circuit. It is then important to understand these effects and lear to recognize them. The stray capacitances and inductances are produced by wires, cables and circuit boards. The ringing or damped oscillations are produced when the signal has sharp transitions, like for example a square wave signals. The oscillations produced by circuits containing theses element are very useful in building oscillators that are used in function generators.

Part I: RLC Transients - Ringing

Part II: RLC Bandpass Filter

Part III: RLC Notch Filter

2.2 Equipment and Components

1. Oscilloscope
2. Function Generator
3. (1) $22\mu F$ Capacitor
4. (1) 1 nF Capacitor
5. (1) 1 $k\Omega$ Resistor

6. (1) 100 Ω Resistor
7. (1) 1 mH Inductor

2.3 Procedure

2.3.1 Part I: RLC Transients - Ringing

Use the circuit shown in Figure (2.1) with $R_1 = 1k\Omega$, $C_1 = 1\ nF$, and $L_1 = 1\ mH$.

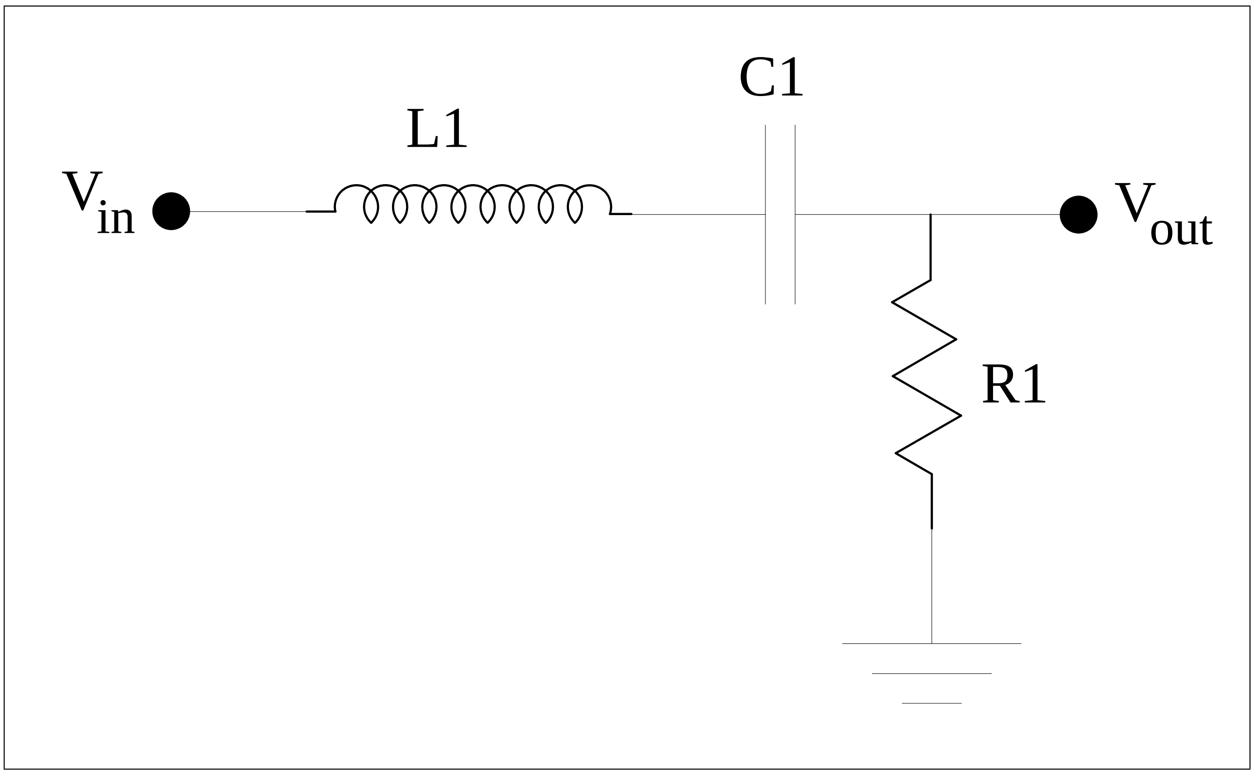

Figure 2.1: RLC Circuit as a Band Pass Filter

Calculate the the natural frequency $f_{\circ c}$ of the circuit from:

$$f_{\circ} = \frac{1}{2\pi\sqrt{LC}} \tag{2.1}$$

using $f_{\circ c}$ calculate the period $T_{\circ c}$ of this frequency.

Set the function generator to produce a square wave with a frequency of about $0.1 \times f_{\circ}$. Observe the output on the oscilloscope. The output at the transition points of the input square wave will exponential. Change the the input frequency slight until you can see the exponential decay. Measure the period of the output signal T_{om} and ues to calculate the frequncy f_{om}. Answer the following questions:

1. *Compare f_{oc} and f_{om}. Is there any discrepancy? Explain.*
2. *One can show that the the exponential decay of the ringing signal is described bu $e^{-R/2L}$, does this agree with your data?*
3. *explain what is happening in the transition region in terms of the time dependence of current, charge and voltage.*

2.3.2 Part II: RLC Bandpass Filter

In Experiment 1, we studied an RC bandpass filter. Here we study an RLC bandpass filter. Use the circuit shown in Figure (2.1) with $R_1 = 100\Omega$, $C_1 = 22\mu F$, and $L_1 = 1\ mH$.

Calculate the resonance frequency f_{oc} of the circuit using Equation (2.1). Set the function generator to a sine wave of 5 V peak-peak. Measure V_{out} and the phase angle ϕ as a function of frequency and plot them as you did in Experiment 1. Answer the following questions:

4. *Determine f_{om} from the graphs and compare with the calculated valuef_{oc}. Is there any discrepancy? Explain*
5. *Explain how a band pass filter works.*

2.3.3 Part III: RLC Notch Filter

A notch filter is the opposite of a band pass filter. It passes all frequencies almost unattenuated and heavily attenuates a specific band of frequencies, "the notch frequency". This can be achieved by connecting the inductor and capacitor in parallel with each other as shown in Figure (2.2). The parallel inductor and capacitor circuit is often referred to as a "tank circuit".

Set the function generato to a sine wave with 5 V peak-peak. Measure V_{out} and the phase angle ϕ as a function of the frequency and plot them as you did in Experiment 1. Answer the following questions:

6. *Derive an expression for the notch frequency for the circuit of Figure (2.2) and calculate it.*
7. *determine the notch frequency from the graphs and compare it to the calculated one. Explain any discrepancy.*

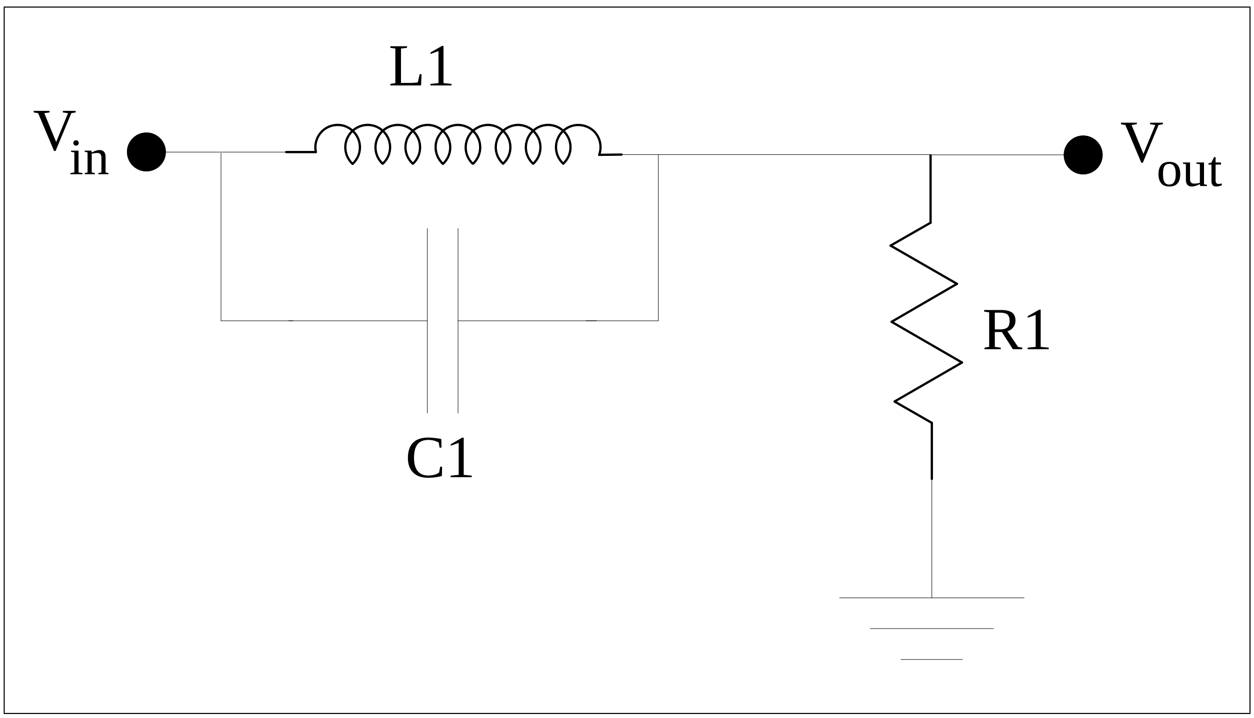

Figure 2.2: RLC Notch Filter

Experiment 3

Rectification and Filtering Using Diodes

3.1 Introduction

The circuits we have been studying so far are linear circuits composed of linear elements, like resistors, capacitors and inductors. Linear elements and circuits exhibit linear relationship between currents and voltages. Diodes are two-terminal devices that are not linear, i.e. the current does not change linearly with the voltage. Moreover, the diode behaves like a good conductor when the current flows in one direction and as an insulator when the current flows in the opposite direction. In other words, the diode conducts well when one terminal is at higher potential than the other and it conducts poorly when the potential is reversed. In this experiment we will make use of this property of the diode.

This experiment consists of following parts:

Part I: Rectification Using a Single Diode

Part II: Filtering

Part III: Bridge Full-Wave Rectifier

Part IV: Zener Diode

Part V: Diode Clipper

3.2 Equipment & Components

1. Oscilloscope
2. Function Generator

3. (1) Resistor Substitution Box
4. (2) $1k\Omega$ Resistors
5. (1) 100Ω Resistor 1/2 W
6. (4) 1N914 Diodes or equivalent
7. (1) 1N4733A Zener Diode or equivalent
8. (1) Isolation Transformer
9. (4) 1 μF Capacitor
10. (1) 100 μF Capacitor

3.3 Procedure

3.3.1 Part I: Single Diode Rectification

One of the main uses of diodes is in rectification of an AC signal to produce a DC signal. The circuit shown in Figure (3.1) is called. You will discover later why it is called so. Identify the direction of the diode as shown in Figure (3.2). Build the circuit and drive it using a 5V peak-to-peak sine wave signal with a frequency of 1 kHz provided by the function generator. Sketch the input and output signals. Repeat for a square wave centered at zero volts.

1. *Do your readings support the rule that a conducting diode has a voltage drop of about 0.6 V? Explain.*

Change the input to a sine wave and reduce its amplitude and reverse the direction of the diode. Sketch the signals.

2. *Is the output as expected?*

While reducing the amplitude of the input signal observe the amplitude of the output.

3. *Record the input voltage at which the output voltage go to zero volts over all time? Is it about 0.6V? Explain.*

Return to a sine wave input of 5 V peak-to-peak and vary the current through the circuit by changing R_1 from 10 Ω to 1 $M\Omega$ in decade increments. Plot the peak-to-peak output voltage as a function of the resistance.

4. *Does the current in the circuit strongly affect the voltage drop across the diode? Explain.*

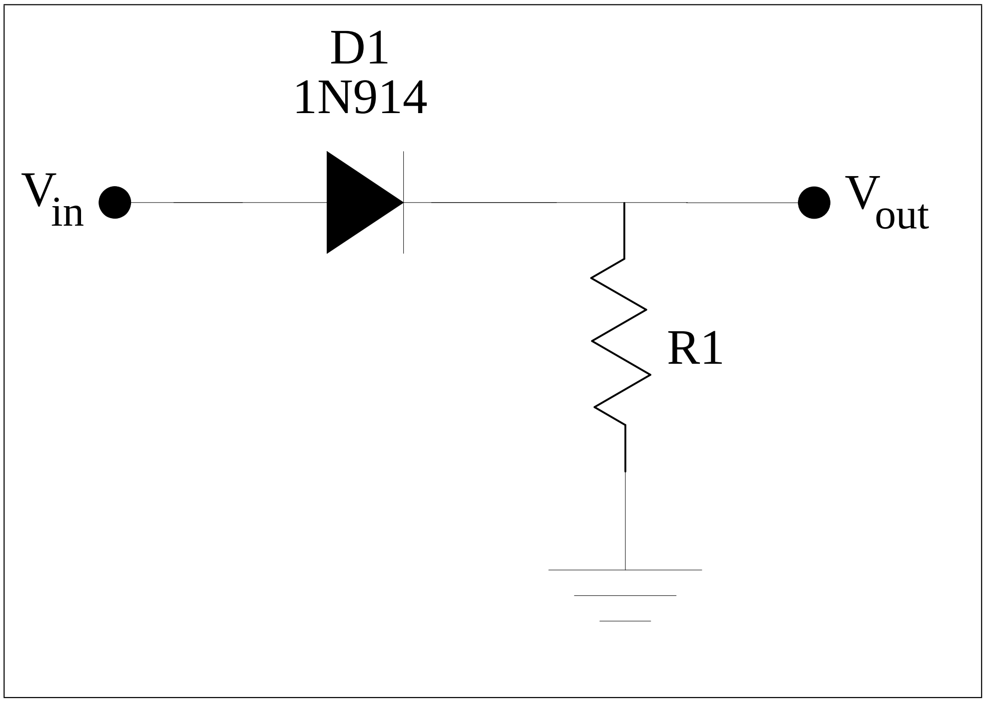

Figure 3.1: Half-wave rectifier

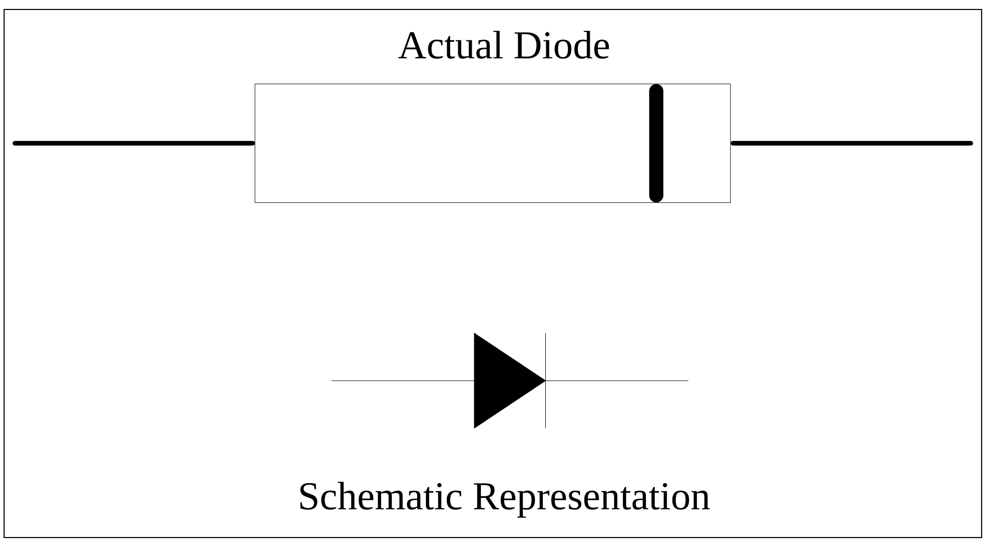

Figure 3.2: Diode orientation

Build the circuit shown in Figure (3.3) and drive it with a 10 kHz square wave of 10 V peak-to-peak amplitude. This circuit is called rectified differentiator. Sketch the input and output signals.

5. *Is the output as expected? Why is the circuit called rectified differentiator? Explain how the circuit works.*

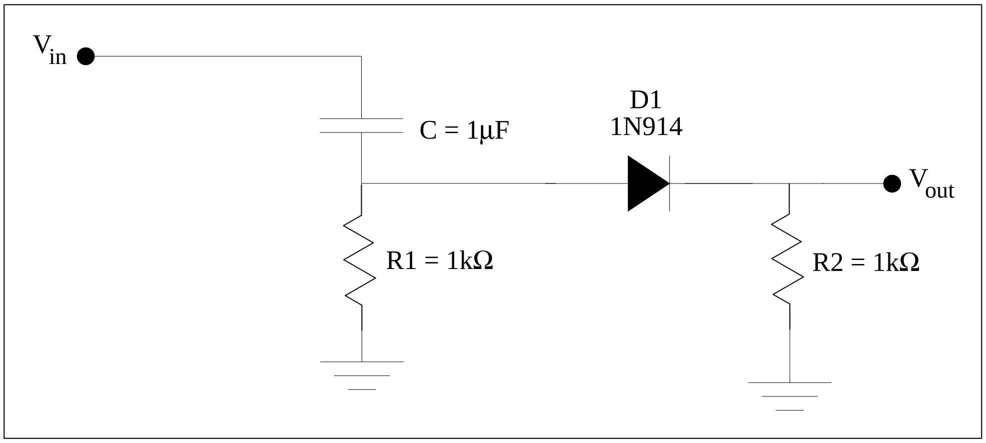

Figure 3.3: Rectified Differentiator

Reverse the direction of the diode in the circuit of Figure (3.3).

6. *Is the output as expected?*

3.3.2 Part II: Filtering

The half wave rectifier converts AC to DC. However, the DC in this case is not a pure DC signal. It goes in on directions as a proper DC should but the magnitude varies with time. This variation is called "ripple". The ripple is measured by the "ripple factor r" given by:

$$r = \frac{\Delta V}{V_{peak}} \tag{3.1}$$

where ΔV is the peak-to-peak variation and V_{peak} is the peak value of the rectified signal. For a half wave rectifier, the voltage changes from zero to the peak value of the input AC signal, i.e. $\Delta V = V_{peak}$, so $r = 100\%$. It is then important to reduce the ripple factor. An ideal DC voltage has no ripple, i.e. $r = 0$. One way of reducing the ripple factor in a half wave rectifier is add add a filtering capacitor as shown in the circuit of Figure (3.4).
Construct the circuit shown in Figure (3.4) using:

- 20 V peak-to-peak input signal with a frequency of 1 kHz.
- C = 1 μF , and
- $R_1 = 1\ k\Omega$

Calculate the ripple factor. Use a larger capacitor of say 100 μF, then measure the ripple factor. Explain the difference. The ripple factor can be approximated by:

$$r \approx \frac{i}{V_{peak} C f} \tag{3.2}$$

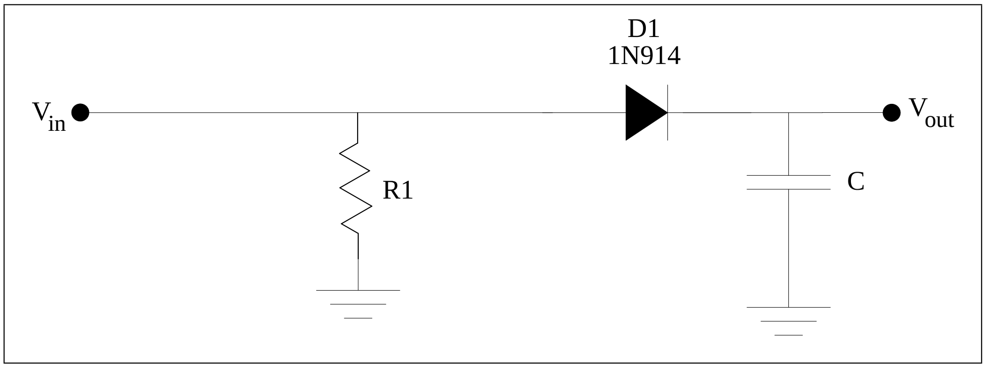

Figure 3.4: Filtered half wave rectifier

7. *Equation (3.2) is an approximation. Dose it reasonably predict the ripple factor for the two above measurements?*

Vary the input frequency in decades (10 Hz, 100 Hz, 1 kHz, 10 kHz, 100 kHz, 1 MHz). Measure ΔV, V_{peak}, and i at each frequency. Calculate the experimental and theoretical ripple factors at each frequency. Tabulate your results.

8. *Does the theoretical and experimental ripple factors agree over the frequency range? Discuss.*

Vary the current i by varying R_1 in decades from 10Ω to $1M\Omega$. Once again measure ΔV, V_{peak}, and i at each case. Calculate the experimental and theoretical ripple factors . Tabulate your results.

9. *Does the theoretical and experimental ripple factors agree over the frequency range? Discuss.*

3.3.3 Part III: Bridge Full-Wave Rectifier

In general, large capacitors are expensive. One way to double the frequency of the to-befiltered DC signal, and thus half the ripple, is through the use of a bridge rectifier. The bridge rectifier adds 3 more diodes to the circuit, but these are often less expensive than a twice-as-large capacitor, particularly when relatively high currents are involved. The bridge rectifier requires a floating ground to operate properly. Since the function generator ground is referenced to the third prong of its line power, we establish a floating ground by adding a 1:1 transformer as shown in Figure (3.5). The output is across the load resistor or points A and B. Construct the circuit shown in Figure (3.5) using $R = 1k\Omega$ and a driving frequency of 100 Hz, 10 V peak-to-peak sine wave. Sketch the input and output voltages.

10. *Explain how the bridge rectifier works.*

Change the input to a square wave and sketch the input and output voltages.

11. *Is the full wave rectified square wave output a pure DC? Explain*

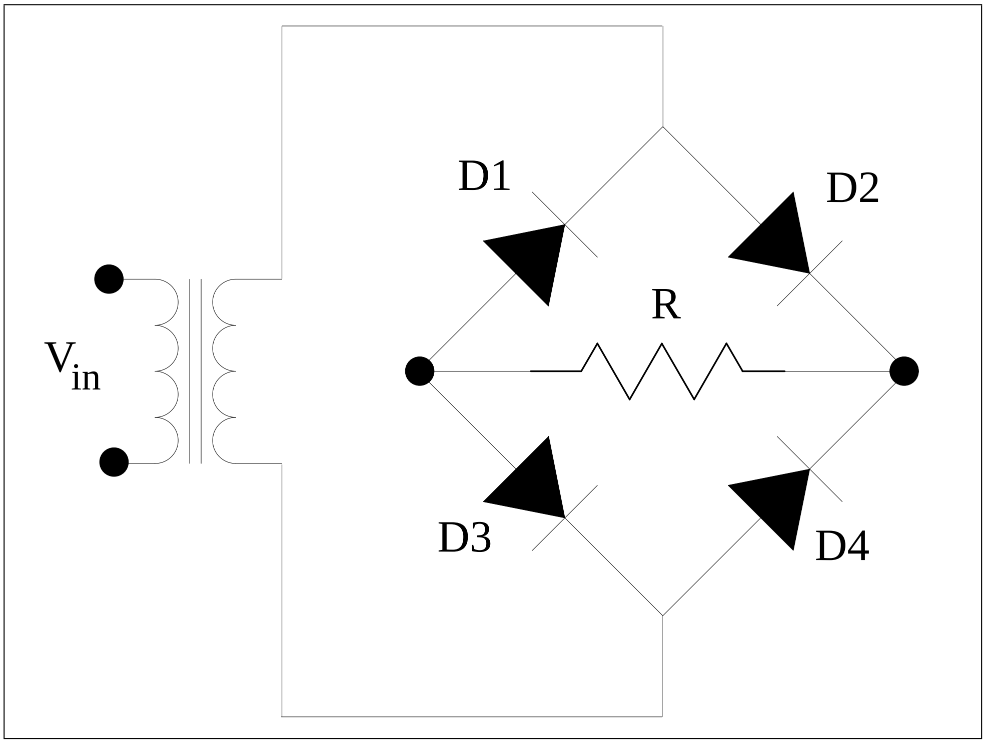

Figure 3.5: Bridge full wave rectifier

3.3.4 Part IV: Zener Diode

Zener diodes are used quite differently than the other diodes we have used (signal 1N914 and power 1N4001). Zener diodes are used as simple voltage regulators generally in low current, non-precise applications. When used properly they act to reduce a DC voltage to a given, regulated DC voltage. Their use is fairly straight-forward as shown in Figure (3.6). Resistor R limits the current through the circuit and RL represents the load resistance the regulator is driving.

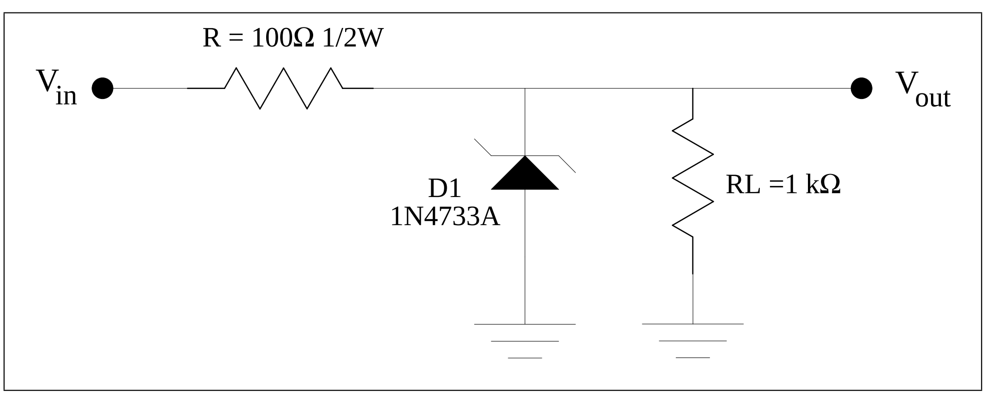

Figure 3.6: Voltage regulator using a Zener diode

Construct the circuit shown in Figure (3.6). Use $V_{in} = 12V$ DC. Measure $Vout$ using a DMM voltmeter. Measure the voltages across each component and calculate the current through each.

3.3.5 Part V: Diode Clipper

The diode clipper is often used to limit a signal. Construct the circuit in Figure (3.7) and drive it with a 20 V peak-to-peak sine wave. Show that the output is limited to approximately $V_L + V_{diodedrop}$ by. Sketch the input and output signals.

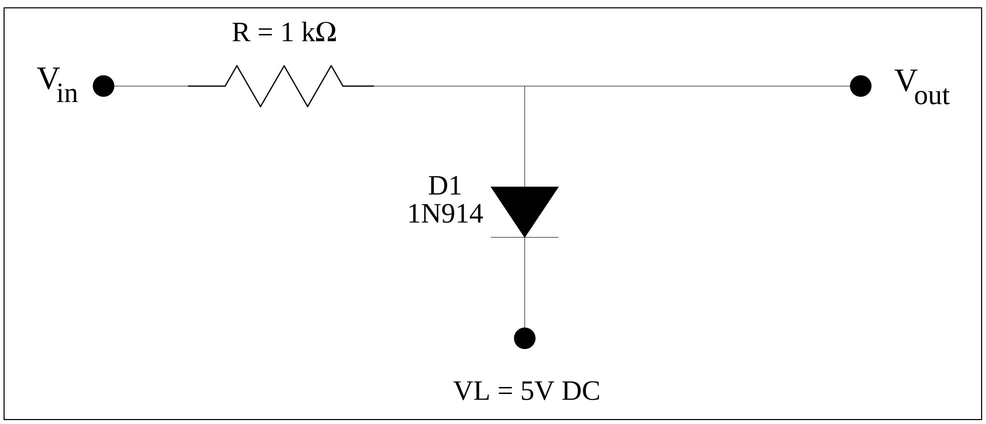

Figure 3.7: Diode clipper circuit

Design and construct a circuit that will limit the positive half of a 20 V peak-to- peak sine wave to 5.6 V and the negative half to -2.5 V. Sketch the circuit below and the resulting input and output signals.

Experiment 4

DC Power Supplies

4.1 Introduction

The majority of instrumentation related circuits are powered from 120 V, 60Hz line voltage in Canada and United States. In many other countries they use 220 V, 50 Hz line voltage, but the design and construction of DC power supplies remains identical with the difference being in the transformer used. In this lab we will investigate a variety of methods for designing and constructing regulated DC power supplies. There are three parts to this laboratory as follows:

Part I Fixed Single Voltage DC Power Supplies

Part II Bipolar Fixed DC Power Supplies

Part III Adjustable DC Power Supplies

4.2 Equipment & Components

1. Oscilloscope
2. Function Generator
3. Resistor Substitution Box
4. (4) 1N4001 Diodes or equivalent
5. (1) 25.2 V Center-tapped Power Transformer or equivalent
6. (1) LM7805
7. (1) LM7905
8. (1) LM317LZ

9. (2) 100μF Capacitors
10. (2) 0.1μF Capacitors
11. (1) 220Ω Resistor
12. (2) 270Ω Resistors
13. (1) 1kΩ Resistor

4.3 Procedure

All circuits in this lab will make use of a 25.2 V center-tapped power transformer driven by line voltage (it plugs into the wall). When the schematic indicates 12.6 V, this implies that only the center secondary lead and one end secondary lead are used. Make sure you do not ground the remaining lead or allow it touch other components.

CAUTION: Line voltage is dangerous. There should be no exposed 120 V connections, unless otherwise unavoidable. Make certain that the primary of the transformer and line power cord have no exposed connections. No exceptions to this rule are allowed.

4.3.1 Part I: Fixed Single Voltage DC Power Supplies

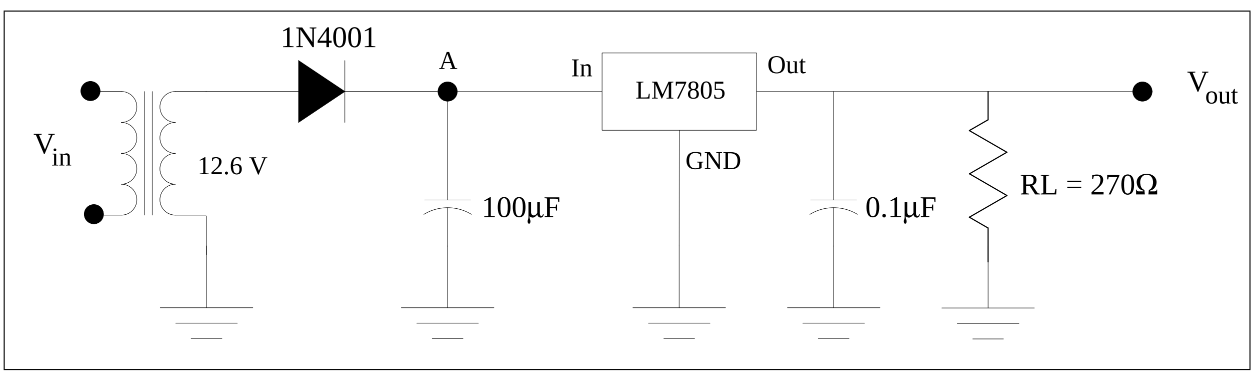

Figure 4.1: Fixed 5.0 V Power Supply

Years ago, before the advent of modern integrated circuits, DC power supply design and construction was rather tedious. Today it is a simple matter of a few calculations and making a choice for a suitable DC voltage regulator IC. In this part of the lab, we will design and construct a fixed 5.0 V power supply using the LM7805 voltage regulator IC and then do the same for a fixed -5.0 V power supply using the LM7905 voltage regulator IC.

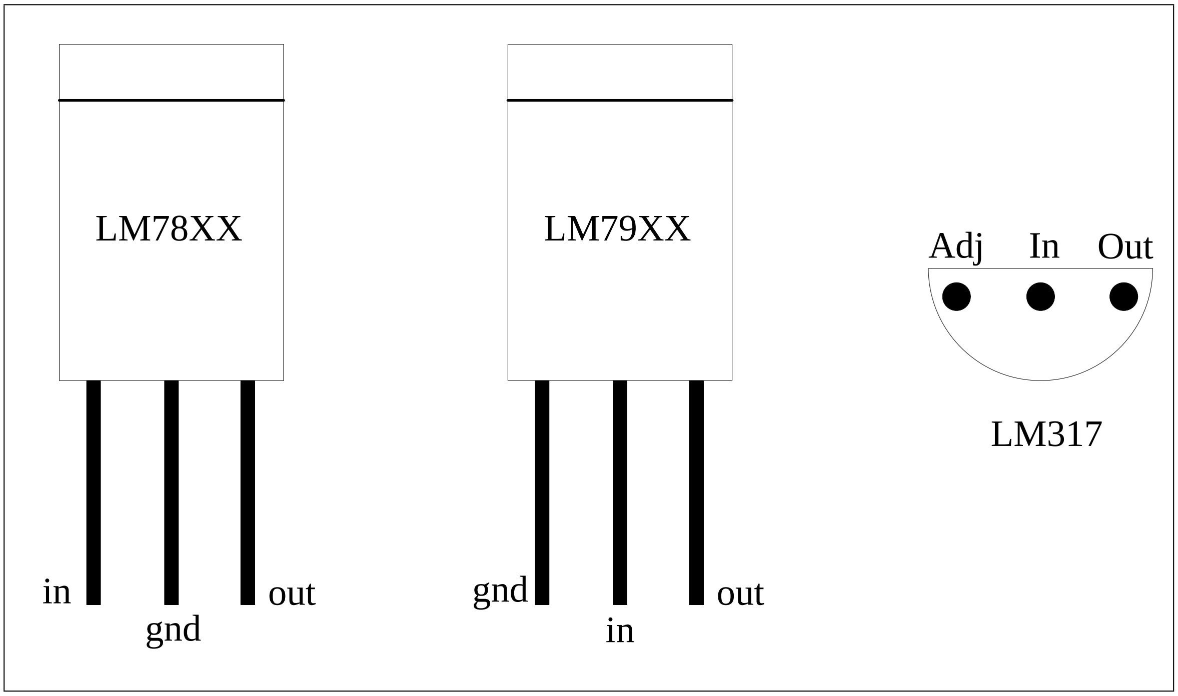

Figure 4.2: Pin assignments for LM78XX, LM79XX and LM317

Figure (4.1) shows a simple fixed 5.0 V power supply producing 19 mA into load resistor R_L. The front end of the circuit consists of a single diode to rectify the signal and a capacitor to filter the rectified signal. The voltage regulator provides regulation and a small capacitor in parallel with the output is included for stability. While this capacitor is sometimes not necessary (when the load is located near the regulator) it is wise to always include it in your design.

Construct the circuit shown in Figure (4.1) noting the pin-out labels of the LM7805 shown in Figure (4.2). If you connect it wrong, chances are it will get very hot and not work. Be careful! Using the oscilloscope, measure the AC and DC voltages at points A and V_{out}. Calculate the ripple factor in percent for the output voltage.

1. *Is the ripple on the output in line with that specified in the data sheet? Is the output voltage within the range specified by the data sheet? Explain.*
2. *Is the ripple voltage at point A in line with that which is expected? Explain.*

Vary the load resistor R_L and note any change in ripple voltage on the output. Is the LM7805 an adequate regulator?

Now construct the negative version of the fixed regulated supply you just completed using an LM7905 regulator and repeat the calculations. This circuit is shown in Figure (4.3). Note the different pin assignments and don't forget to reverse the polarity of the filter capacitor.

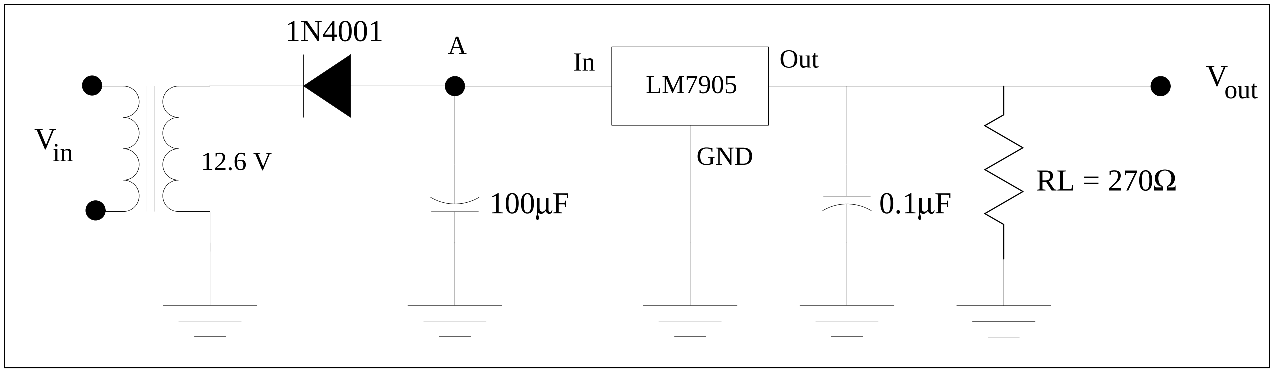

Figure 4.3: Fixed -5.0 V power supply

3. *Is the ripple on the output in line with that specified in the data sheet? Is the output voltage within the range specified by the data sheet? Explain.*

4. *Is the ripple voltage at point A in line with that which is expected? Explain.*

4.3.2 Part II: Bipolar Fixed DC Power Supplies

Often both negative and positive voltages are necessary in a circuit. We can combine the two above circuits and use the center-tap of the transformer for ground to construct a bipolar ±5 V regulated power supply. Design and construct said circuit and measure the output voltages and their respective ripples.

4.3.3 Part III: Adjustable DC Power Supplies

There are a number of ways to produce a variable DC voltage. The simplest we have already seen - it consists of a potentiometer connected between ground and some fixed voltage with the wiper supplying a variable voltage between ground and the fixed supply voltage. While this scenario works well in some applications, it is fairly limited, particularly when high currents are required and it requires a fixed regulated supply to begin with. One versatile adjustable DC voltage regulator is the LM317. It is available in a negative flavor as the LM337.

The basic LM317 adjustable voltage regulator circuit is shown in Figure (4.4). Pin-outs for the LM317LZ are shown in Figure (4.2). V_{out} is given by the ratio of R_2 to R_1 as:

$$V_{out} = 1.25\left(\frac{R_2}{R_1} + 1\right) \tag{4.1}$$

Construct the circuit shown in Figure (4.4) using the resistor substitution box for R_2 and R_1 equal to 220Ω. Leave the load resistor out of the circuit. Vary R_2 from 0 Ω to 1 $k\Omega$ in 200 Ω intervals and from 1 $k\Omega$ to 9 $k\Omega$ in 2 $k\Omega$ intervals and measure V_{out} using a DMM. Calculate the theoretical output voltages and compare to the measured.

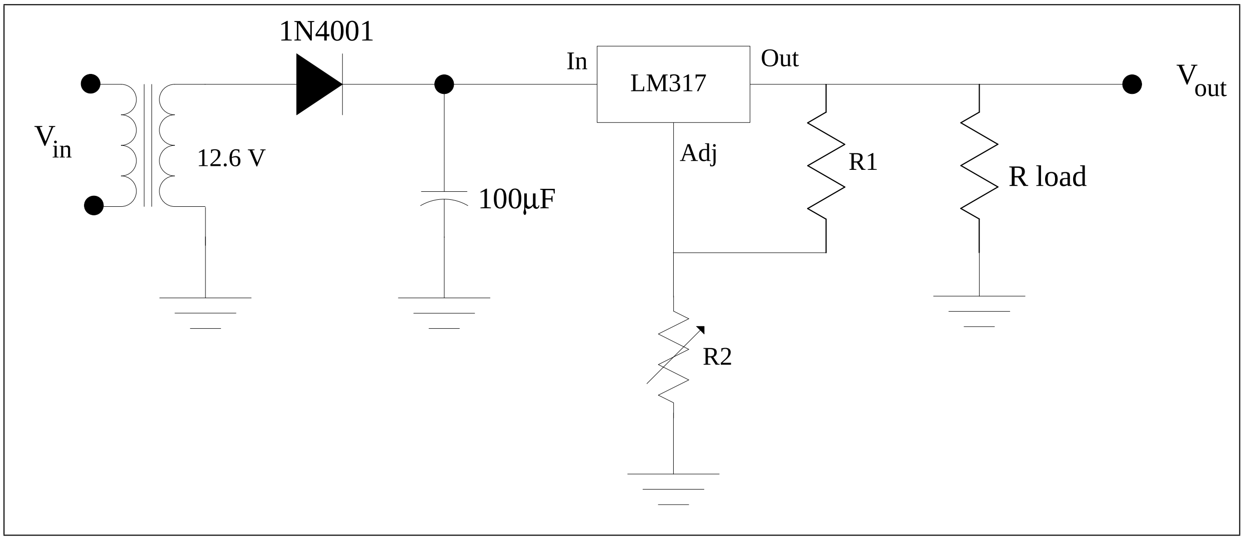

Figure 4.4: Adjustable positive voltage power supply.

5. *Explain any discrepancies between measured and theoretical values of* V_{out}.

One drawback of the LM317 is that its lower voltage limit output is about 1.2 Volts. Often one wants a variable supply to go to zero volts. Figure (4.5) gives one limited solution to this problem. Add 3 1N4001 diodes and a $1k\Omega$ resistor to your circuit as shown in Figure (4.5). Does the circuit work?

Remove the $1k\Omega$ resistor. Does the circuit's output voltage still go to zero volts?

6. *Explain the purpose of the 1 kΩ resistor in the circuit above.*

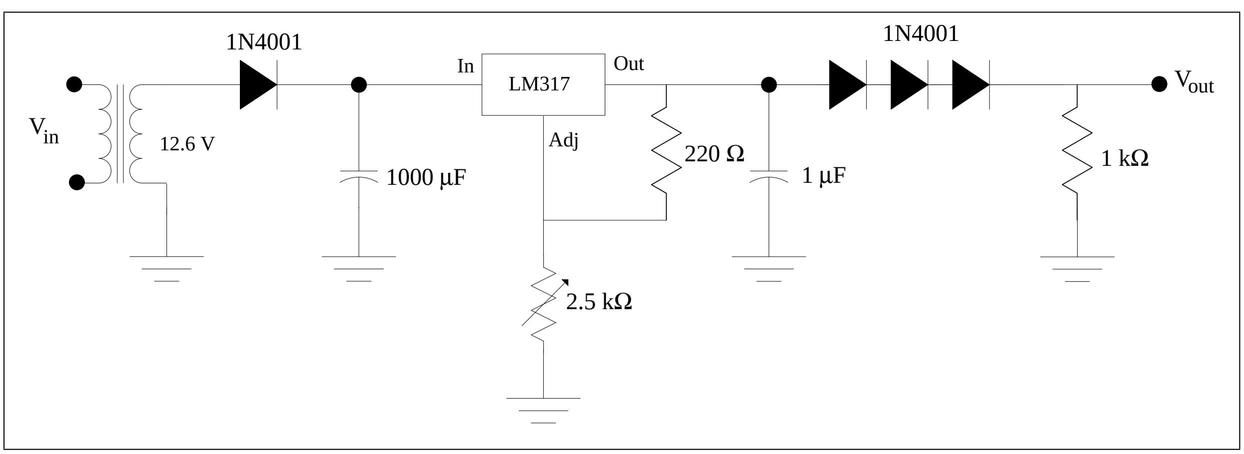

Figure 4.5: Variable DC power supply, 0-12 V.

Experiment 5

Transistors

5.1 Introduction

In the experiment we will study the use of bipolar junction transistors in various circuits. The experiment has the following five parts:

Part I Current Amplification.

Part II The Transistor as a Switch.

Part III Voltage Follower.

Part IV Common-Emitter (Voltage) Amplifier.

Part V Darlington Circuit

5.2 Equipment & Components

1. (1) Oscilloscope
2. (1) Function Generator
3. (1) Resistor Substitution Box
4. (2) 2N3904 Transistors
5. (2) 1 $k\Omega$ Resistors
6. (1) 270 Ω Resistor
7. (1) 47 $k\Omega$ Resistor
8. (1) 470 Ω Resistor

9. (2) 4.7 $k\Omega$ Resistors
10. (1) 2.7 $k\Omega$ Resistor
11. (2) 1 μF Capacitor

5.3 Procedure

5.3.1 Part I: Current Amplification

The transistor can be used as a current amplifier. To demonstrate this point build the circuit shown in Figure (5.1). This circuit uses a PNP transistor (2N3904) and two ammeters to measure the current into the transistor's base I_B and collector I_C. The emitter current I_E is then the sum of I_B and I_C.

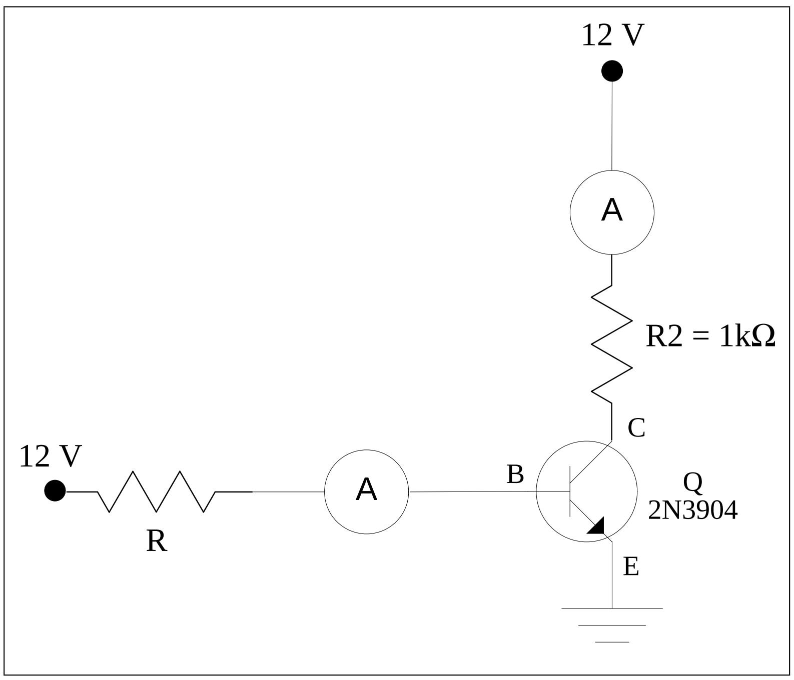

Figure 5.1: The transistor as a current amplifier

Use a substitution box for R and vary it from 1 $k\Omega$ to 10 $M\Omega$ in the following steps: $1k$, $5k$, $10k$, $50k$, $100k$, $500k$, $1M$, $5M$, $10M\Omega$. Measure I_C and I_B in each case and calculate the current gain $\beta = I_C/I_B$. *Make sure that R does get below 1 $k\Omega$ otherwise the base current gets too high raising the transistor temperature.*

1. *How does β behaves over the I_B range used in the experiment? Does it stay contant?*
2. *Set $R = 1\ M\Omega$. Touch the transistor lightly with the tip of a hot soldering iron. How does β change?*

5.3.2 Part II: The Transistor as a Switch

An important application of a transistor is to use it as a switch. If one needs to use a low current source to control a large current source, then a transistor in the switch mode is used. A low current source is one with high impedance while a high current source has low impedance. In this experiment, we are going to simulate this situation with a square wave signal from a function generator to control a light emitting diode (LED). The function generator has high internal impedance of 10 $k\Omega$, thus providing a 10 V peak-to-peak signal with a maximum of 0.5 mA. The LED requires more than 0.5 mA to turn on and produce light. A transistor is used to amplify the input current to the level required to turn on the LED as shown in the circuit of Figure (5.2).

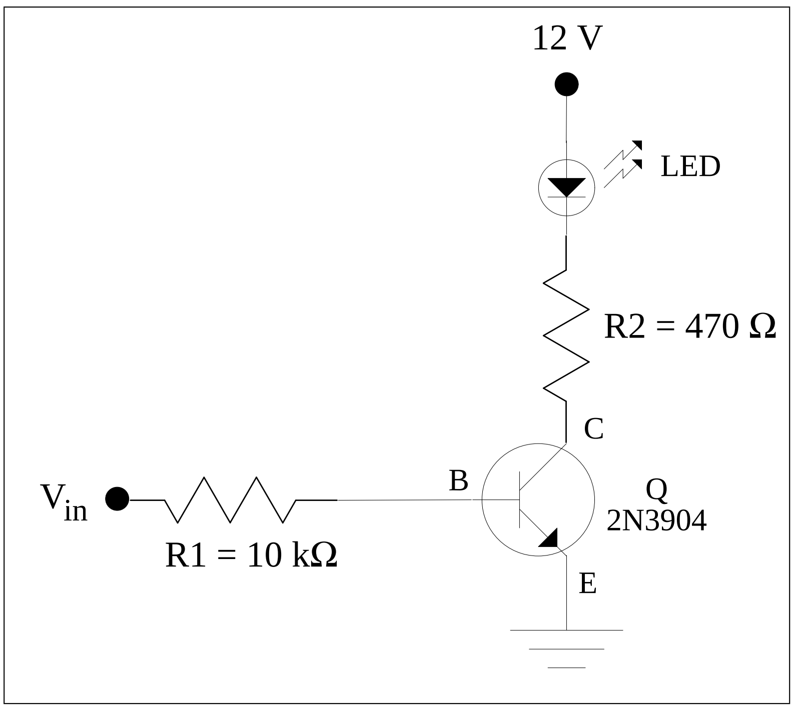

Figure 5.2: The transistor as a switch

The input is a square wave and is connected to the base of the transistor. When the input is positive the transistor conducts, when the input is negative the transistor turns off. When the transistor is on the collector produces a current that is higher than the base current and is enough to turn the LED on. When the transistor is off the base and collector currents are both zero and the LED is turned off. Build the circuit shown in Figure (5.2) and drive it with a 10 V peak-to-peak square wave signal with frequency of 1 Hz or less. Using a DMM notice when the LED is on or off.

3. *Explain the operation of the circuit. In particular why the diode does turn on when the signal is high and off when the signal is low.*

4. *Why is the 470Ω resistor is needed in the circuit?*

5.3.3 Part III: Voltage Follower

In this part we use the transistor to amplify a the current of an analogue signal while the input voltage does not change i.e. the voltage gain is close to one. The circuit shown in Figure (5.3) is a voltage follower. The input impedance of the circuit is much larger than the input impedance and thus the driving signal is buffered (not loaded down) by whatever circuitry is connected to the output. This circuit is useful when a very weak signal must drive a some device that requires more current than the signal can provide.

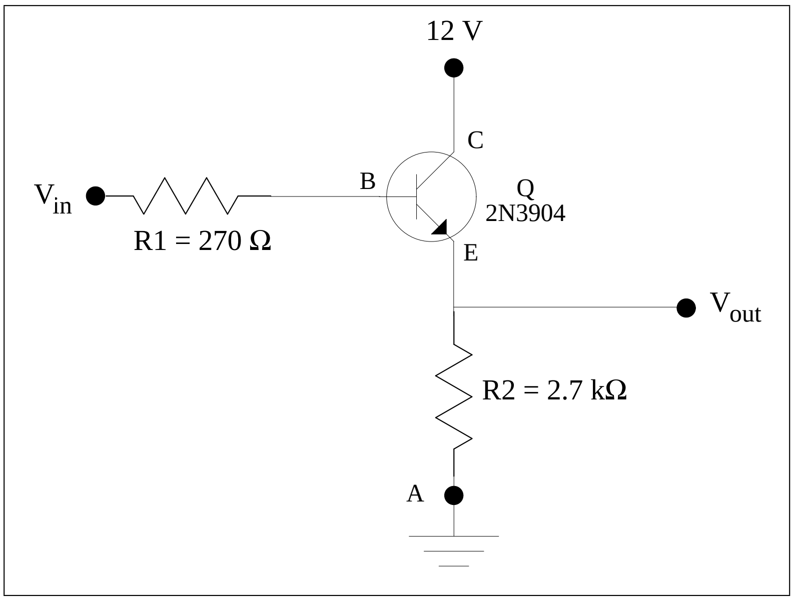

Figure 5.3: Voltage follower (Emitter follower)

Construct the circuit shown in Figure (5.3) and drive it with a 5 V peak-to-peak sine wave with frequency of 1 kHz. Sketch the input and output voltages.

5. *Is there a phase difference between the input and output?*

6. *Does the output signal replicate the input signal? Explain why oe why not in terms of the transistor's function*

5.3.4 Part IV: Common Emitter Voltage Amplifier

The transistor can be used a voltage amplifier, a common emitter amplifier is shown in Figure (5.4). Build the circuit shown in Figure (5.4) and drive with a 0.2 V peak-to-peak, sine wave

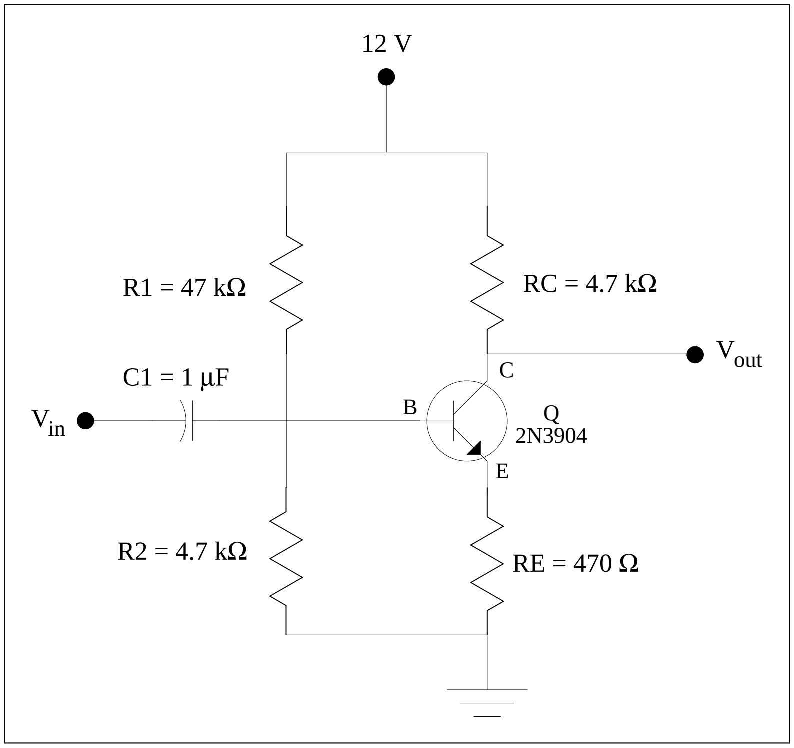

Figure 5.4: Common Emitter voltage amplifier

signal. Vary the signal's frequency from 1 Hz to 1 MHz in the following steps: 1, 10, 100, 1 k, 10 k, 100 k, 1 MHz. Plot the output voltage, voltage gain and phase between the input and the output as a function of frequency.

7. *Explain the dependence of the voltage gain on the frequency*

8. *Is it possible to change the circuit to achieve a higher gain, say of 1000? Explain how.*

9. *Does the input and output signals look the same? In other words, does the amplifier distort the input signal while amplifying it? If there is any distortion, is it avoidable?*

5.3.5 Part V: Darlington Circuit

Most single transistor amplifiers offer small current gain. It is possible to achieve higher gain by connecting two transistors as in the circuit shown in Figure (5.5) (known as Darlington circuit).

Use a substitution box for R and vary it from 1 $k\Omega$ to 1 $M\Omega$ in the same steps followed in part I. Measure I_B, I_C, and $\beta = I_C/I_B$ in each step and tabulate your results. *Once again: Make sure that R does get below 1 kΩ otherwise the base current gets too high raising the transistor temperature.*

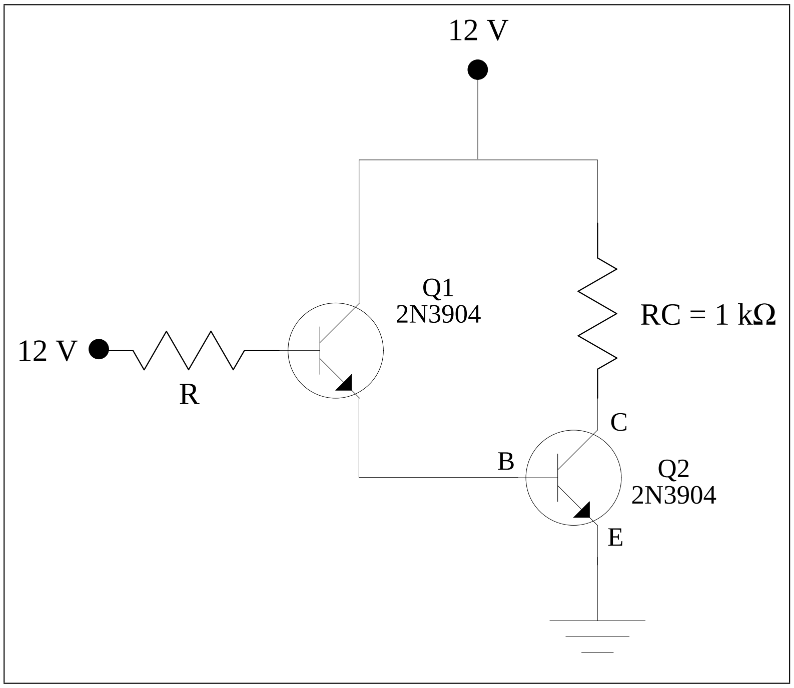

Figure 5.5: Darlington circuit

10. *Is the gain of the circuit twice the gain of a single transistor? Explain.*

Experiment 6

Operational Amplifiers I

6.1 Introduction

Operational amplifiers are a special class of amplifiers that use the concept of negative feed back. For a very brief description of the principles of operational amplifies, see the appendix section of this experiment. An operational amplifier can perform many functions that are very useful in instrumentation. One can build an operational amplifiers using discrete components, however, it is more efficient to build operational amplifies on integrated circuits. Operational amplifier is the best when it comes to simplicity of use in voltage and current amplification and other mathematical functions. Although the operational amplifier, or op-amp, is virtually a black box to most users, the black box can be used as long as its function is understood and limitations are realized. Here we move into the world of integrated circuits and the world of straightforward, modem, instrumentation circuits. There four basic rules and characteristics that are essential for the operation of an ideal op-amp:

1. The circuitry of an operational amplifier with a closed, negative feedback loop will adjust its output in any way it can, in order to make the inverting input (IN_-) and non-inverting (IN_+) input terminals of the device equal in voltage.
2. The inputs draw no current (the input impedance is infinite).
3. The gain, or voltage amplification, is infinite.
4. The output impedance is zero.

This experiment is composed of the following five parts:

Part I Voltage Follower

Part II Inverting Amplifier

Part III Non-inverting Amplifier

Part IV Difference Amplifier

Part V Voltage Summer

6.2 Equipment & Components

1. (1) Oscilloscope
2. (1) Function Generator
3. (1) Resistor Substitution Box
4. (1) 741 Operational Amplifier
5. (1) 100 $k\Omega$ Resistor
6. (1) $1\mu F$ Capacitor
7. Assorted other resistors depending on student design

6.3 Procedure

6.3.1 Part I: Voltage Follower

In the previous experiment we explored the voltage follower. Here we use an op-amp in its most simple configuration, as a voltage follower. A follower is not very complicated in its function. It merely serves to buffer a high impedance (can not provide much current) circuit or transducer to one of lower impedance. The voltage "into" the voltage follower is identical to the voltage "out" of the follower. It's the current gain that's important.

Figure (6.1) shows a 741 op-amp connected as a voltage follower. Construct the circuit using pin-outs shown in Figure (6.2).

CAUTION: The circuits we have used in earlier labs are more forgiving if you power them improperly. IC's are not as forgiving. Always double check all connections, particularly the power connections, before turning on the power. Always power the circuit before powering the input source. If an IC gets hot, chances are something is wrong with the power connections.

Drive the follower with a 100 mV sine wave of 10 kHz. Is V_{out} and V_{in} identical? Is there a phase shift between them? Vary the input frequency and record the upper frequency at which the output drop from being equal to the input? Is there a lower limit?

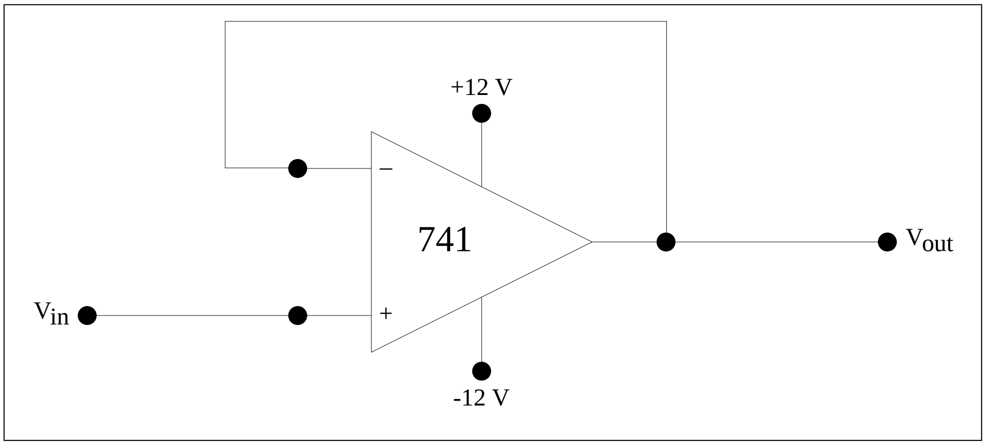

Figure 6.1: Op-Amp Voltage Follower

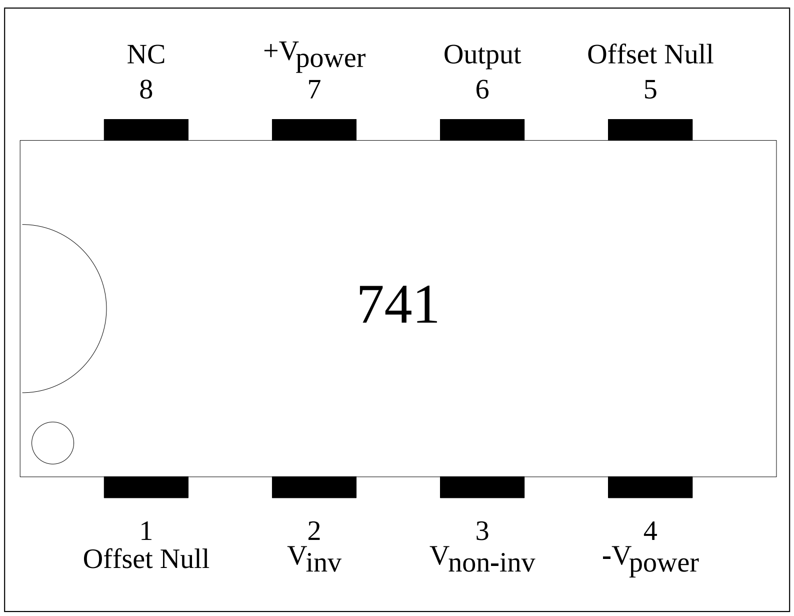

Figure 6.2: Pin-outs for 741 op-amp, note numbering scheme (counter-clockwise starting bottom left) which is general for virtually all IC's

Vary the input amplitude. What happens when V_{in} is large? Does the signal get clipped?

The input impedance of the op-amp follower is very large. Convince yourself of this by charging a 1 μF capacitor to 5 volts and measuring the voltage across the capacitor with a DMM voltmeter. What happens and why? Recharge the capacitor and connect it between V_{in} of the follower and ground and measure the voltage at V_{out} using the same DMM. Now what happens?

1. *Explain what is happening in the two readings above.*

6.3.2 Part II: Inverting Amplifier

The op-amp can be used as a voltage amplifier which does not suffer from the design problems associated with the transistor amplifier of the previous experiment. Figure Figure (6.3) shows an inverting amplifier (V_{in} and V_{out} are of different polarities for DC signals , or 180° out of phase for AC signals). Construct the circuit and verify that

$$\begin{aligned} Gain &= \frac{V_{out}}{V_{in}} \\ &= -\left(\frac{R_2}{R_1}\right) \end{aligned}$$

by holding V_{in} constant, vary R_1 from 1 $k\Omega$ to 100Ω in decade increments, measure V_{out} and compute the voltage gain in each case. Compare this to the theoretical gain given above by R_2/R_1.

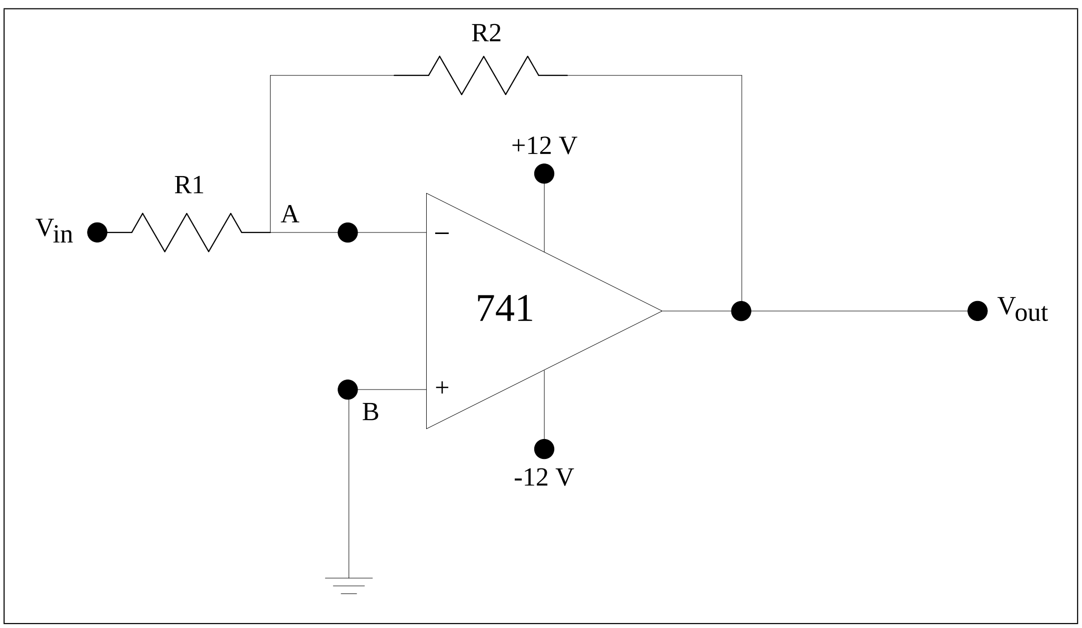

Figure 6.3: Inverting Voltage Amplifier

2. *How well is the voltage gain predicted by the* $Gain = -R_2/R_1$*?*

Set R_1 = 10 $k\Omega$ and R_2 =100 $k\Omega$ verify that the circuit works, and measure the voltage between points A and B using the oscilloscope. Is it what you expected? Explain.

Change R_2 and R_1 to produce an inverting amplifier with gain = 10^4. (Why is leaving R_2 = 100$k\Omega$ and setting R_1 = 100 Ω a bad idea?)

Drive the circuit with a 0.5 mV sine wave of frequency 1 kHz. What can you say about using op-amps for very high gain amplifiers?

3. *Is it possible to produce a circuit with gain -0.1 ? If so, design one and sketch it, if not explain why not.*

6.3.3 Part III: Non-inverting Amplifier

The impedance of an inverting amplifier is controlled by R_1. The non-inverting amplifier does not have this problem. A non-inverting amplifier circuit is shown in Figure (6.4).

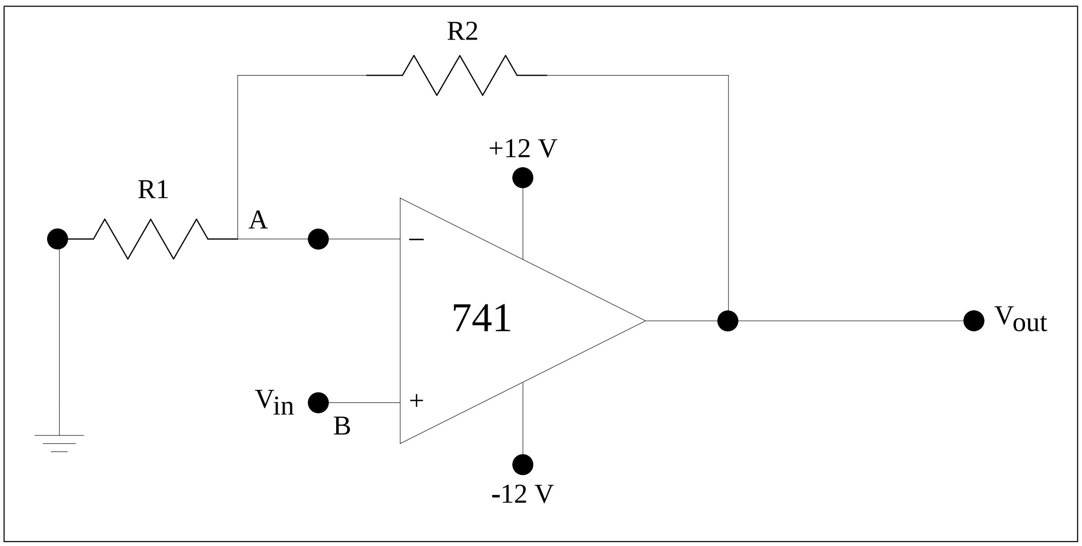

Figure 6.4: Non-Inverting Voltage Amplifier

The gain of the non-inverting amplifier is given by

$$\begin{aligned} Gain &= \frac{V_{out}}{V_{in}} \\ &= \left(1 + \frac{R_2}{R_1}\right) \end{aligned}$$

Repeat the steps used in Part II and verify that the gain is given by the equation above.

4. *How well is the voltage gain predicted by* $Gain = 1 + (R_2/R_1)$*?*

6.3.4 Part IV: Difference Amplifier

In certain applications, one wants to measure the difference in two voltages, such as the voltage drop across a resistor. The difference amplifier can do this. The circuit in Figure (6.5) is a

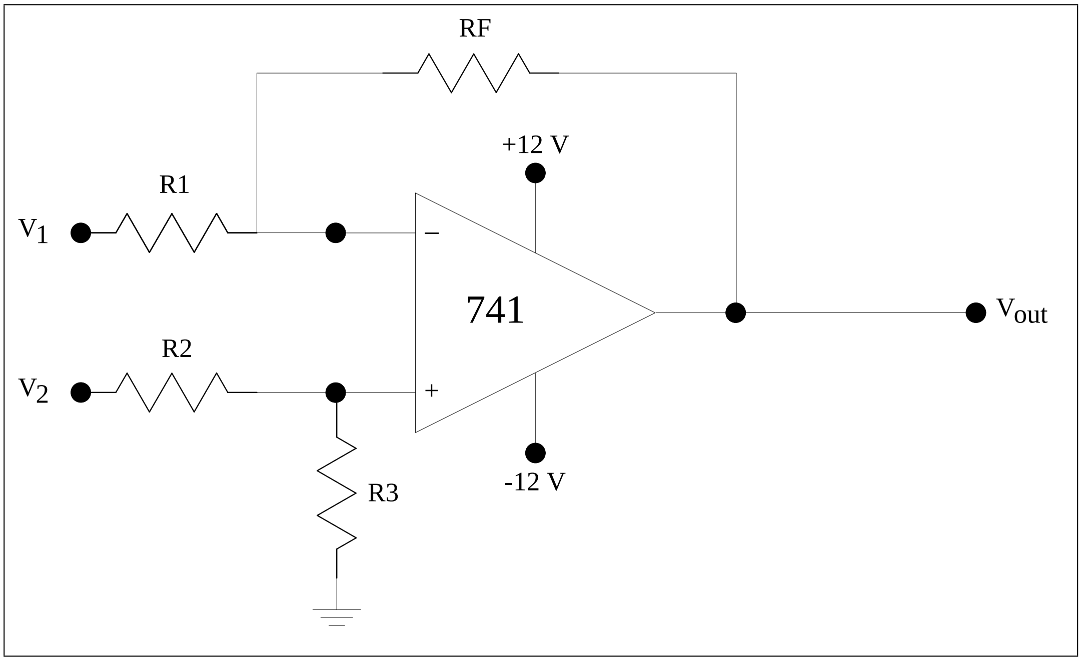

Figure 6.5: Difference Amplifier

difference amplifier that ca be used to measure the difference between V_1 and V_2. The gain is given by:

$$\begin{aligned} Gain &= \frac{V_{out}}{V_2 - V_1} \\ &= \frac{R_F}{R_1} \end{aligned}$$

when $R_1 = R_2$ and $R_3 = R_F$

Choose values for the resistors to yield the Gain $= V_{out}/(V_2 - V_1) = 10$. You probably want to use DC voltages for V2 and VI . Wipers of potentiometers between 12V and ground work well for this. Construct and verify the circuit. Sketch your circuit and any pertinent data.

6.3.5 Part V: Voltage Summer

Operational Amplifiers can be used to perform mathematical functions (operations). The circuit shown in Figure (6.6) is an adder, or summer circuit. V_{out} is given by

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

Construct the circuit choosing values of R_1, R_2, R_3 and R_F and V_1, V_2 and V_3 and verify its operation.
Sketch your circuit and note relevant data readings.

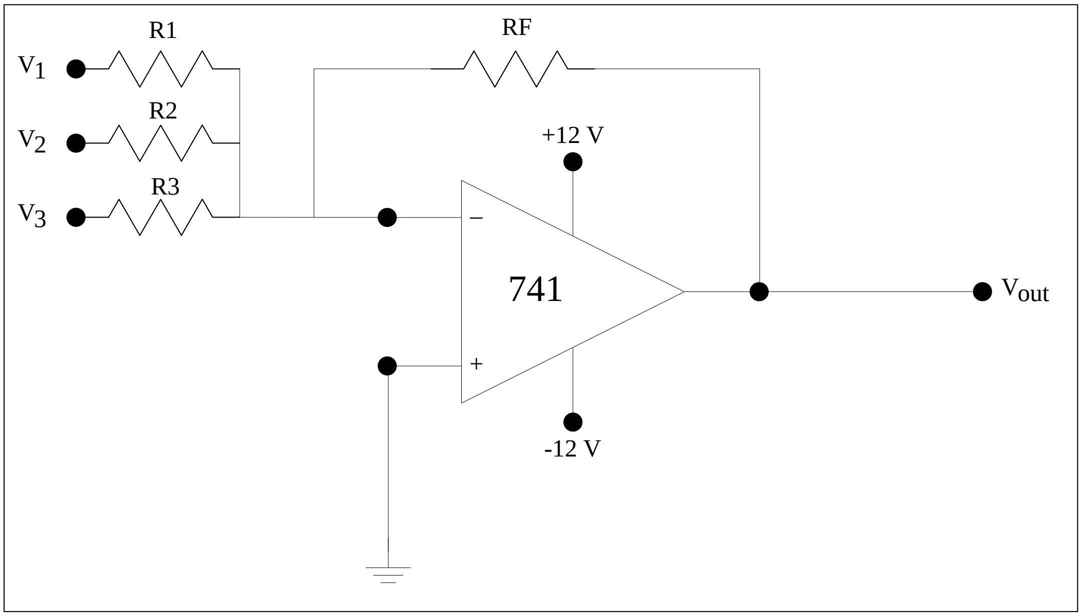

Figure 6.6: Op-amp as an Adder

6.4 Appendix: Brief Description of the Principles of Op-Amps

6.4.1 Amplifiers with Feedback

If a fraction of the output of an amplifier is extracted and added to the input, one can say that the amplifier is operating with feedback. Feedback can be either positive or negative. The former means that the feedback signal is in phase with the input signal while the latter means the feedback signal and input signal are out phase by 180°. An amplifier without feedback and with feedback is shown in Figure (6.7). In this section we will consider negative feedback only. As shown in Figure (6.7) the amplifier has a gain of A withopt feedback given by:

$$A = \frac{v_o}{v_i} \tag{6.1}$$

while the overall gain A_f of the circuit from input to output is given by:

$$A_f = \frac{v_o}{v_s} \tag{6.2}$$

where v_i and v_o are the actual input to the amplifier and the output signals respectively. β determines the fraction of the output that is being fedback to the input, v_s is the input before adding the feedback signal to the and v_f is the feedback signal. We than have for a negative feedback:

$$v_o = Av_i \tag{6.3}$$

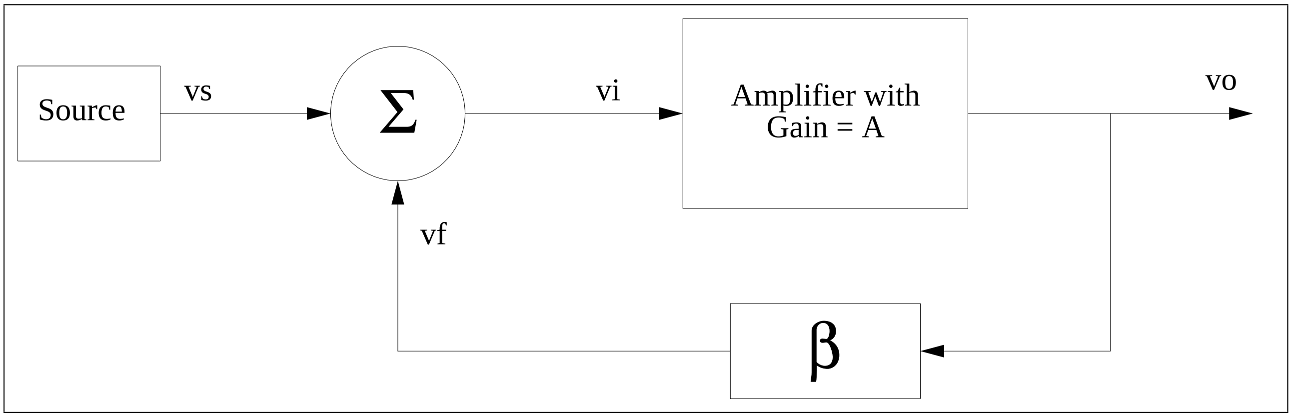

Figure 6.7: An amplifier with feedback

$$
\begin{aligned}
v_f &= \beta v_o && (6.4)\\
v_i &= v_s - v_f && (6.5)\\
A_f &= \frac{v_o}{v_s} && (6.6)\\
&= \frac{Av_i}{v_i + v_f}\\
&= \frac{Av_i}{v_i + \beta v_o}\\
&= \frac{Av_i}{v_i + A\beta v_i}\\
&= \frac{A}{1 + A\beta} && (6.7)
\end{aligned}
$$

The amplifier gain A can be very large, while β is a positive number that is less than one. When $A\beta \gg 1$ then $A_f = 1/\beta$. The overall gain of the circuit is independent of the amplifier gain A. As a result the overall gain is much more stable than the amplifier gain.

Using the above equation we can arrive at the following expression for the feedback signal v_f:

$$v_f = \frac{A\beta}{1 + A\beta} v_s \qquad (6.8)$$

Thus for $A\beta \gg 1$ we see that $v_f \approx v_s$, which implies that v_i at the input of the amplifier is reduced to almost zero.

It is possible to show that negative feedback will increase the input impedance and reduce the output impedance. These are very desirable feature of an amplifier, the former reduces the current drawn from the source and results in a reduced dependence of the input voltage on the input current. A lower input impedance means that more power can be drawn from the amplifier. The input and output impedances of the overall circuit are given by:

$$R_{if} = R_i(1 + A\beta) \tag{6.9}$$

$$R_{of} = \frac{R_o}{1 + A\beta} \tag{6.10}$$

where R_i and R_o are the input impedance and output impedance without feedback. R_{if} and R_{of} are the impedances with feedback.

To explain how an op-amp works, let us analyze the circuit of the inverting amplifier that is being investigated in part II of this experiment. The circuit is shown in Figure (6.3). The negative feedback is provided by R_2. The potential difference between point A and point B is almost zero. Since point B is grounded, then the input to the amplifier at point A is almost zero. Ideally it should be exactly zero. Point A is then called *virtual ground*. This results in a very small current flowing into the amplifier at point A. All the current drawn from the input flows through R_1 and R_2. The amplifier then behaves like an open circuit as shown in Figure (6.8). Applying Kirchoff's law we get:

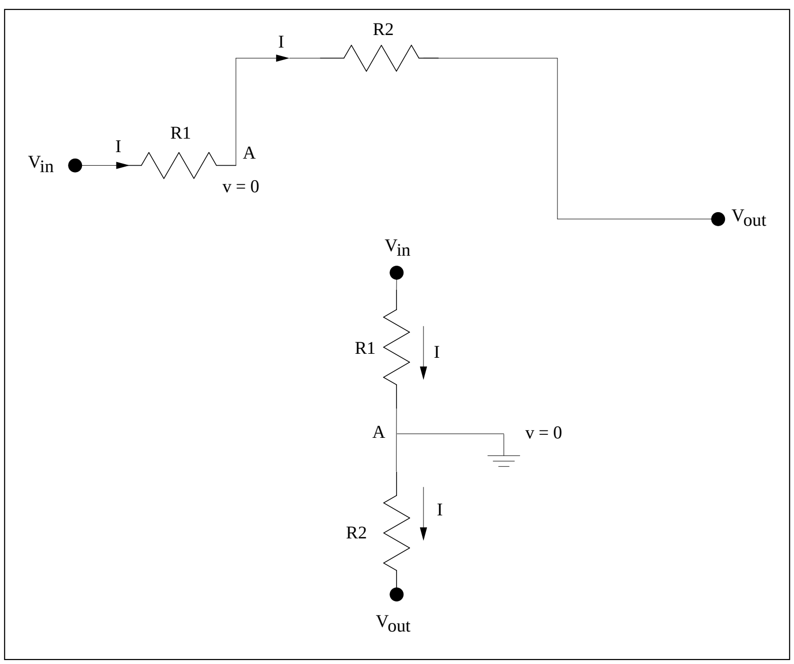

Figure 6.8: Feedback circuit for an inverting amplifier. Note that the two circuits are identical.

$$v_{in} = IR_1 \tag{6.11}$$

$$v_{out} = -IR_2 \tag{6.12}$$

$$v_{out} = -v_{in}\frac{R_2}{R_1} \tag{6.13}$$

$$\begin{aligned} A_f &= \frac{v_{out}}{v_{in}} \\ &= -\frac{R_2}{R_1} \end{aligned} \tag{6.14}$$

The gain is negative, i.e. the output is out of phase by 180° with the input. The gain depends only on the ratio of the two resistors and nothing else. If $R_1 = R_2$ the gain is then equal to one and $v_{out} = -v_{in}$. Using this technique one can then analyse any op-amp circuit. The key to this analysis is that the input at the inverting terminal of the op-amp is virtually at ground potential. The amplifier itself has a very large input impedance and behave as a virtual open circuit for the input current. So, there is very little or no current that flows into the amplifier itself.

Another example, is to analyze the non-inverting amplifier of Part III. The circuit is shown in Figure (6.4). As usual the potential at point A must be equal to the potential at point B. The current through R_1 and R_2 must be the same (they are connected in series) the voltage across R_1 is V_{in} so the current through R_1 is then $I = V_{in}/R_1$ and the same current must also flow through R_2 as well. The output voltage V_{out} is then gieven by the sum of the voltages across R_1 and R_2 i.e.

$$\begin{aligned} V_{out} &= v_{R1} + v_{R2} \\ &= IR_1 + IR_2 \\ &= I(R_1 + R_2) \\ &= \frac{V_{in}}{R_1}(R_1 + R_2) \\ &= V_{in}\left(1 + \frac{R_2}{R_1}\right) \end{aligned}$$

Experiment 7

Operational Amplifiers II

7.1 Introduction

In the previous experiment we investigated basic op-amp circuits. Here we will look at some more advanced circuits and explore the limitations of op-amps which can sometimes be severe. There are three parts to this experiment as follows:

Part I Filters

Part II Comparators

Part III Op-amp limitations

7.2 Equipment and Components

1. (1) Oscilloscope
2. (1) Function Generator
3. (1) Resistor Substitution Box
4. (1) 741 Operational Amplifier
5. Assorted other resistors depending on student design
6. (1) 10 $k\Omega$ Potentiometer

7.3 Pocedure

NOTE: Many of the circuits in this laboratory require carefully matched components ($R_1 = R_2$, $C_1 = C_2$). For best results, care should be taken to match these as closely as possible.

CAUTION: The circuits we have used in earlier labs are more forgiving if you power them improperly. IC's are not as forgiving. Always double check all connections, particularly the power connections, before turning on the power. Always power the circuit before powering the input source. If an IC gets hot, chances are something is wrong with the power connections.

7.3.1 Part I: Filters

Op-amps make excellent foundations for active filters. Figure (7.1) shows an active high pass filter. When both resistors are equal and both capacitors are equal value the breakpoint frequency is given by:

$$f_B = \frac{1}{2\pi RC}$$

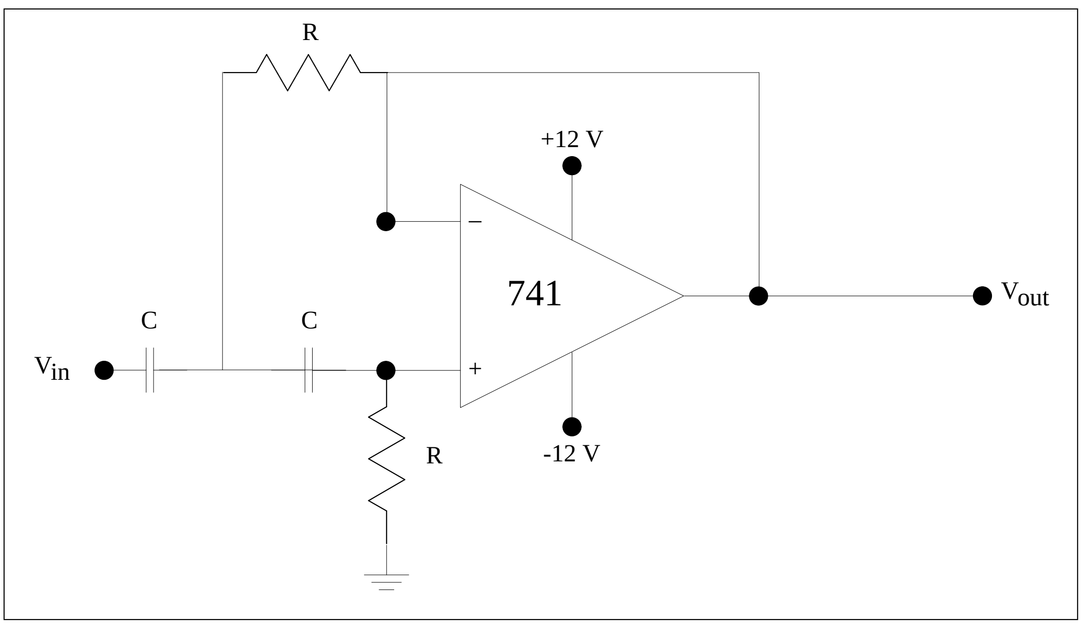

Figure 7.1: Op-Amp as Active High Pass Filter

1. *Describe, either quantitatively or qualitatively how the circuit shown in Figure (7.1) works.*

2. *Derive the breakpoint frequency for the active high pass filter.*

Construct the active high pass filter in Figure (7.1) with a breakpoint frequency of about 1 kHz. Drive the circuit with a 1 V peak-to-peak sine wave. Sketch the circuit and label the values you chose. Vary the frequency of the driving signal and plot the Gain (dB) versus the frequency. What is the maximum gain in dB?

3. *How does the measured break point frequency compare with your calculated value?*
4. *Compare and contrast the active high pass filter and the passive high pass filter of Experiment # 1.*

Figure (7.2) shows an active low pass filter. When both resistors are equal and both capacitors are equal the breakpoint frequency is given by:

$$f_B = \frac{1}{2\pi RC}$$

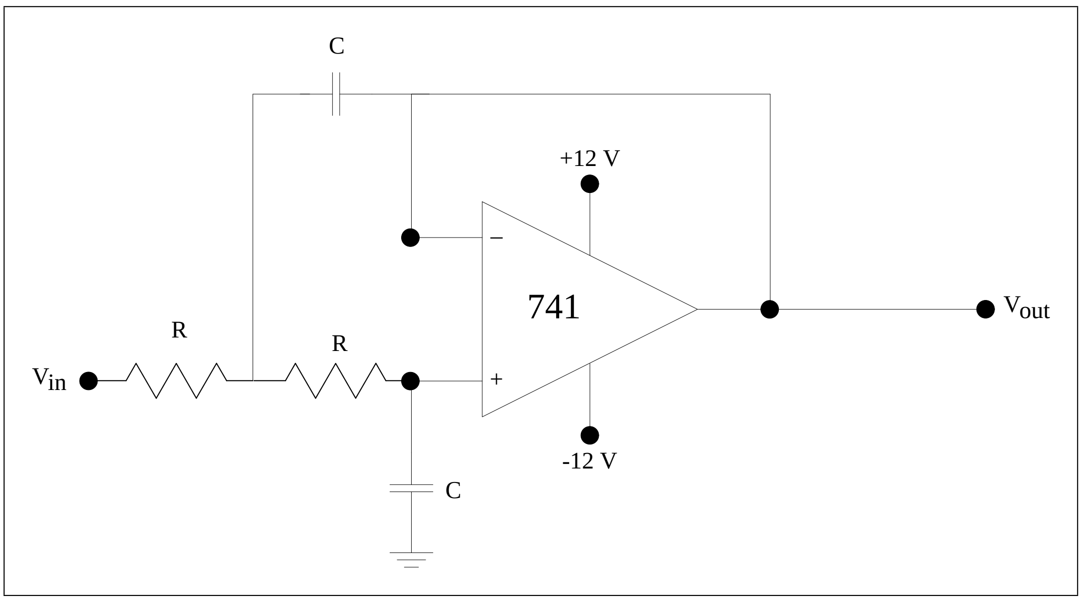

Figure 7.2: Op-Amp as Active Low Pass Filter

5. *Describe, either quantitatively or qualitatively, how the circuit shown in Figure (7.2) works.*
6. *Derive the breakpoint frequency for the active high pass filter.*

Construct the active low pass filter in Figure (7.2) with a breakpoint frequency of about 1 kHz. Drive the circuit with a 1 V peak-to-peak sine wave. Sketch the circuit and label the values you chose and Vary the frequency and plot the Gain (dB) versus the frequency. What is the maximum gain in dB for the circuit shown in Figure (7.2)?

7. *How does the measured breakpoint frequency compare with your calculated value?*
8. *Compare and contrast the active low pass filter and the passive low pass filter of Experiment # 1.*

7.3.2 Part II: Comparators

Op-amps can be used to compare two voltages and produce an output that depends on that comparison. Consider the circuit shown in Figure (7.3). The circuit offers no feedback from the op-amp output to the inputs. Therefore the op-amp amplifies the difference between the inputs by its gain (of the order of 10^5) and saturates the output to either the positive or negative power supply voltage limit.

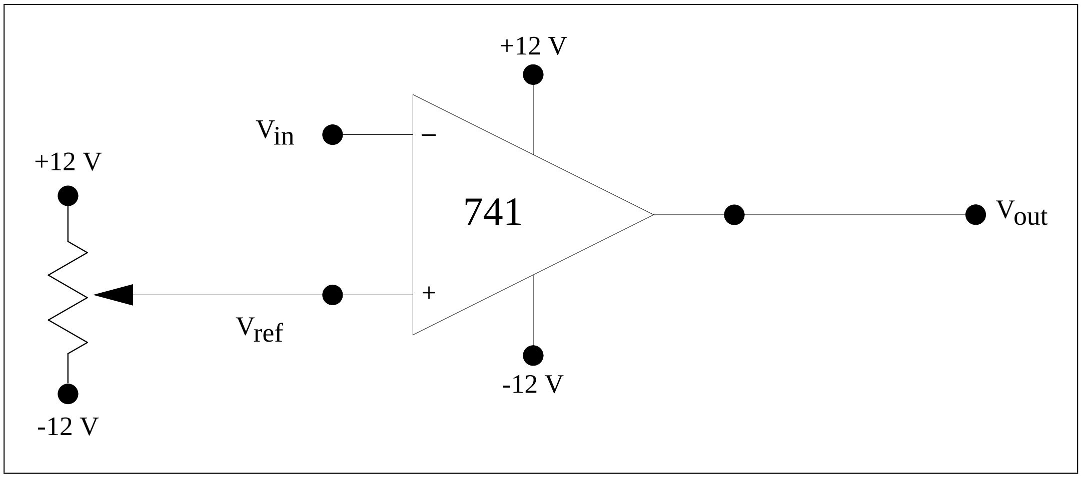

Figure 7.3: Op-amp Comparator Circuit

In Figure (7.3) the non-inverting input serves as a reference voltage, V_{ref}, which is set by the 10 $k\Omega$ variable resistor. If V_{in} is greater than V_{ref} the output will saturate to the negative rail and vice versa.

Construct the circuit shown in Figure (7.3) and drive it with a 5 V peak-to-peak sine wave. Observe the output. Vary V_{ref} and observe the output. Record your observations.

7.3.3 Part II: Op-amp Limitations

In general we have assumed that we are using ideal op-amps, in this experiment and Experiment 6. Op-amps are not ideal, they do have limitations. One serious limitation is their frequency response and related slew rate. Op-amps do not make good high frequency amplifiers as we shall soon see.

Construct the non-inverting amplifier shown in Figure (7.4) with gain of 1 and $R1 = 1\ k\Omega$. Measure its gain in dB as a function of frequency in decade intervals from 1 Hz to 10 MHz (if your function generator goes that high) for a 1 V peak-to-peak sine wave and plot the gain in dB versus frequency. Note the depanure from ideal in the high frequency range.

Change the gain to 10 by changing R_2 and repeat your measurements for an input signal of

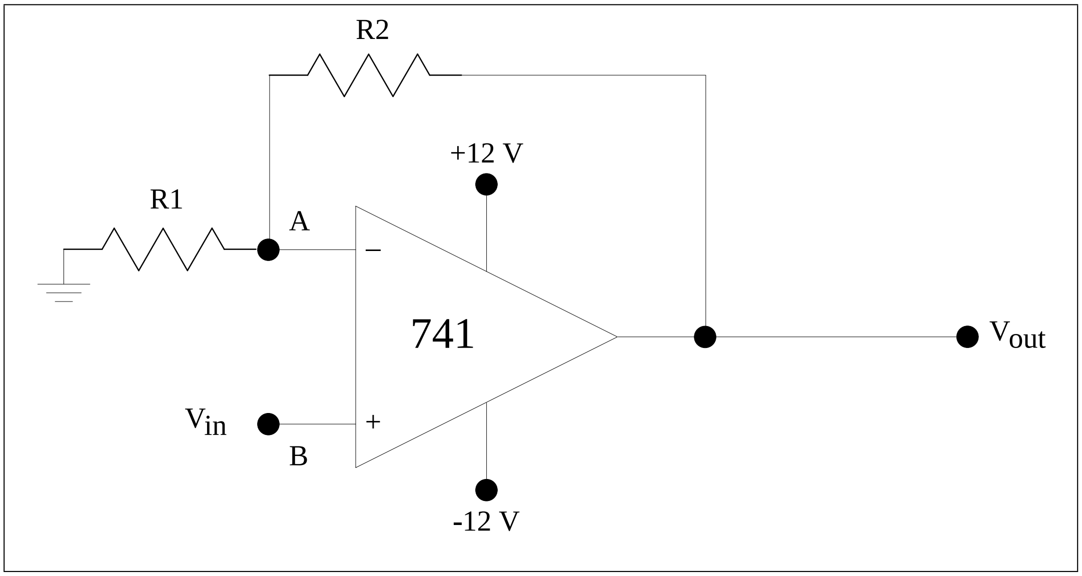

Figure 7.4: Non-inverting amplifier

0.1 V. Do the same for a gain of 100 and 1000 (lower the input voltage accordingly).

9. *Discuss the gain vs. frequency limitation of an op-amp.*

Experiment 8

Oscillators

8.1 Introduction

So far we have used the function generator as our prime oscillator. But one can not include a function generator in every circuit. Imagine a circuit that simply flashes an LED that must include a $300 function generator! We must know how to produce our own specific oscillators from integrated circuits and resistors and capacitors. This experiment explores just that.

Before we proceed, we need to define "Duty Cycle" ($D.C.$). When a signal is pulsating, like the one produced by a half-wave rectifier, the signal is on (or has high magnitudes) for a certain period of time and off (or has low magnitudes) for another period. In other words the signal is effective only part of the time. For example a DC signal has 100% duty cycle and symmetrical square wave has duty cycle of 50%. Duty cycle defines how effective the signal is. Let us assume that the signal has high magnitudes during a time interval "t_1" and has low magnitudes during a time interval "t_2". The Duty Cycle ($D.C.$) is then defined as:

$$D.C. = \frac{t_2}{t_1 + t_2} \tag{8.1}$$

There are three parts to this laboratory as follows:

Part I: Waveform Shaping

Part II: ICL 555 Timer IC

Part III: 8038 Precision Function Generator IC

8.2 Equipment and Components

1. (1) Oscilloscope
2. (1) Function Generator

3. (1) 555 Timer IC
4. (1) 311 Comparator IC
5. (1) ICL 8038 Precision Waveform Generator IC
6. Miscellaneous Capacitors and Resistors

8.3 Procedure

8.3.1 Part I: Waveform Shaping

Before we delve into discrete waveform generators, we will explore ways of changing one waveform to another. Note that a square wave is the easiest signal to produce - it goes on and off. Shaping is a necessity if we wish to develop other waves.

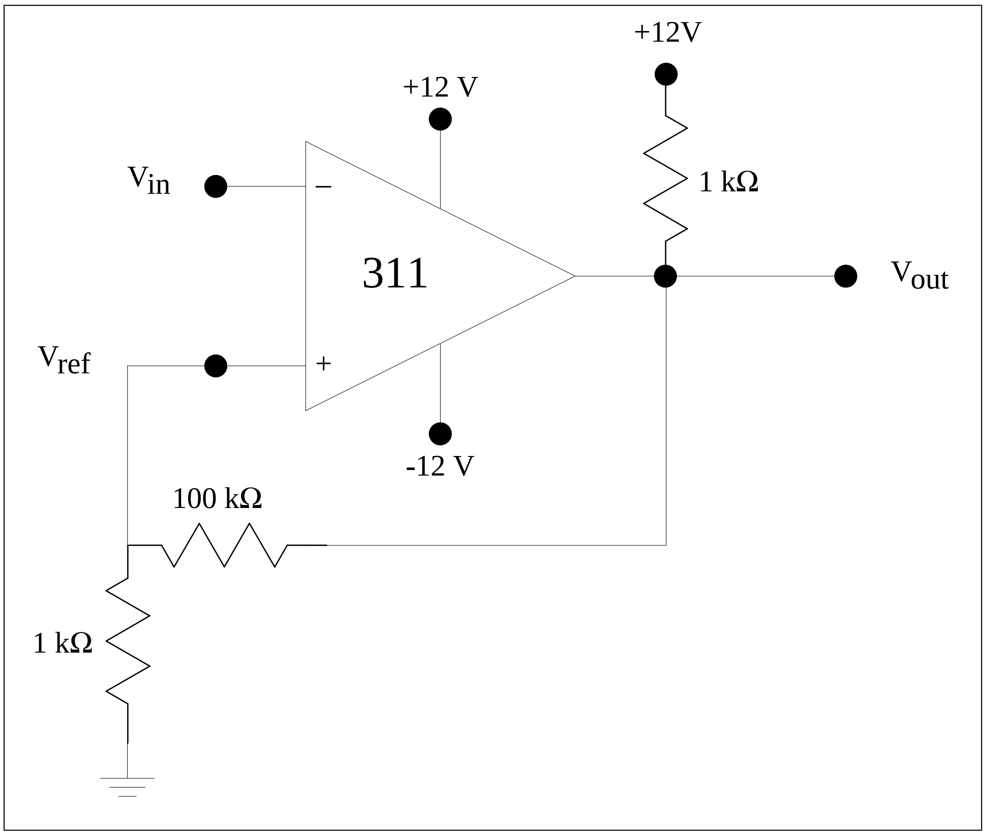

Figure 8.1: Comparator circuit for shaping sine wave to square wave

When V_{in} is less than the reference voltage V_{ref}, the output voltage V_{out} of the comparator is V_+ (12 volts in this case) or high output. When V_{in} is more than V_{ref} then the output is low of V_- (-12 V in this case). V_{ref} is set up by V_{out} and the voltage divider network composed of the 1 $k\Omega$ and the 100 $k\Omega$ resistors.

Construct the comparator circuit shown in Figure (8.1). Drive the circuit with a 1 V peak-to-peak sine wave centered at zero volts. Observe the output. Is it what your expected?

1. *Is the square wave totally symmetric? Explain why or why not.*
2. *Can you change the circuit to produce a 25% duty cycle square wave? If so explain how, if not explain why.*
3. *Can you change the circuit to produce a 75% duty cycle square wave? If so explain how, if not explain why.*

8.3.2 Part II: 555 Timer IC

Few integrated circuits are as versatile as the 555 timer. Many volumes have been written on it and its various uses. Here we will use it in its most straightforward application - as a square wave generator.

When the output is low or zero, the capacitor in the circuit of Figure (8.2) discharges via an internal transistor connected to pin 7 through resistor R_2 when the voltage across the capacitor reaches $V_+/3$, the output goes high to V_+, and the the transistor is turned off. The capacitor then starts to charge through resistors R_1 and R_2. When the capacitor voltage reaches $2V_+/3$ the output voltage goes low and the transistor turns on and the cycle is repeated. Charging and discharging times are therefore given by exponential charge and discharge characteristics as:

$$\begin{aligned} t_1 &= 0.693(R_1 + R_2)C_1 && \text{charging time, output high} \\ t_2 &= 0.693R_2C_1 && \text{discharging, output low} \end{aligned}$$

The frequency of the oscillation f is thus given by:

$$\begin{aligned} f &= \frac{1}{T} \\ &= \frac{1}{t_1 + t_2} \\ &= \frac{1.44}{(R_1 + 2R_2)C_1} \end{aligned}$$

and the duty cycle D.C. is:

$$\begin{aligned} D.C. &= \frac{t_2}{(t_1 + t_2)} \\ &= \frac{R_2}{R_1} + 2R_2 \end{aligned}$$

The last definition of the D.C. is the manufacturer definition which is different from the standard definition.

Construct the classic 555 oscillator circuit shown in Figure (8.2) choosing R1, R2 and C1 for an output of 1 kHz with as close to 50% duty cycle as you can. Verify that the circuit works and note the values you have chosen. Figure (8.3) shows the pin arrangement of the 555 IC.

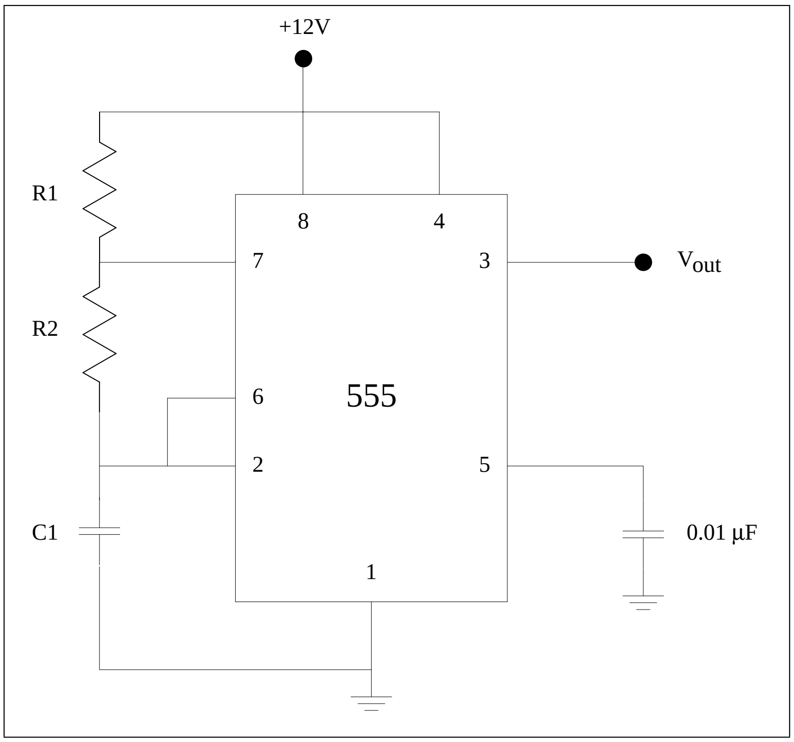

Figure 8.2: 555 Oscillator circuit

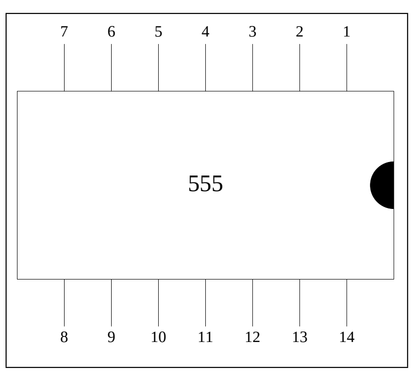

Figure 8.3: Pin arrangement of 555 timer IC

4. *How would you change your circuit to produce a pulse of relatively small duration? Try it out and explain what you changed and why.*

5. *If you disconnected pin 2 and instead connected it to a wiper of a 10 $k\Omega$ potentiometer that was connected between 12 V and ground, how would the output change? (You might want to try this before answering...)*

6. *If you disconnected pin 6 and instead connected it to a wiper of a 10 $k\Omega$ potentiometer that was connected between 12 V and ground, how would the output change? (You might want to try this before answering...)*

7. *If you disconnected pin 5 and instead connected it to a wiper of a 10 $k\Omega$ potentiometer that was connected between 12 V and ground, how would the output change? Note that such a circuit is called a voltage controlled oscillator (VCO) - why? (You might want to try this before answering...)*

Just for fun change components of the circuit in Figure (8.2) such that the 555 oscillates with a square frequency of about 1 Hz. Connect an LED to the output with one lead of the LED to +12V and a 510 $k\Omega$ resistor and the other to the output of the 555. Which lead goes to the output - anode or cathode? Enjoy the light show.

8.3.3 Part III: ICL 8038 Precision Function Generator IC

As you have seen, constructing precision sine, triangle and square waves is not an easy task. To the rescue comes the Intersil ICL 8038 function generator IC. The ICL 8038 IC is a precision waveform generator which will produce high-quality square, triangle, and sine waves (all at the same time) over a frequency range of 0.001 Hz to over 300kHz. The signal can be swept over a given range, or frequency modulated, by externally applied control voltage. Figure (8.4) shows the pin-out arrangement of this 14-pin, DIP chip, while Figure (8.5) shows a typical application. The frequency of the output signal is simply given by:

$$f = \frac{0.15}{RC}$$

for the circuit shown in Figure (8.5).

Construct the circuit shown in Figure Figure (8.5) and verify its operation. Does it produce signals of higher quality than those which you have designed and constructed (don't feel bad- the difficulties involved explain why the ICL 8038 is such a popular device!).

8. *What components would you add to the sine wave output of the ICL 8038 to increase the current available to 1 A? Draw the schematic below.*

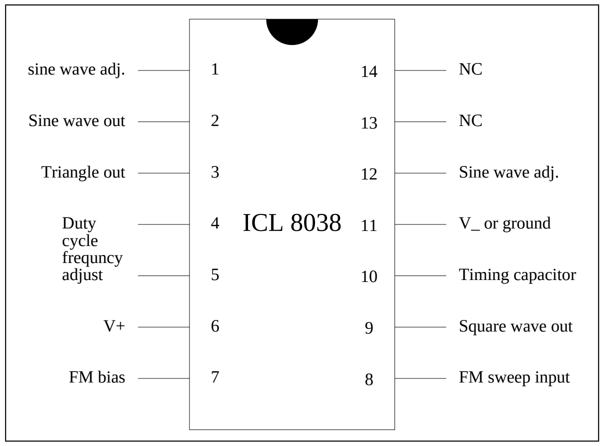

Figure 8.4: Pin arrangement of Intersil ICL 8038 function generator IC

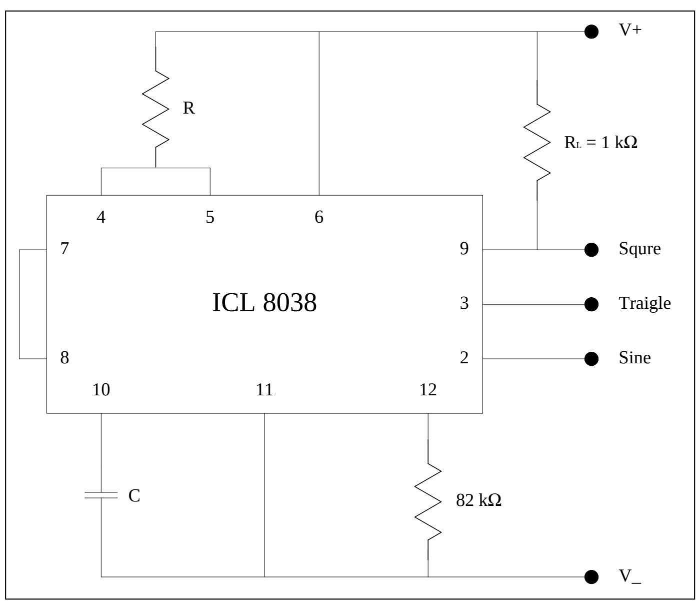

Figure 8.5: Typical application of ICL 8038 function generator IC

Experiment 9

Digital Circuits: Gates and Digital Logic

9.1 Introduction

Digital circuitry is much simpler to use than analog circuitry. In digital circuits the output is either on or off. In this lab we will explore some simple digital logic gates.
There are two parts to this laboratory as follows:

Part I: Simple Gates

Part II: Three-state logic

Equipment Needed

1. (1) Oscilloscope
2. (1) Function Generator
3. (1) Logic Probe or DMM voltmeter
4. (2) 4011 CMOS NAND Gates
5. (1) 7400 TTL NAND Gate
6. (1) 7403 TTL Open-collector NAND Gate

9.2 Procedure

[NOTE: Power supply connections are generally not shown in digital circuit schematics. Make sure the power supply connections are made. While TTL is very demanding in its tolerance for 5 V DC power, CMOS is much less so. We will adopt to powering all digital circuits with 5 VDC.]

The pin assignment of the he 7400 and 7403 NAND IC's is shown in inFigure (9.1). The pin assignment for 4011 is shown in Figure (9.2).

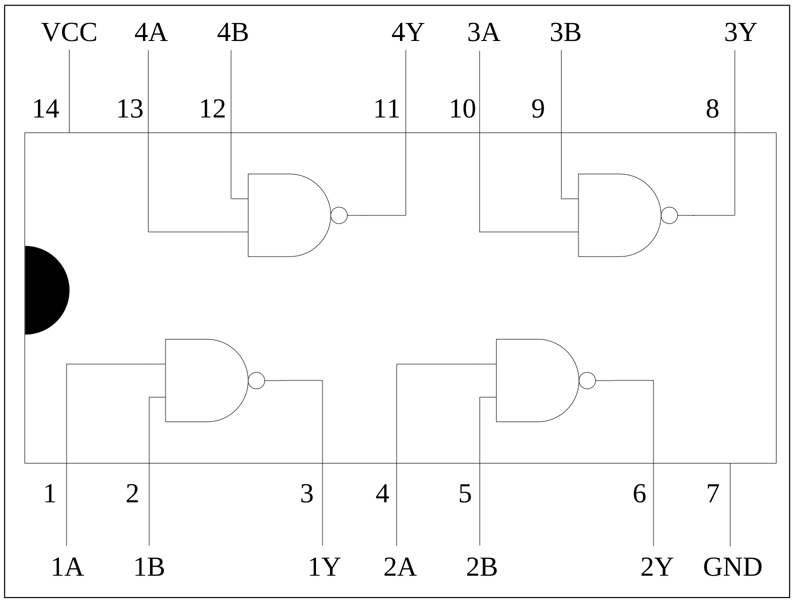

Figure 9.1: Pin assignment for 7400 and 7403.

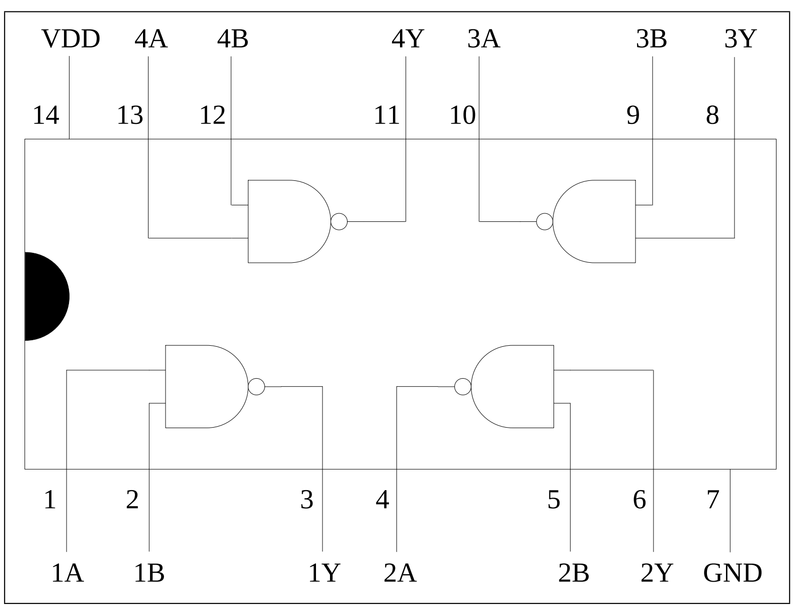

Figure 9.2: Pin assignment for 4011.

9.2.1 Part I: Simple Gates

Power a 4011 NAND gate and change the inputs and find out the status of the output, then list your results in logic table for one of the NAND gates. You can use a digital logic probe to accomplish this or a DMM voltmeter. Remember for outputs, high is over 3 V and low is less than 2 V as a rule of thumb. Connect the inputs via jumper wires to your +5V and ground power rails.

Using NAND gates, design and construct an AND gate, NOR gate, OR gate and an Inverter. Sketch your designs and test them. Summarize your results in an appropriate logic table for each case.

Connect the 4011 NAND gate as an inverter as shown in Figure (9.3). Connect a $10k\Omega$

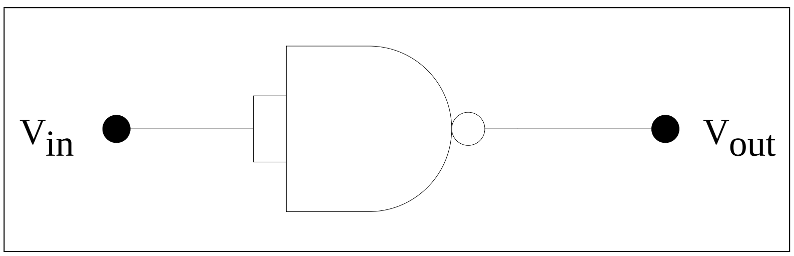

Figure 9.3: NAND gate as an inverter

potentiometer across 5V and ground with the wiper attached to produce V_{in}. Raise V_{in} from 0 V until V_{out} changes state. Record this threshold limit.

Repeat for the 7400 TTL NAND gate.

9.2.2 Part B: Three-state logic

Connect the 7400 TTL NAND gate as shown Figure (9.4). A seemingly simple circuit designed to produce a high signal when any of the four outputs is low. Verify that the circuit does not work very well.

1. *"OPTIONAL" When does the circuit work, when does it not and why? Try to answer if can.*

Now construct the circuit shown in Figure (9.5) and verify that it is a good circuit.

2. *"OPTIONAL" Explain why the circuit shown in Figure (9.5) works.*

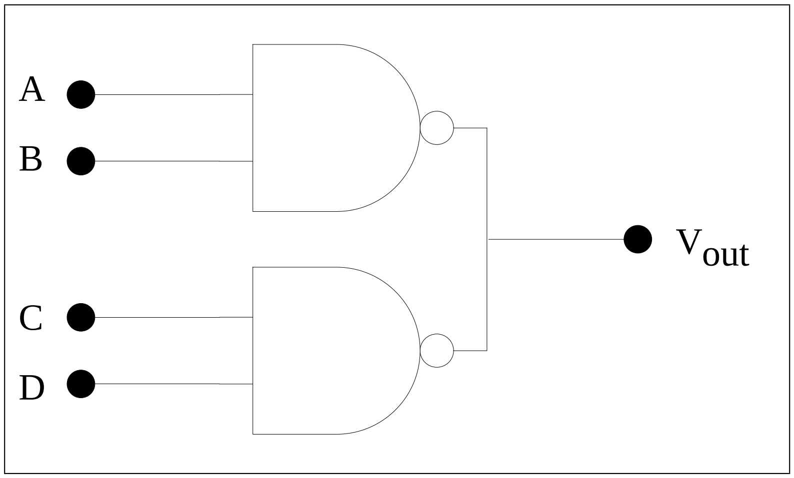

Figure 9.4: Using 7400 in a poor logic configuration o produce high output when any input is low.

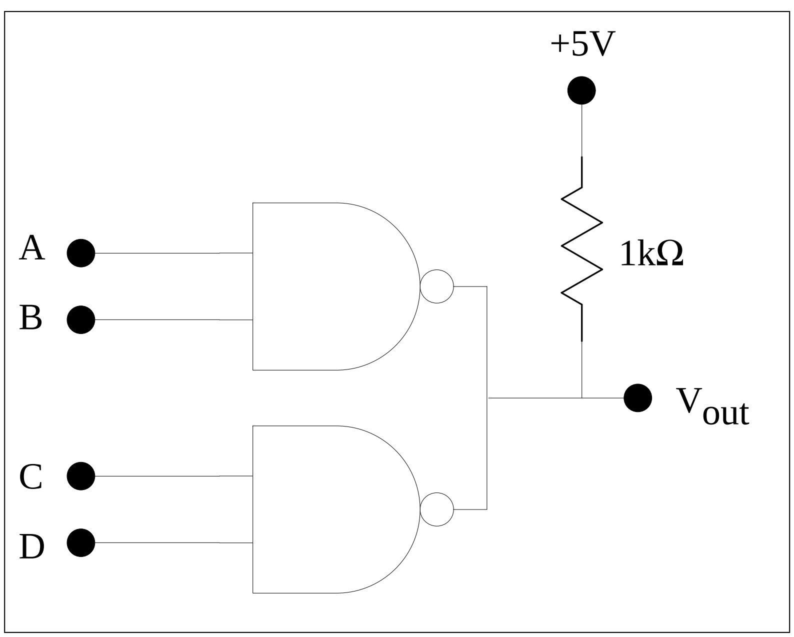

Figure 9.5: Using 7403 Open-collector circuit to replace poor circuit in Figure (9.4).