Version 3.1

PHYS 576
Electronics

K. Eid

Acknowledgements

The first version of the *Electronic Instrumentation Manual* was produced by John W. Snider and Joseph Priest in 1988. At the time, Apple IIe computers were used along with an analog-to-digital converter that was built in-house. IBM-type computers and a commercial data acquisition system were introduced in 1991. Graduate students Terry Howald and Tim Burt did yeoman work in adapting the exercises to the new system. LabVIEW technology was introduced to the laboratory by Michael Pechan and Joseph Priest in 1999. Digital electronics exercises were introduced by Jeffrey Clayhold in 2006 and 2007. Analysis of the pendulum experiment was modified in 2008 by Michael Pechan and Jeffrey Clayhold and rewritten by Herbert Jaeger in 2011. Exercises on high-gain amplification and lock-in detection were added in 2007 and 2008 by Joseph Priest, Jeffrey Clayhold, and Michael Pechan. The use of the *Arduino* microcontroller was introduced by Herbert Jaeger in 2012. Many students have contributed to the development of both the laboratory and manual, and the authors are grateful for their efforts. We, and the Department of Physics, in general, are appreciative of significant support from Miami University and grants from the Instrumentation and Laboratory Improvement program conducted by the National Science Foundation.

CONTENTS

ELECTRICAL MEASUREMENTS

Objectives

1. To evaluate the effect of the internal resistance of a voltmeter and an ammeter on measurements of voltage and current.

2. To use a spreadsheet to perform a linear regression analysis of experimental data.

Background

Whenever a voltmeter or ammeter is connected to measure a voltage or current, the instrument alters to some extent the voltage or current that existed before the meter was connected. Very likely, measurements of voltage and current that you make will utilize meters that display a numerical result on a screen on the instrument. Generally, these instruments make minimal alterations to the voltage or current being measured. However, you should always be aware that the instrument can affect the measurement being recorded and you may have to correct for the presence of the meter. While the experiment at hand involves a rather old fashioned instrument it does provide a lesson in the effect a voltmeter or ammeter has on the measurement of a voltage or a current.

Voltmeters and ammeters using a pointer that rotates in front of a calibrated numerical scale probably employ a current sensitive instrument called a d'Arsonval galvanometer.1 The pointer is deflected through an angle that is proportional to the current in the meter. A d'Arsonval galvanometer can detect currents as small as a picoampere (10^{-12} A). With rather simple modifications it can be converted into an instrument for measuring rather large currents and voltages. It is important to remember that even when the d'Arsonval galvanometer is used in a voltmeter, it is basically a current-sensitive instrument.

An ammeter using a d'Arsonval galvanometer will usually have a resistor (shunt) connected in parallel with the coil of the galvanometer. The resistance of the shunt in the milliammeter in this experiment is chosen so that 1 mA in the meter produces a full-scale deflection of the pointer. To measure the current in an element such as a resistor the current in the meter must be the same as the current in the element. Accordingly, the milliammeter must be connected in series with the element in which the current is to be measured (Figure 1).

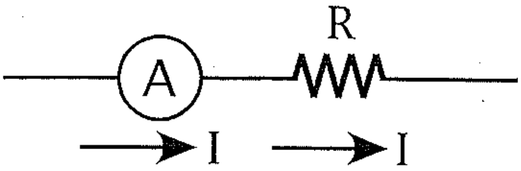


Figure 1: Ammeter connection in a circuit. The circuit is broken and the ammeter connected to measure the current.

The d'Arsonval galvanometer used in the voltmeter in this experiment has a resistance of about 10,000 Ω connected in series with the galvanometer coil. The series resistance is chosen so that 10 V across the voltmeter's terminals produces a full-scale deflection of the pointer. Because a voltmeter measures the potential difference between two points such as the ends of a resistor, the potential difference between the terminals of the voltmeter must be the same as the potential difference being measured. Accordingly, the voltmeter must be connected in parallel with the element across which the voltage is to be determined (Figure 2).

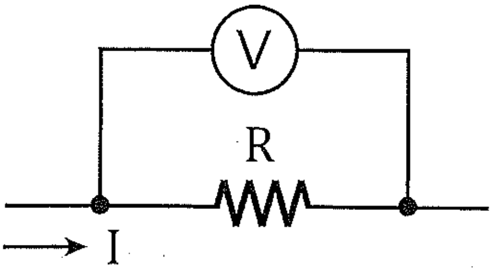


Figure 2: Voltmeter connections in an electrical circuit. The leads of the voltmeter are connected to the ends of the resistor.

Procedure

Part I:

Construct the circuit on a solderless circuit board as shown in Figure 3a. Use a 100 Ω resistor for R, a 0–1 mA analog ammeter to measure the current, a digital voltmeter to measure the voltage, and a variable power supply. Good, solid connections are required. There should be no connection where two wires are twisted together. Note that the variable power supply is symbolized in the diagram as a battery. With the circuit connected in this manner, the ammeter correctly measures the current in the resistor, but the voltmeter measures the potential difference across the series combination of resistor and ammeter.

After your circuit has been checked, turn the power supply on and record the current and voltage for at least 10 values of the current between 0 and 1 mA. Do not disassemble your apparatus. You may want to check some of the measurements or repeat all of the measurements after you have made a graph of your data. Using a digital multimeter, measure the resistances of the 100 Ω resistor and the ammeter. For the circuit in Figure 3a the voltmeter measures the sum of the voltages across the ammeter and resistor. Since the current is the same in the ammeter and resistor it follows that

$$V = IR_A + IR = I(R_A + R) \qquad (1)$$

Thus the slope of a plot of V versus I has a slope of R_A + R.

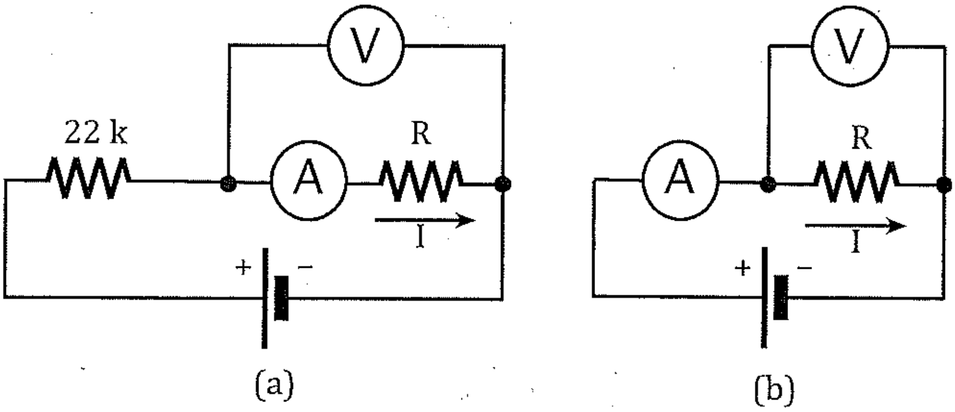


Figure 3: **(a)** Voltmeter and ammeter connections for Part I. **(b)** Voltmeter and ammeter connections for Part II.

Prepare a graph of the data and analyze the results following the instructions given below.

1. Load EXCEL into the computer.

2. Enter the values of current (I) and voltage (V) in separate columns.

3. Plot the data as voltage (V) versus current (I) using the graphing features inEXCEL. Use symbols for the data points. **Do not connect the data points with lines.**

4. a) On the graph containing the experimental results, plot the function V = IR where R is the resistance of the resistor. This plot should be a straight line with no data points.

b) On the graph containing the experimental results, plot the function V = IR where R is the sum of the resistance of the resistor and the resistance of the ammeter. This plot should also be a straight line with no data points.

5. Label each column in the spreadsheet to avoid confusion. Obtain a hard copy of the graph and the spreadsheet data for your report.

Part II:

Wire the circuit shown in Figure 3b using a 0–1 mA analog ammeter to measure the current, a 0–10 V analog voltmeter to measure the voltage, and a 10 kΩ resistor for R. Measure the current and voltage for at least 10 values of the voltage between 0 and 10 V.

For the circuit in Figure 3b the ammeter measures the sum of the currents in the voltmeter and resistor. Since the voltage is the same for the voltmeter and resistor it follows from Kirchhoff's junction rule that

$$I = \frac{V}{R_V} + \frac{V}{R} = V\left(\frac{1}{R_V} + \frac{1}{R}\right) = V\frac{R_V + R}{R_V R} \qquad (2)$$

Thus a plot of V versus I has a slope $\frac{RR_V}{R + R_V}$.

Prepare a graph of the data and analyze the results following the instructions given below.

1. Open EXCEL.

2. Enter the values of current (I) and voltage (V) in separate columns.

3. Plot the data as voltage (V) versus current (I) using the graphing features in EXCEL. Use symbols for the data points. **Do not connect the data points with lines.**

4. a) On the graph containing the experimental results, plot the function V = IR where R is the resistance of the resistor. This plot should be a straight line with no data points.

b) On the graph containing the experimental results, plot the function V = IR where R is the resistance of the parallel combination of the resistances of the voltmeter and resistor. This plot should also be a straight line with no data points.

5. Label each column in the spreadsheet to avoid confusion. Obtain a hard copy of the graph and the spreadsheet data for your report.

Your report should include:

1. Graphs of the measurements and the spreadsheets showing the experimental measurements and the theoretical fits to the data.

2. Appropriate comments about the effects of meter resistance on measurements of current and voltage.

References

The d'Arsonval galvanometer is discussed in most general physics texts. See, for example, ***University Physics***, Hugh D. Young, Addison-Wesley, 8th ed. (1992)

Apparatus Notes: The Breadboard Workstation

Most of the circuit work in this course will be done using a breadboard workstation (Figure 4). It consists of four solder-less breadboard mounted on the central area of the workstation. The breadboard is attached by velcro and can easily be replaced if the circuit needs to be saved. The workstation incorporates an adjustable power supply, function generator, two debounced switches, and a number of other tools that are useful when setting up and testing circuits on a breadboard.

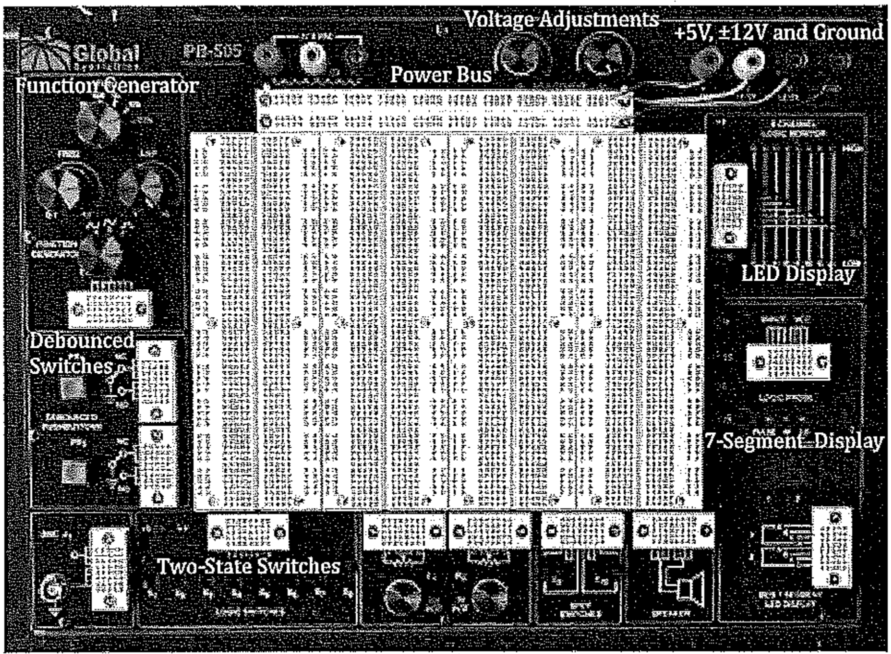
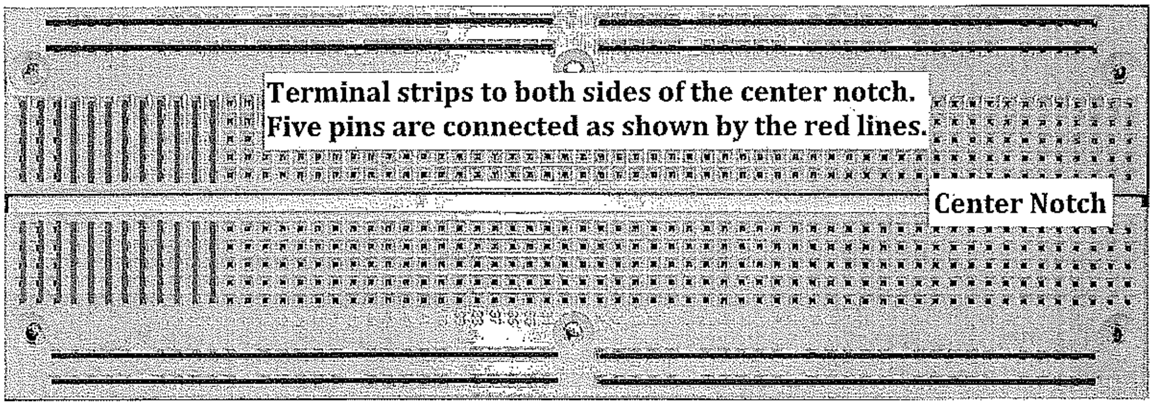


Figure 4: Breadboard work station used in this course.

Each of the 4 breadboards consists of terminal and bus strips (Figure 5). Most of the components will be arranged in the terminal strip area. Integrated circuits such as op-amps and TTL chips will be mounted straddling the center notch.



Two bus strips on each side of the terminal strip area. The bus strips are connected as shown by the blue lines. Note that there is no connection between the right and the left group of bus strips.

Figure 5: The anatomy of a breadboard.

VOLTAGE DIVIDERS AND KIRCHHOFF'S LOOP RULE

Objectives

To gain confidence in using a voltage divider and Kirchhoff's loop rule.

Procedure

Part I: Voltage Divider

The electric current (I) is the same in the series connection of the two resistors shown in Fig. 1. Since the resistors obey Ohm's law (V = IR) it follows that $\frac{V_1}{E} = \frac{R_1}{R_1+R_2}$. For this reason the circuit is termed a "voltage divider." Voltage dividers are extremely useful when situations arise for establishing a voltage different from the supply voltage E.

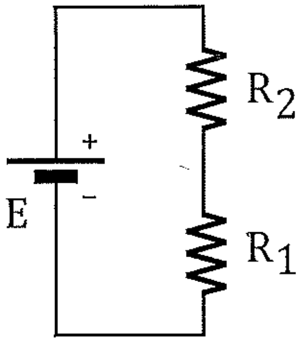


Figure 1: Series connection of two resistors and a voltage source.

I. Choose two resistors having resistances in the range of a few kilohms. Measure the resistances with an ohmmeter. Connect the two resistors as shown in Fig. 1 using a power supply producing about 10V for the source (E). Measure and record the voltage of the source and the voltage on each resistor. Using the resistances measured and the measured voltage for E, calculate the voltage on each resistor and compare with the measured values.

II. Suppose that you vary the value of R_1 but do not change R_2 and E. How would the voltage on R_1 change as R_1 changes? Confirm your reasoning using two or three different resistances for R_1. What would the voltage on R_1 be in the limit as $R_1 \rightarrow 0$ and $R_1 \rightarrow \infty$?

Replacing the resistor for R_1 with a short piece wire essentially makes $R_1 \rightarrow 0$. And removing the resistor for R_1 essentially makes $R_1 \rightarrow \infty$. Do this to confirm your reasoning.

III. Suppose that you change the resistances of R_1 and R_2 but keep their sum constant. How would the voltage on R_1 change as the resistance of R_1 changes?

Inserted into the solderless circuit board on your work station, you will find a small box-shaped component looking like the figure in Fig. 2. Carefully pull this component from the board and measure the resistance between all possible pairs of wires extending from the component. Try to figure out some arithmetic relationship between the resistances. Using a small screwdriver, turn the shaft extending from the top of the component and repeat the resistance measurements. Note what resistances changed and what resistance did not change.

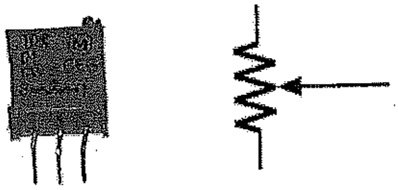


Figure 2: An example of a variable resistor. On the left is the actual device, and the circuit diagram symbol is shown on the right.

The component you have examined is an example of a variable resistor often referred to as a potentiometer or pot, for short. Essentially it is two resistors in series. The resistance between the outer ends is fixed. The resistance between the center wire and an end wire can be changed by turning the shaft on top of the pot. However, the sum of the two resistances equals the resistance between the ends. Symbolically, a pot is shown as a resistor with an arrow (Fig. 2, right). The arrow denotes the variable connection to the pot. The variable connection is called the wiper.

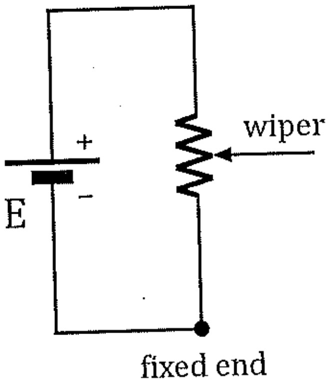


Figure 3: The use of a variable resistor and a fixed voltage source to produce a variable voltage source.

In Fig. 3 the two fixed resistors R_1 and R_2 in Fig. 1 are replaced with a pot. The voltage between the wiper and either of the fixed ends can now be varied continuously by turning the shaft on top of the pot.

Connect the circuit shown in Fig. 3 using about 5V for **E**. Measure the voltage between the wiper and the fixed end. As you turn the shaft on the pot, explain why the voltage you measure must fall between 0 and **E** volts.

IV. A variable voltage in the range of 0 to **E** volts could be produced with any pot regardless of its total resistance. However, the current supplied by the power supply depends on the resistance of the pot. For example, the current produced by a 10V supply and a 10kΩ pot is 1mA. Were the resistance 100Ω the current increases to 100mA. All power supplies have a limit to the current they can provide and the builder of a variable voltage source must be aware of the limitation. It could well be that a 10V power supply could not provide the 100mA for a 100Ω pot.

V. Suppose that you connect a resistor R_L in parallel with R_1. Would you expect the voltage on the parallel combination to increase or decrease? Confirm your reasoning by connecting a resistor in parallel with R_1 and measuring the voltage across the parallel combination. Using measured values of the resistances involved and the source voltage, calculate the voltage on the parallel combination and compare with the measured value.

You must always be aware of the resistance of anything you connect in parallel with one of the resistors in a voltage divider because the presence of the resistor will cause the voltage to change.

VI. Here is an exercise for you to work and to check experimentally. Consider a voltage divider with **E** = 10V, R_1 = 22kΩ and R_2 = 10kΩ. If a 10kΩ pot is connected in parallel with R_1, calculate the range of voltages that can be produced between the wiper and a fixed end of the pot? Confirm your calculations experimentally.

Part II: Kirchhoff's Loop Rule

The circuit in Figure 4 is called a Wheatstone bridge. It is a relatively simple circuit having considerable use in electronics. Assemble the circuit using $R_1 = R_4 = 10$ kΩ, $R_2 = R_3 = 22$ kΩ, $R_5 = 47$ kΩ, and V = 10 V. After the lab instructor has checked your circuit, measure the voltages across each resistor.

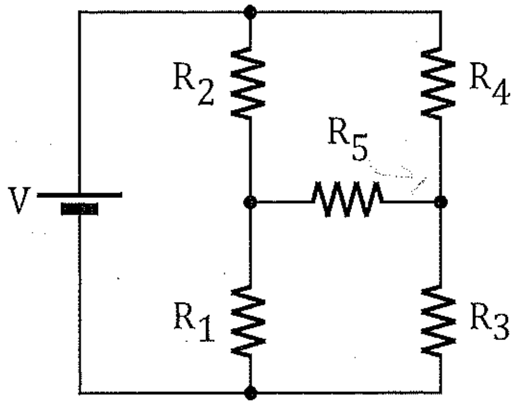


Figure 4: A

Identify five (5) loops in the Wheatstone bridge circuit and, using a voltmeter, verify Kirchhoff's loop rule by showing that the voltages sum to zero in each of the five loops.

For your report for this exercise, use Kirchhoff's rules to calculate the voltages. Note that there are **no series connections** and **no parallel connections** in this circuit.

References

1. Kirchhoff's rules are discussed in most general physics texts, for example, ***Principles of Physics***, Raymond A. Serway, Saunders Publishing.

LabVIEW CONCEPTS

Motivation

The **LabVIEW** programming language is the industry standard for data acquisition, data analysis, and for process control. Accordingly, skill in using **LabVIEW** is valued by employers. **LabVIEW** differs from traditional programming languages in that it is graphical rather than text-based. More importantly, it is "data flow" rather than sequential in nature. This means that execution of a program subset is initiated by arrival of all necessary inputs (data) rather than by a call from the main program. This facilitates programming for data acquisition, because the programmer does not have to build in unnecessary delays to ensure that all data has arrived from the experiment before proceeding to the next operation.

LabVIEW Concepts and LabVIEW Programming Exercises introduce you to the **LabVIEW** programming environment. The goals are

- to understand the operation of a "canned" program,
- to develop capabilities for modifying a program to suit your needs, and
- to construct simple programs from "scratch".

Objective

To gain experience with the basic tools of the **LabVIEW** programming environment by reconstructing a virtual instrument (VI).

Procedure – Part I

Run the **LabVIEW** program and select "Blank VI" in the appropriate dialog box. Two windows will appear (Figure 1). One is the **Front Panel** window and the other is the **Block Diagram** window. The **Front Panel** portrays instruments and controls, and is the window through which the user interacts with the program. The **Block Diagram** is the window containing the program's source code, which is comprised of objects wired together to perform specific functions.

Prior to doing any exploring, open the "**Help**" window and select "**Show Context Help**" from the menu bar. This will provide information on items you investigate.

Several important features of **LabVIEW** are summarized in a two-sided, laminated, color diagram on your lab table. These diagrams can be very helpful as you gain confidence in using **LabVIEW**.

If the "**Tools Palette**" is not visible, select "**Show Tools Palette**" from the "**View**" menu bar. Select a tool via mouse-click; pressing the tab key returns the setting to "auto-tool" (wrench and screw-driver with green light on). Activate the **Front Panel** window and examine the tools. Activate the **Block Diagram** window and repeat this procedure. Note that different tools are used in the two windows.

While in the **Front Panel** window, display the **Controls Palette** by either right-clicking on any blank spot in the window or by choosing "**Show Controls Palette**" from the "**View**" menu bar. Examine and make a mental note of the items in the palette by placing the cursor on the item (remember the "**Context Help**" window). You will need some of these items in your experiments.

While in the **Block Diagram** window, display the **Functions Palette** by either right clicking on any blank spot in the window or by choosing "**Show Functions Palette**" from the "**View**" menu bar. Examine and make a mental note of the items in the palette by placing the cursor on the item. You will need some of these items in your experiments.

When you place a control or indicator in the **Front Panel** window, **LabVIEW** automatically places a corresponding "**Terminal**" in the **Block Diagram** window. A **Terminal** can only be deleted by deleting its **Function** from the **Front Panel** window. Input (control) and output (indicator) terminals are connected via **Wires** through program execution elements known as **Nodes.** Examples of these can be observed in Figure 1, which is the VI prepared for your use. You will be duplicating this VI in this experiment.

Examine the laminated color diagram at your table to understand the operation of each item in the toolbar on top of the **Front Panel** and **Block Diagram** windows respectively.

Also examine the laminated color diagram at your table to see that wires have different shapes (and colors) according to the data type being passed.

Note the helpful keyboard shortcuts also listed the laminated color diagram at your table.

Procedure – Part II

Open the VI titled "**LabVIEW Exercise 1**" from within the PHY 294 folder on the desktop. Run the program in the continuous mode with the "**Save Data?**" switch turned off. Adjust the two controls and describe in your notebook how the output changes. Describe how each control affects the waveform chart and the x-y graph.

Run the program once ("**Run Continuously**" turned off) with the "**Save Data?**" switch turned on. Save the data to the desktop, or in a folder labeled with your name(s).

Run a spreadsheet program and open the file you just saved. Examine and plot the data there. Does the data look like it did when plotted from within LabVIEW?

Now examine the **Block Diagram** of the "**LabVIEW Exercise 1**" VI. Describe in your own words what the program is doing.

With the "**LabVIEW Exercise 1**" VI still open, create a new VI with a blank **Front Panel** and **Block Diagram**. Your goal is to create a working version of the existing VI without copying anything from the existing VI. Start by placing and labeling the appropriate objects in the **Front Panel** window. Then go to the **Block Diagram** window to generate the source code. Each item in both the **Front Panel** and **Block Diagram** has a helpful "description" which can be accessed by right-clicking on the item. As a rule, you would document your program in these "description" areas as you generate code. Frequently save the VI under a different name, e.g., **My LabVIEW Exercise 1**.

Once you have generated a working version of the VI, print the **Front Panel** and **Block Diagram** windows for inclusion in your report.

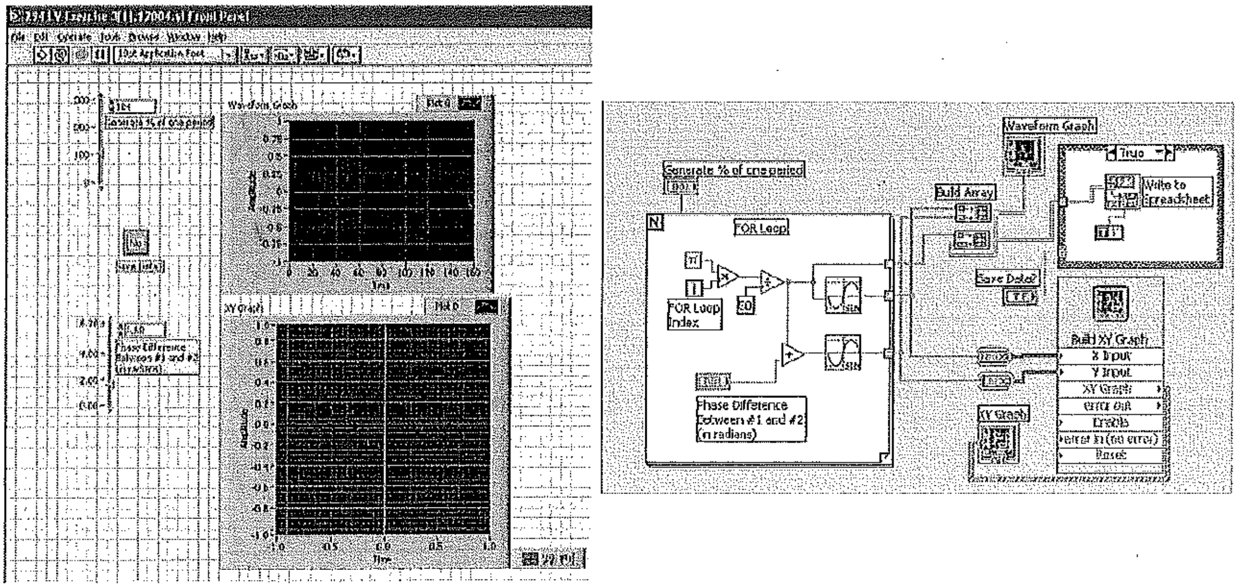


Figure 1: Front panel window (left) and block diagram window (right)

LabVIEW PROGRAMMING EXERCISES

Part 2 of this exercise was designed and written up by Peter Siegfried, Spring 2012

Objectives:

To increase your facility with LabVIEW by modifying an existing VI and to produce a VI on your own.

To introduce the DAQ assistant, and data acquisition functions of LabVIEW and generate a plot with noise from a simple voltage divider.

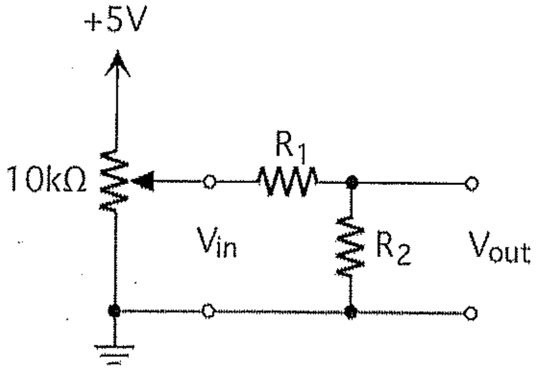
Part 1:

In this exercise, you will be creating a new VI by modifying "LabVIEW Exercise 1". Your new program should generate the function $y = 3x^2$ from $x = 0$ to x_{final} in steps of Δx, where x_{final} and Δx are to be adjustable from the LabVIEW panel. The program should also present the results as an x-y graph and save the data in a spreadsheet format. [Hint: The x^y or power of x function can be found in the Mathematics/Elementary & Special Functions/Exponential Functions/ palette.] Save your program as "LabVIEW Exercise 2" on the desktop or in your PHY 294 folder.

Print the panel and diagram of your VI for inclusion in your report. Also, print out an Excel spreadsheet showing a sample of the data stored by your VI.

Part 2:

For this part you will be constructing a simple voltage divider with a variable input voltage. Figure 1 shows the circuit diagram; we use a 10-kΩ potentiometer connected to +5V to provide the input voltage for the voltage divider. The voltage drop across R_2 is the output voltage, and its relation to the input voltage is given in Figure 1. Choose the values for R_1 and R_2 such that $V_{out} \approx V_{in} / 1000$. The reason for such a small output voltage is so that our data has some amount of scatter and therefore the result of the fit is not as obvious as when all data points lie perfectly along a line. Connect a multimeter to V_{out} and verify that the output voltage varies between 0 and approx. 5mV as you adjust the pot from one extreme to the other.



$$V_{out} = \frac{R_2}{R_1 + R_2} V_{in}$$

Figure 1: Voltage divider circuit with variable input voltage. The output voltage depends on Vin and the ratio of the two resistors.

Instead of the multimeter, we will use the LabVIEW data acquisition system (DAQ) to perform measurement of V_{in} and V_{out} and prepare a graphical representation. Figure 2 shows the NI USB-6211 DAQ unit that will be used for this and future labs. This model has 16 analog inputs that are sampled at a maximum of 250 kS/s single-channel sampling rate. In addition there are two analog outputs, four digital input lines, four digital output lines, and two counter/timers. No external power supply is required as the unit is powered via USB connection. All connections are made through the built-in terminal strip.

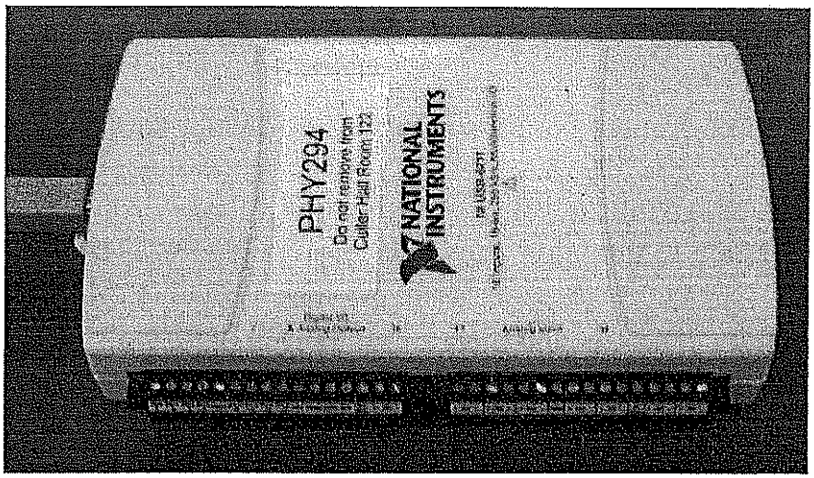


Figure 2: National Instruments USB-6211 DAQ unit.

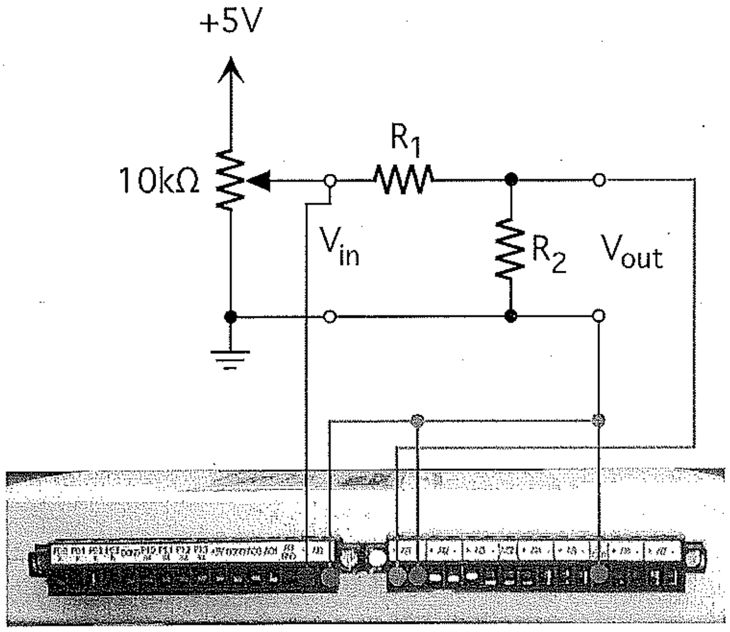


Figure 3: National Instruments USB-6211 DAQ unit connected to the voltage divider circuit.

Your first connection should always be from the ground ('GND' or common) line on your breadboard to terminal 28 ('AIGND') on the DAQ terminal strip. That guarantees that your circuit and the DAQ system both have the same reference voltage for zero volts. Next connect AI0 (terminals 15 and 16) to V_{in} and ground (Figure 3). Channel AI1 (17, 18) will be connected to V_{out} and ground so as to measure the voltage across R_2.

The VI which you will construct will measure the input voltage, V_{in}, as well as the output voltage, i.e. the voltage drop over R_2 and plot V_{out} vs. V_{in}. A line of best fit will also be created for the data. Start by creating an empty while-loop (Programming -> Structures) on the block diagram and use a button on the front panel that stops the loop. To gather the data use an Express DAQ Assistant (Express -> Input -> DAQ Assist) and configure the data acquisition unit. In the Create... window click on *Acquire Signals, Analog Input, Voltage.* When you get to select the AI-channel, you will select AI0 and AI1 simultaneously (shift-click), then click *Finish*. In the *Acquisition Mode* select *1 Sample (on demand)*, then click OK. In order to access the two channels, you need to place a signal splitter (Express -> Sig Manip -> Split Signals) and then expand it to two outputs by dragging on the bottom side. The upper output is channel AI0 (V_{in} in Fig. 3), and the lower output is channel AI1 (V_{out} in Fig. 3),

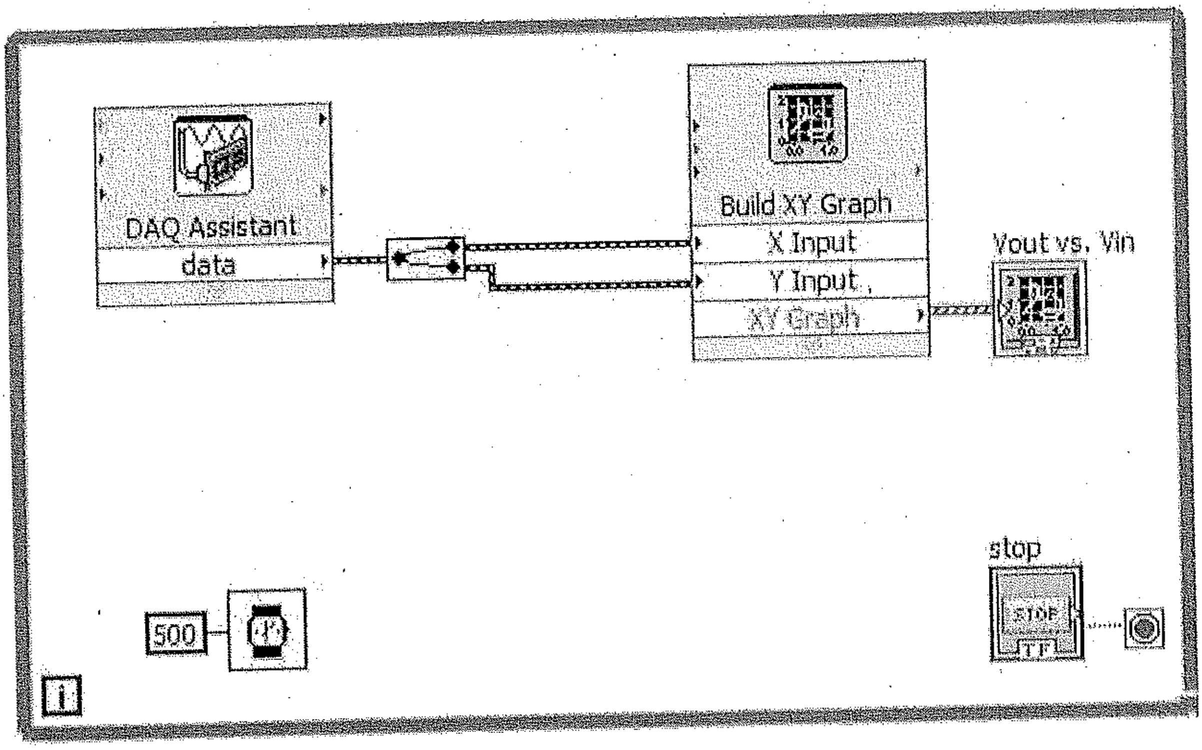


Figure 4: VI to collect input and output voltages and to plot V_{out} vs. V_{in}.

Next place an Express XY Graph on the front panel (Express -> Graph Indicators -> XY Graph). Switch back to the block diagram and move the Build XY Graph icon and the XY graph inside the loop, then right-click on the express icon, select *Properties* and uncheck *Clear Data at each Call.* The graph will plot V_{out} vs. V_{in}, thus AI0 will need to be connected to the X-input, and AI1 to the Y-input of the Build XY Graph express icon. You might want to add a 500-ms time delay (Programming -> Timing -> Wait) to limit the number of data points that are accumulated as the vi runs. Figure 4 shows what your vi should look like by now. Use the same approach you did for the previous LabVIEW exercise to save your data to a spreadsheet file once the loop is terminated, i.e the stop button is pushed.

Before going on and adding the linear fit to the vi give it a test drive. As the vi runs, adjust the input voltage with the pot. As you adjust the pot data points should appear on your XY graph. If instead line elements fill your graph, right-click in the plot legend (Plot 0) and from the Common Plots menu select the one that shows data points not connected by lines. The plot should be "noisy", i.e. the data points should scatter somewhat and not exactly form a straight line.

Now let's add the curve fitting to the vi. We'll use the express curve-fitting vi (Express -> Signal Analysis -> Curve Fitting). The vi must be placed outside the while-loop; double-click the icon and click on the Linear radio button (top left), then click OK. Before the x and y-data leave the loop, they will have to be converted from dynamic data to a single scalar (Express -> Signal Manipulation -> From DDT). Make sure that the tunnel leading the x and y-data out of the while loop is indexing (wire turn to bold wire when it emerges from the loop). The x and y-values must then be connected to the location and signal inputs, respectively. As you connect the lines to the inputs, they are automatically converted to dynamic data. Complete the VI by adding numerical indicators to the slope and intercept outputs and place another express XY-graph on the front panel that is used to display the best fit to the data.

Extra Credit Challenge: Construct a graph that shows both the measured data and the best fit line in the same graph. Once working, print the panel and diagram of your VI for inclusion in your report.

INVERTING AND NON-INVERTING AMPLIFIERS

Objectives

1. To confirm the basic concepts of an operational amplifier.

2. To construct an inverting voltage amplifier and a non-inverting voltage amplifier using an operational amplifier.

Background

Generally, the output of a transducer is not in the proper electrical form for the application at hand. For example, a millivolt signal from a thermocouple (a transducer) must be amplified to accommodate an analog-to-digital converter requiring inputs in the range of 0 to 5 volts. On other occasions a signal might have to be differentiated or integrated, or a voltage may have to be related to a current. The operational amplifier is an electronic device designed to perform these signal-conditioning requirements.

There are many brands of operational amplifiers (op-amp, for short) but here we are concerned only with the Type 741 which serves general purpose applications. It measures about 0.5 cm x 1 cm and has eight connecting pins (Figure 1). Inside the package is a rather complicated electronic circuit consisting of transistors, resistors, and capacitors. To use the device we need not know the details of the circuitry but need only know the function of the eight pin connections.

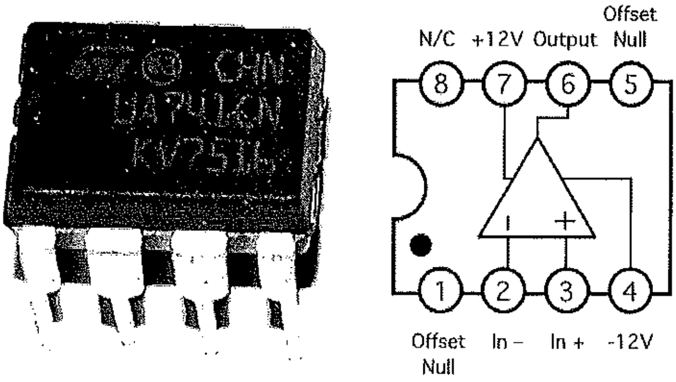


Figure 1: Chip (left) and pin connections (right) of the type 741 op-amp. The small dot on the side of the notch marks pin 1.

The op-amp is powered by +12 V connected to Pin 7 and -12 V connected to Pin 4. It is important that these voltages be the same and be stable. However, voltages in the range of 12 to 18 V will generally be satisfactory. Input signals are connected to Pins 2 and 3 and the output signal is obtained from Pin 6. Pin 8 is not used at all. Let's defer explanation of the offset null connections (Pins 1 and 5) and see how the op-amp functions as a voltage amplifier.

Symbolic representation of the op-amp is shown in Figure 2. The power connections are not shown but bear in mind the op-amp must be powered in any application. Suppose that two voltages V_+ and V_- are connected to the op-amp as shown in Figure 2. Both V_+ and V_- are measured relative to a common connection (ground). There will appear at the output a voltage V_O (relative to common) which is given by

$$V_o = A_o(V_+ - V_-) \tag{1}$$

where A_O is called the open circuit gain. For the Type 741 op-amp, A_O is about 10^5. The large amplification is a general feature of op-amps. Since the output voltage cannot exceed the ±12 V provided by the power supply this means that the voltage difference on the inputs, $V_+ - V_-$, cannot exceed 120 mV. Keep this in mind; the voltage between the input terminals is always very small and for most applications will be negligible compared to other potential differences in the circuit utilizing the op-amp.

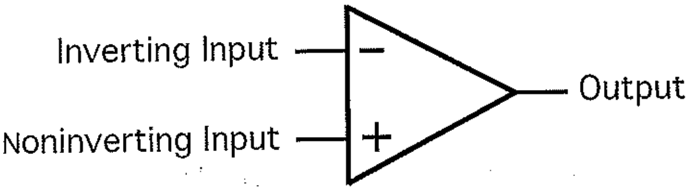


Figure 2: Symbolic representation of an operational amplifier.

There is an internal resistance associated with the input of the op-amp. Generally, this input resistance is very high, about 1 M∧ for the Type 741 op-amp. In practical terms this means that very little current enters the input terminals and for most applications this current is negligible compared to other currents in the circuit.

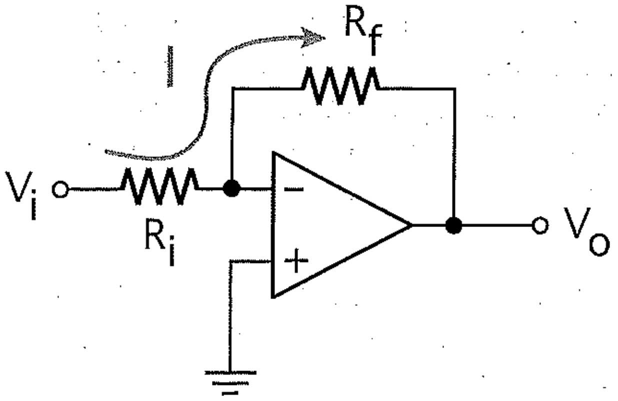


Figure 3: Op-amp with negative feedback configured as an inverting amplifier. Voltages are measured with respect to ground unless specified otherwise.

The circuit in Figure 2 is a very high-gain, but unstable amplifier; its gain will fluctuate. By "feeding back" a portion of the output to the input such that the polarity of the signal fed-back is exactly opposite to the input signal we can dramatically improve the stability. The price for stability is a reduction in gain. The circuit shown in Figure 3 is a practical, stable voltage amplifier[1] for amplifying either AC or DC voltages. We assume a voltage V_i is applied to the input terminal. Voltages usually are measured with respect to ground (a.k.a. as common). This gives rise to a current I having the direction shown. Assuming no current is directed into the op-amp terminal because of the high internal resistance, the current through R_i and R_f is the same. Applying Kirchhoff's loop rule to a loop containing V_i, R_i, and the – and + inputs we have

$$V_i - IR_i + V_d = 0 \tag{2}$$

where V_d is the voltage between the – and + inputs. Picking a loop involving V_o, R_f and the voltage between the – and + inputs and applying Kirchhoff's loop rule we have

$$V_o + IR_f + V_d = 0. \tag{3}$$

According to previous arguments, $V_d = 0$ and we can write

$$\frac{V_o}{V_i} = -\frac{R_f}{R_i} \tag{4}$$

$\frac{V_o}{V_i}$ is the voltage gain which we label A_V. Note that to within the approximations used, A_V depends only on the two external resistors. The minus sign tells us that the phase of the output is exactly opposite to the phase of the input. If the input terminal is positive with respect to ground then the output terminal is negative with respect to ground. This is the reason for calling this amplifier an "inverting amplifier." Choosing accurate, stable resistors for R_i and R_f produces a stable amplifier.

The circuit in Figure 4 is a slight variation of the circuit shown in Figure 3. It is called a non-inverting amplifier because its output has the same phase as the input. To see how the amplifier functions, suppose that a positive voltage V_i (relative to ground) is applied to the + input of the op-amp. Because the signal on the + input of the op-amp is positive, a positive signal appears at the output. The positive output signal gives rise to a current I in the feedback resistor R_f. Assuming that the current into the - input of the op-amp is negligible the current in R_i is also I.

Applying Kirchhoff's loop rule to a loop containing V_o, R_f and R_i we have

$$V_o - IR_f - IR_i = 0 \tag{1}$$

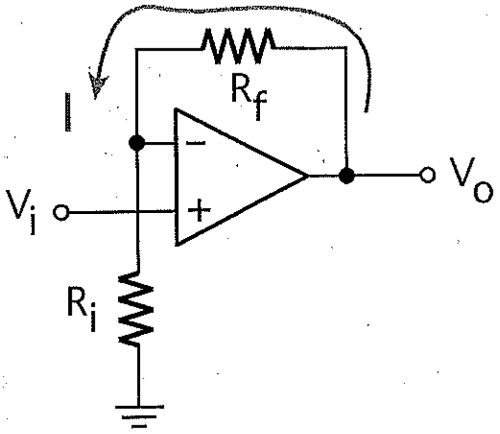


Figure 4: Op-amp configured as a noninverting amplifier. Note that crossing lines do not represent an electrical connection unless there is a dot symbolizing a solding joint.

Picking a loop involving V_i, the voltage between the - and + terminals and R_i and applying Kirchhoff's loop rule we have

$$V_i - V_d - IR_i = 0 \tag{2}$$

According to previous arguments, $V_d = 0$ and we can write

$$\text{Voltage gain} = A_V = \frac{V_o}{V_i} = \frac{R_i + R_f}{R_i} = 1 + \frac{R_f}{R_i} \tag{3}$$

Within the approximations used, the voltage gain (A_V) depends only on the two external resistors R_f and R_i. The fact that the gain is positive tells us that the phase of the output is the same as the phase of the input. If the input terminal is positive with respect to common then the output terminal is positive with respect to common. This is the reason for labeling this circuit as a "noninverting amplifier." For the same combination of R_i and R_f the gain of the noninverting amplifier exceeds the gain of the inverting amplifier by unity. While the magnitude of the gain of the inverting amplifier can be less than unity the gain of the noninverting amplifier is always greater than unity.

Because the input signal is connected directly to the + input of the op-amp the input impedance of the non-inverting amplifier tends to be fairly high, in the megohm range, for example. This is very distinct from the inverting amplifier where the input impedance is essentially the resistance of R_i. The non-inverting amplifier is especially useful when the input transducer has high impedance. In a subsequent experiment, you will take advantage of the high input impedance of a non-inverting amplifier to build an electronic thermometer using a thermocouple as a transducer.

Procedure

The object of this exercise is to build and test both inverting amplifier and noninverting amplifiers with a gain magnitude of 3 to 4. Both amplifiers will be assembled on the breadboard and connected to a potentiometer via a buffer; the circuit diagram is shown in Figure 5. Measure the output voltages V_{out1} and V_{out2} for input voltages V_{in} ranging from 0 V to +5 V.

You have some freedom in selecting the resistors that determine the gain. However, there are limits. The feedback resistor R_2 or R_3 should not be larger than about 1 MΩ, and the input resistor R_1 or R_4 should not be smaller than about 1 kΩ.

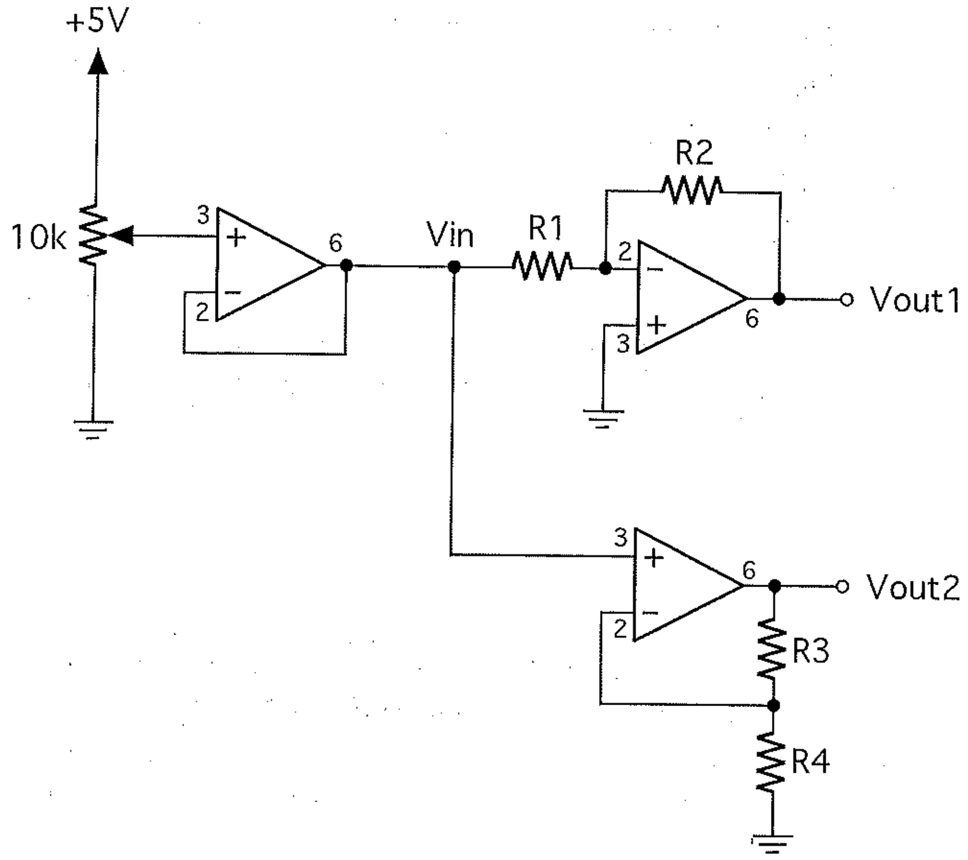


Figure 5: Experimental setup for measuring the gain of the amplifiers. The op-amp with R_3 and R_4 is a noninverting amplifier drawn the conventional way, while the diagram in Figure 4 is drawn to stress the similarity to the inverting amplifier.

The input voltage is controlled manually by the variable resistor. Even though a general purpose multimeter works well for measuring V_{in} and V_{out}, we will again be using the LabVIEW data acquisition system (DAQ) to measure V_{in}, V_{out1} and V_{out2} simultaneously. (Figure 6).

Your first connection should always be from the ground ('GND' or common) line on your breadboard to terminal 28 ('AIGND') on the DAQ terminal strip. That guarantees that your circuit and the DAQ system both have the same reference voltage for zero volts.

Now connect AI0 (terminals 15 and 16) to point V_{in} and ground (Figure 5). Channels AI1 (17, 18) and AI2 (19, 20) will be connected to the op-amp outputs V_{out1} and V_{out2} and ground.

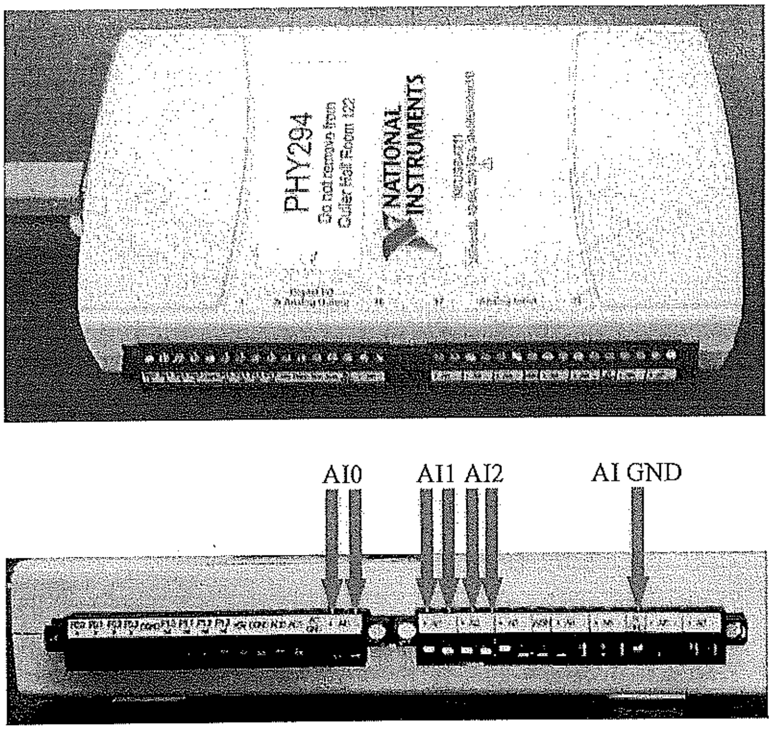


Figure 6: National Instruments USB-6211 DAQ unit. The arrows show the the + and – connections to analog inputs channel 0 through 2 and the analog input ground.

The VI which you will construct is to measure the input and output voltages of the inverting and noninverting amplifiers and save the data into a file. Once saved, the data is analyzed with a spreadsheet. Start by placing an empty while-loop (Programming -> Structures) on the block diagram and use a button on the front panel that stops the loop.

Next use the DAQ assistant (Express –> Input –> DAQ-Assistant) to configure the data acquisition unit. In the Create... window click on *Acquire Signals, Analog Input, Voltage.* When you get to select the AI-channel, you will select AI0, AI1, and AI2 simultaneously (shift-click), then click *Finish.* In the *Acquisition Mode* select *1 Sample (on demand),* then click OK. In order to access the three channels, you need to place a signal splitter (Express -> Sig Manip -> Split Signals) and then expand it to 3 outputs by dragging on the bottom side. The upper output is channel AI0 (V_{in} in Fig. 5). The other two outputs are channels AI1 and AI2 (V_{out1} and V_{out2}), in that order.

Next place two Express XY Graphs inside the loop, then right-click on the icon and select *Properties* and uncheck *Clear Data at each Call.* The graphs will plot V_{out1} vs. V_{in} and V_{out2} vs. V_{in}, thus AI0 will need to be connected to both X-inputs, and AI1 and AI2 go to each Y-input. Alternatively, you can use one Express XY Graph and plot V_{out1} and V_{out2} vs. V_{in}. Use the same approach you did for the LabVIEW exercises to save your data to a text file once the loop is terminated. Now you are ready for a test drive. As the vi runs, adjust the input voltage with the pot.

Import your data into a spreadsheet and make a graph of output voltage V_{out} versus input voltage V_{in} and fit a straight line to the data using the regression analysis routine. Use EXCEL's regression analysis (not the trendline), determine the slope and intercept. The slope of the line gives the gain which can be compared with the theoretical value of $-\frac{R_2}{R_1}$ for the inverting amplifier and $1+\frac{R_3}{R_4}$ for the non-inverting amplifier.

When finished with the DC measurements (and time permitting), replace the DC source (i.e. the potentiometer) with a function generator and observe the output of the inverting op-amp, V_{oiut1} with an oscilloscope as you increase the frequency from 100 Hz to 1 MHz. Repeat this after you change R_2 so that a gain of –100 results. Suggestions for measuring the gain are given below.

Your report should include:

1. Plots of V_{out} versus V_{in} and a best fit line for both the inverting and non-inverting amplifiers.

2. A hard copy of the spreadsheets showing the data and regression analyses.

3. An error analysis of the results.

4. A graph of gain versus the logarithm of the frequency for the AC source and a brief interpretation of the graph (if performed).

Some useful advice

In constructing circuits on the circuit board for this and later experiments, you should keep in mind that neatly wired circuits function better than those which are sloppily wired (of course, this assumes the wires go to the correct places). The wires should fit between the two connection points without great loops of wire standing above the board. Bend the stripped ends so that they enter straight into the connection points. A connecting wire or a circuit component should never be routed over an integrated circuit chip. Spending a few extra minutes constructing a neat circuit could well save several tens of minutes trying to find out what is wrong with it later.

For the measurements of gain as a function of frequency, the d.c. input should be replaced with an a.c. source capable of producing frequencies between 100 Hz and 1 MHz. The a.c. voltages are measured with an oscilloscope rather than a d.c. voltmeter. If you have an oscilloscope with two input channels then you can simultaneously display both the input and output signals. The gain should be measured at enough frequencies so that you can produce an accurate representation of the gain versus frequency curve. A qualitative relationship between frequency and phase shift should be noted as your measurements proceed.

The output voltage cannot swing higher or lower than the power supply voltages employed. For example, if you are using +12 V and -12 V the output cannot go higher than +12 V or lower than -12 V. With a gain of –10 this means that your input signal cannot exceed ±1.0 V in order that the output signal be a scaled reproduction of the input signal. When the output reaches the limit established by the power supply voltages we say the amplifier is saturated. The effects of saturation may be observed in the form of "flattening" of the tops and bottoms of a sine wave if the input signal is made too large. Be sure you take the time to observe the effect of saturation. Saturation in a HiFi amplifying system results in distortion of the sound produced.

References

1. The inverting amplifier using an operational amplifier is discussed in the following electronics textbooks:

Electronics with Digital and Analog Integrated Circuits, Richard J. Higgins, Prentice-Hall (1983)

Analog Electronics for Scientific Applications, Dennis Barnaal, Waveland Press, Inc., Prospect Heights, IL 60070 (1989)

The Art of Electronics, Paul Horowitz and Winfield Hill, Cambridge University Press, 2nd ed. (1989)

Introductory Electronics for Scientists and Engineers, Robert E. Simpson, Allyn and Bacon, 2nd ed. (1987)

An Introduction to Modern Electronics, William L. Faissler, Wiley, (1991)

THERMOCOUPLE AMPLIFIER

Objective

To design, construct, and test an electronic thermometer using a copper-constantan thermocouple and a two-stage amplifier employing operational amplifiers. To use LabVIEW to record the output of the amplifier every 2°C in the temperature range 28°C to 90°C.

Background

One end of a wire electrically connected to the end of another wire forms an electrical junction (Figure 1). If the wires are of different composition a potential difference related to the temperature of the junction will develop between the free ends of the wire. Such an arrangement is called a thermocouple. If the relationship between temperature and potential difference is known the thermocouple can be used as a thermometer. Because the output is electrical, a thermocouple is extremely useful for measuring temperatures in remote areas because the temperature information can be transmitted by wires.

The potential difference produced by a thermocouple is in the range of millivolts. Measuring this small potential difference accurately requires a sensitive voltmeter that does not draw appreciable current. Because the non-inverting amplifier has high input impedance it satisfies this latter requirement very nicely.

The thermocouple used in this experiment has one wire made of copper and the other made of constantan which is an alloy of equal parts of copper and nickel. Each Celsius degree change of temperature of the thermocouple junction produces 0.0427 millivolts change in the voltage on the open ends. Thus a temperature change of 100°C produces a total voltage change of 4.27 mV.

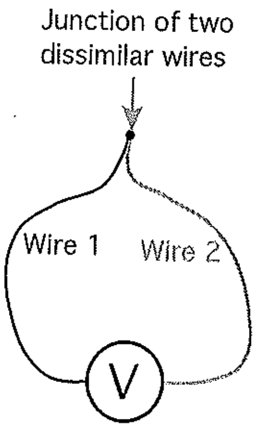


Figure 1: A thermocouple connected to a voltmeter.

In this experiment you want to construct an amplifier having a gain of 1000 so that the output voltage is in the range of volts. In principle, you could do this with a single amplifier. However, it is difficult to build a stable, single-stage amplifier having a gain of more than 100. This problem can be circumvented by connecting single stages in series. This is called cascading. The output of one amplifier forms the input of a following amplifier. If the gains of two cascaded amplifiers are A_1 and A_2 then the overall gain (A) of the cascaded amplifiers is

$$A = A_1 A_2 \tag{1}$$

Take a moment to convince yourself of this result. If $A_1 = A_2$ and $A = 1000$ then $A_1 = \sqrt{1000}$.

If the op-amps employed had the ideal characteristics assumed then the output of the amplifier would be zero volts when the input was zero volts. Generally, this is not the case. A non-zero output when the input is zero is called an offset. If the gain in the amplifier is relatively low this often is of no consequence. This was the case in the previous experiment where the amplifier gain was less than 10. In this experiment the overall gain is 1000 and the offset is a consideration. To overcome this problem manufacturers of op-amps include an offset null. The offset null is an adjustment that allows you to set the output to zero volts when the input is zero volts. Or in other circumstances where you might want an offset voltage you can use the offset null adjustment to produce the desired offset voltage. Pins 1 and 5 on the Type 741 op-amp are labeled offset null. As shown in Figure 2, the fixed terminals of a 10 kΩ variable resistor are connected to Pins 1 and 5. The sliding contact (wiper) of the variable resistor is connected to -12 volts. In the past, some students have been interested to use a Type 747 op-amp, which has two op-amps on a single chip. The offset null connections for the dual 747 are pins 3 and 14.

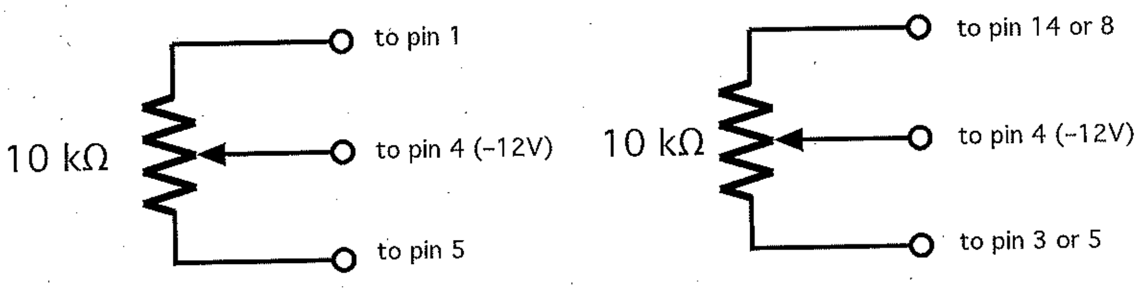


Figure 2: Offset null connections for op-amps type 741 (left) and for type 747 (right)

Procedure

The circuit for the two-stage amplifier used in this experiment is shown in Figure 3. After calculating the values for the resistors R_1, R_2, R_3 and R_4 build the circuit shown in Figure 3[1]. The same guidelines for the resistors as for the previous experiment apply, i.e. the feedback resistor R_2 or R_4 should not be larger than about 1 MΩ, and the other resistors, R_1 or R_3, should not be smaller than about 1 kΩ. Let both amplifier stages have the same gain, approximately. Don't forget the power supply connections. You have the option of offset null connections in both stages. However, you probably can get by with an offset null on the first stage. When the amplifier has been wired and checked, connect a voltmeter to the output and connect the input terminal to common. Connecting the input to common establishes the input voltage at zero. Adjust the offset null variable resistor until the output is zero volts.

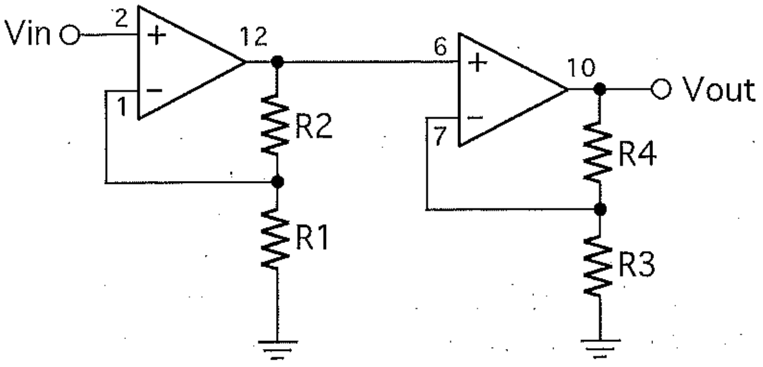


Figure 3: Thermocouple amplifier. Pin numbers are for the 747 type op-amp.

In practice, a thermocouple is used to measure the temperature of some environment relative to a reference temperature. Usually this reference temperature is the ice point, 0°C, so that the temperature recorded is the Celsius temperature of the environment. This is achieved by connecting two thermocouples together as shown in Figure 4. When connected as shown the two thermocouples behave like two batteries connected with like poles together. If the voltage developed by each "thermocouple battery" is the same then the voltages across the open connections add to a net zero. For the thermocouple system the voltage across each thermocouple is the same if both thermocouples are at the same temperature. If both thermocouples are placed in the ice bath the voltage across the open ends should be zero. Then if the measuring thermocouple is placed in a different environment the voltage across the open ends is the difference in voltages produced by the two thermocouples. Relating this voltage to the temperature yields the temperature of the environment. The environment in this experiment is a water bath whose temperature will

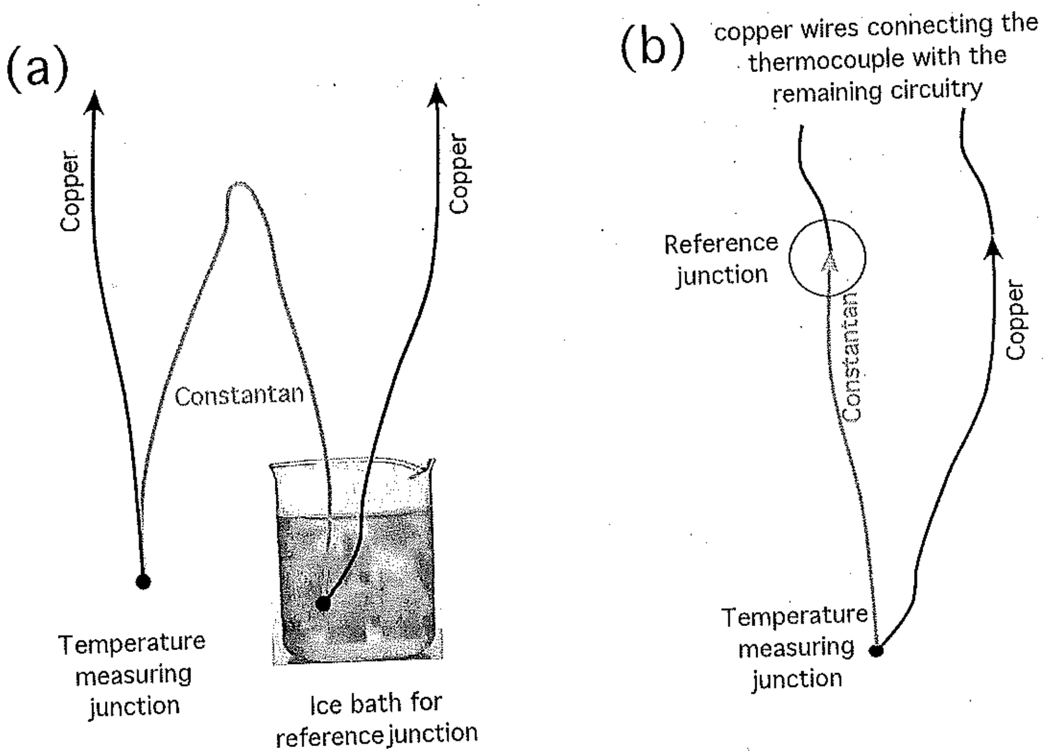


Figure 4: (a) A thermocouple pair with the reference junction in an ice bath for accurate reference temperature. (b) When a single thermocouple is used the junction between the constantan wire and the copper wire of the remaining circuitry becomes the reference junction. The reference temperature in this setup is not well-defined, thus this measurement is not as accurate as that in (a). However, for simplicity we will use setup (b) for this lab.

be changed from room temperature to 90°C. For simplicity we will use only one thermocouple with the reference junction at room temperature (Figure 4). The water is contained in a small beaker and is heated with a hot plate. The temperature of the water will be recorded with a thermometer. The thermocouple should be in close proximity to the temperature-sensitive part of the thermometer.

In this exercise you are going to use LabVIEW and the National Instruments (NI) data acquisition board (DAQ) as a voltmeter to measure the output of your thermocouple amplifier. Your first task is to construct the VI for the voltmeter. Following is a guide for constructing and testing the VI:

Select Front Panel

- Right Click to get Controls Palette
- Select Num Inds, select Meter and drag the Meter to the Front Panel
- Select Num Inds, select Num Inds and drag a Num Ind to the Front Panel

Select Block Diagram

Right Click to get Functions Palette

Select Exec Ctrl and drag a While loop to the Block Diagram

Note that the While loop contains a Boolean operation and a Boolean button on the Front Panel. Pressing the button on the Front Panel will stop the program.

Move the Meter and Num Ind icons inside the While loop

Right Click to get Functions Palette

Select Input then DAQ Assist and drag DAQ Assist icon inside the While loop. A dialog box is displayed to configure the DAQ

Select Analog Input

Select Voltage

When Supported Physical Channels comes up, select ai0, Click Finish

Another dialog box is displayed

Select Acquire 1 Sample, click OK

Right Click to get the Functions Palette

Select All Functions, then Time & Dialog, then Wait (looks like a wrist watch). Drag the watch to the While loop.

Right Click on the left side of the watch. From the dialog box, select Create Constant. Enter 500 for the constant.

Wire the Data output from the DAQ Assistant to the Meter and Numeric Indicator.

When finished, your Front Panel and Block Diagram should look something like Figure 5.

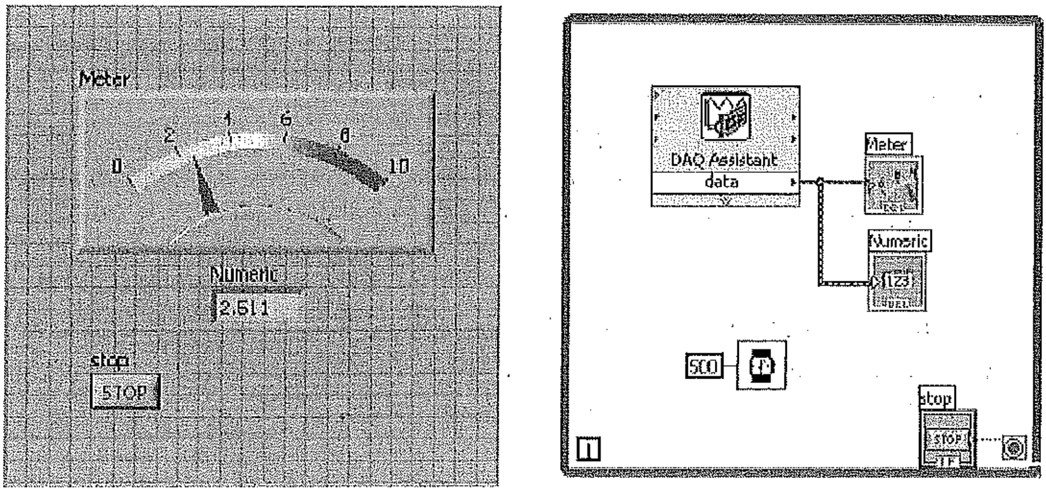


Figure 5: Front panel (left) and block diagram (right) of the VI.

As in the previous experiment, add and Express XY Graph inside the loop so that the VI prepares a graph of your data as the measurement proceeds.

Electrical Connections to the Data Acquisition (DAQ) system

Your first connection should always be from the common (or ground 'GND') line on your protoboard to connector number 28 ('AIGND') on the DAQ system. That guarantees that your circuit and the DAQ system both have the same reference voltage for zero volts. Now you want to measure the voltage difference across your 0-5 V source. Connect the Common (Ground) connection to connector number 16 ('AI0-') and the Output connection to connector number 15('AI0+') on the National Instruments DAQ system. Run the VI you have constructed and verify that the VI is working properly.

As first configured the Numeric Indicator will be displaying 6 figures. You should change this to read 4 figures (3 digits behind period) by right clicking on the Numeric Indicator, clicking Properties, selecting Format and Precision and changing the Significant digits to 4.

With the VI running continuously and with both thermocouples in an ice bath, adjust the offset null to produce an amplifier output of zero. Keeping one thermocouple in the ice bath, put the other one in a beaker of water on the hot plate. Warm the water slowly while taking measurements of temperature and voltage approximately every 2°C. Manually record the measurements in a spreadsheet as you take data and perform a linear regression fit as you proceed.

Your report should include:

1. A graph and linear regression analysis of the experimental data.

2. An explanation of the design of the amplifier.

3. An explanation for using an offset null.

References

1. Textbooks presenting discussions of multi-stage amplifiers include

Electronics with Digital and Analog Integrated Circuits, Richard J. Higgins, Prentice-Hall (1983)

Analog Electronics for Scientific Applications, Dennis Barnaal, Waveland Press, Inc., Prospect Heights, IL 60070 (1989)

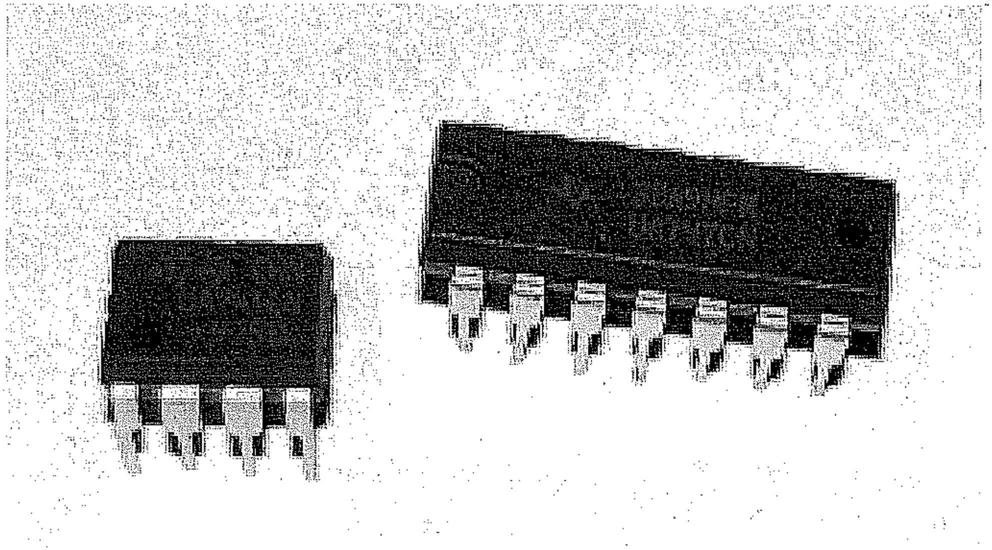
The Art of Electronics, Paul Horowitz and Winfield Hill, Cambridge University Press, 2nd ed. (1989)

Introductory Electronics for Scientists and Engineers, Robert E. Simpson, Allyn and Bacon, 2nd ed. (1987)

An Introduction to Modern Electronics, William L. Faissler, Wiley, (1991)

Apparatus Notes

The Type 747 integrated circuit chip contains two type 741 op-amps in one 14-pin case as shown in Figure 6 below. The two op-amps are identified as A and B. Note carefully that the order of the non-inverting and inverting terminals is different for the two op-amps. It is necessary for both pins 13 and 9 to be connected to +12 V when both op-amps are used.



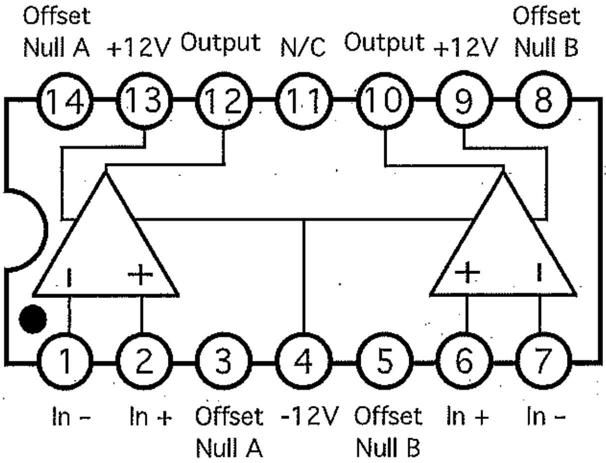


Figure 6: Image of a type 747 (right) in comparison with the type 741 chip (left). The pin connections for the 747 integrated circuit are shown below.

STRAIN GAUGE AMPLIFIER

Objectives

1. To gain experience with the strain gauge as an example of a resistive transducer capable of measuring very small deflections.

2. To explore the utility of a difference amplifier.

Background

In all of the op-amp applications encountered thus far, the input voltage has been measured relative to common (ground). In this experiment, we shall use the op-amp in a circuit in which the output voltage is related to the difference between two input voltages neither of which is at ground potential.

The circuit in Figure 1 is called a Wheatstone bridge. The bridge is said to be balanced when the potential at A equals the potential at B. This balanced condition occurs for $R_1R_4 = R_2R_3$. When the bridge is not balanced there is a potential difference between points A and B. Resistive transducers are often used in a Wheatstone bridge and the potential difference is measured between the points A and B. This potential difference is usually very small and a difference amplifier is often used for amplification.

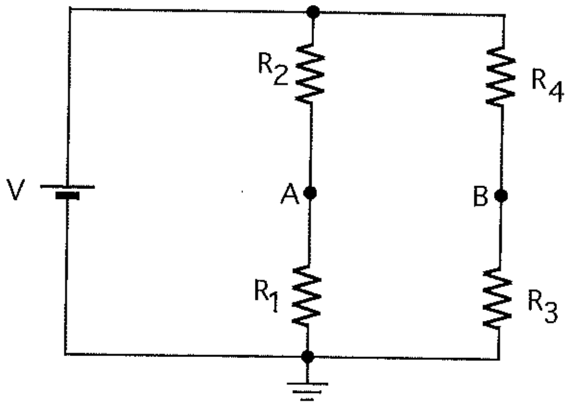


Figure 1: The Wheatstone bridge.

In this experiment, you will use two commercial strain gauges in a Wheatstone bridge circuit having the points A and B connected to the inputs of a difference amplifier. We expect the change in output voltage from the amplifier to be linearly related to the change in resistance produced by a strain gauge. One purpose of the experiment is to test the linearity of the system.

The functioning of the strain gauge as a transducer is based on the fact that the resistance of a wire is related to its resistivity ρ, length L and cross-sectional area A by

$$R = \rho \frac{L}{A} \tag{1}$$

If the wire is stretched (strained) by an amount ΔL and the cross-sectional area does not change significantly, the resistance changes by an amount ΔR given by

$$\Delta R = \rho \frac{\Delta L}{A} \tag{2}$$

Dividing Equation 2 by Equation 1 yields

$$\frac{\Delta R}{R} = \frac{\Delta L}{L} \tag{3}$$

By definition, the ratio, $\frac{\Delta L}{L}$, is the tensile strain in the wire. Therefore, to within the approximation that only the length changes when the wire is stretched, the fractional change in resistance, $\frac{\Delta R}{R}$, is a measure of the tensile strain.

The totality of all effects that occur when the wire is stretched is accounted for by the introduction of a gauge factor which must be determined for each gauge by either the experimenter or the manufacturer. If the gauge factor is denoted by *G*, Equation 3 is modified to read

$$\frac{\Delta R}{R} = G \frac{\Delta L}{L} \tag{4}$$

Strain gauges are made by chemical deposition of a long, folded line of pure nickel onto a plastic substrate which is then cemented to the structure being tested. Strain gauges are available in a wide variety of sizes and shapes to serve in an equally wide variety of applications. Figure 2 (left) shows the configuration of a common style which is used in this experiment.

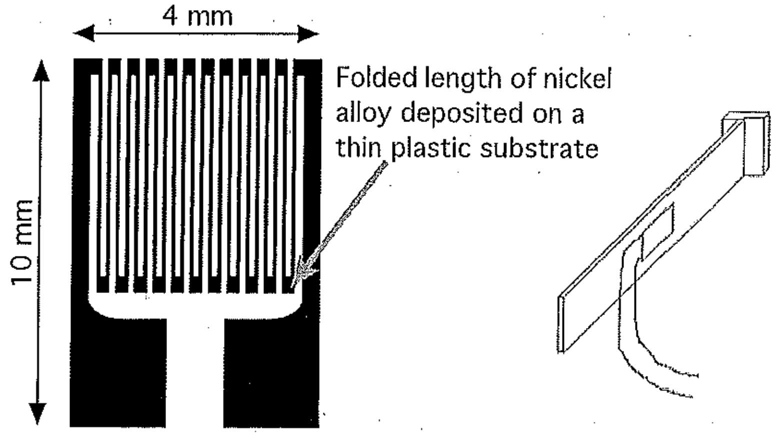


Figure 2: Construction of a commercial strain gauge (left). A strain gauge cemented to the face of a thin strip of stainless steel (right).

It is always necessary to use two gauges since the resistance of the gauge changes with temperature as well as with strain; if the two gauges are connected into adjacent arms of the bridge, the changes due to temperature will be the same for both gauges and will cancel. If possible, the gauges are mounted so that one of them is stretched while the other is compressed by an equal amount. If this is not possible, one gauge is left unmounted but kept at the same temperature of the other for temperature compensation. In the case at hand, the two gauges are mounted on opposite sides of a steel strip (Fig. 2 right).

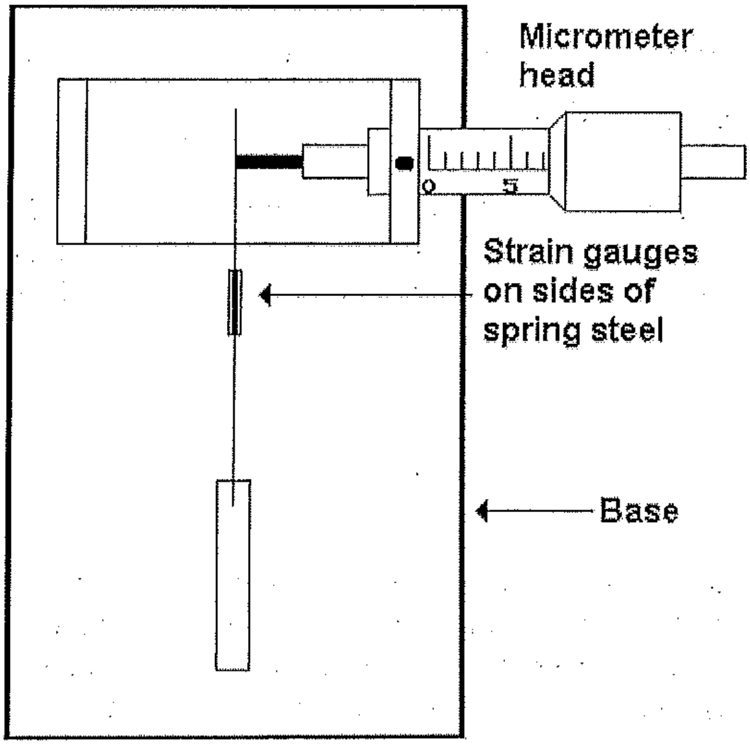


Figure 3: Bending jig for the strain gauge.

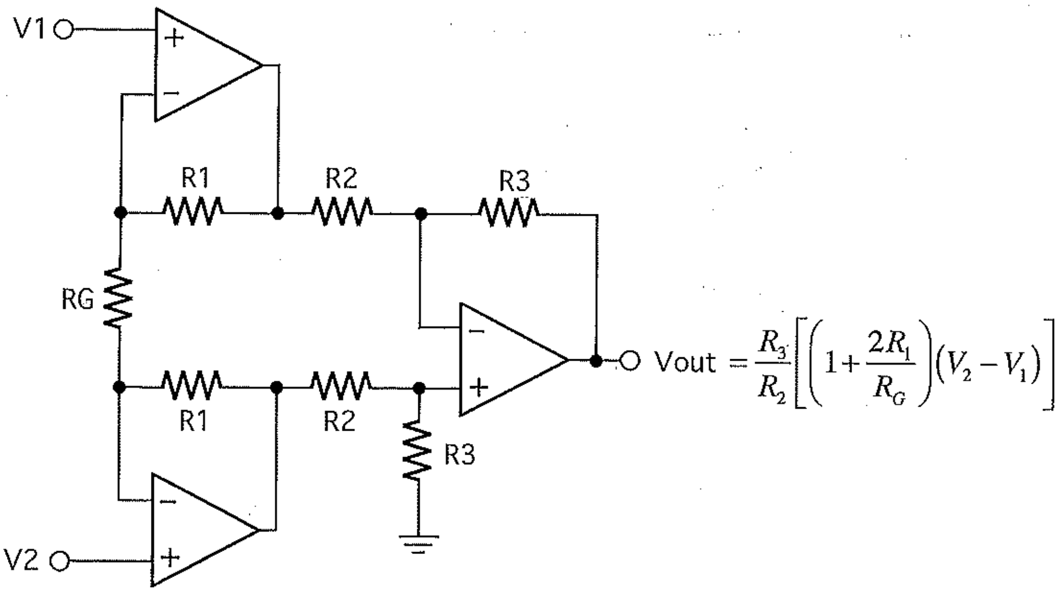


Figure 4: An instrumentation amplifier consists of two non-inverting amplifiers followed by a difference amplifier. The overall gain of the amplifier can be changed by R_G.

The moving head of a micrometer pushing against the end of the steel strip flexes the strip when the micrometer handle is turned (Figure 3). The strain gauges sense the flexing. The object of the experiment is to relate the output of the strain gauge amplifier to the deflection of the free end of the steel strip as measured with the micrometer.

Note that in the Wheatstone bridge circuit (Figure 1) neither point A nor point B is at ground potential. Therefore we cannot use our standard inverting or non-inverting op-amp circuit as these measure the potiential of the input with respect to ground. Instead we will use a so-called instrumentation amplifier, the basic circuit of which is shown in Figure 4. We could just build this instrumentation amplifier with a type 747 and type 741 op-amp, but it is generally hard to match the resistors pairs R_1, R_2, and R_3 precisely. Instead we will be using the AD620, an integrated precission instrumentation amplifier.

Figure 5 shows the block diagram and the pin assignment of the AD620. It comes in a dual-inline 8-pin case, the same type as a 741 type op-amp. The gain of the instrumentation amplifier is determined by a single gain resistor, R_G, that must be connected between pins 1 and 8. The output voltage given by

$$V_{out} = \left[1+\frac{49.4k\Omega}{R_G}\right](V_2-V_1) \qquad (5)$$

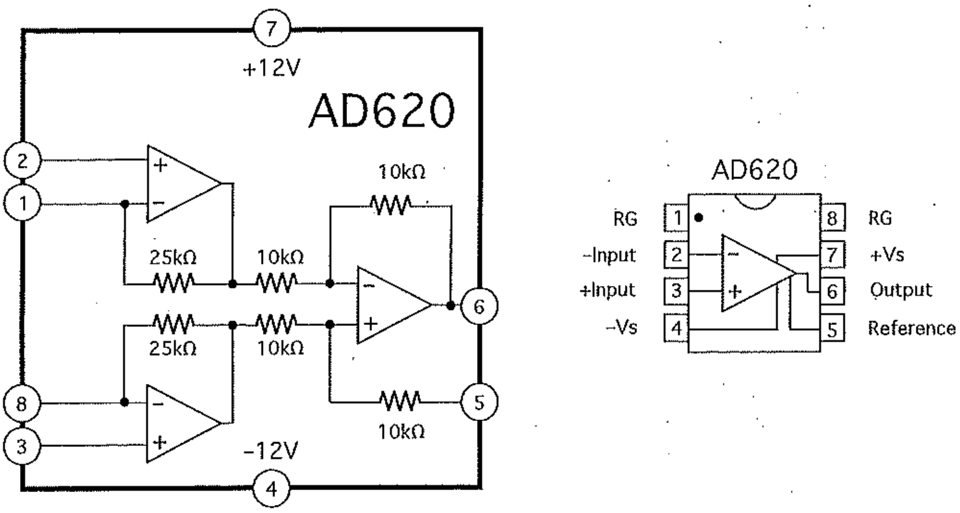


Figure 5: Simplified blockdiagram of the AD620 (left) and pin assignment (right). The AD620 has the same foot print as a regular 741 op-amp. Comparison with the instrumentation amplifier (Fig. 4) shows that if pin 5 is grounded and R_G is added between pins 1 and 8, the AD620 becomes an instrumentation amplifier.

The Wheatstone bridge circuit and amplifier are shown in Figure 6. Note that the 100Ω-pot is included to allow for adjustments making up the difference in the two resistors R_b. Select a value for R_G that will result in an overall gain of 1000.

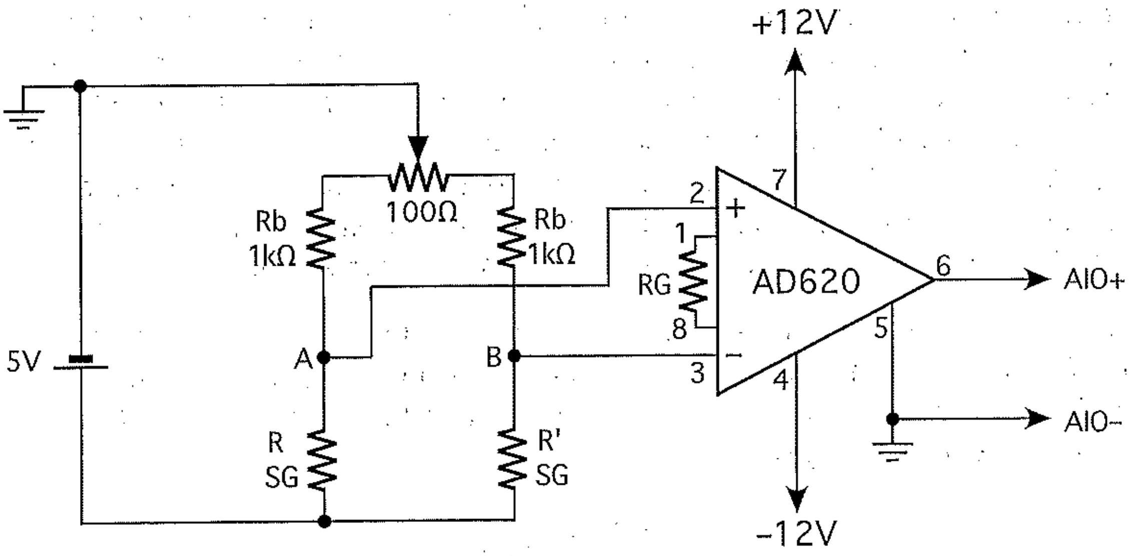


Figure 6: Wheatstone bridge and instrumentation amplifier circuit. R and R' are the strain gauges, and the 100Ω-pot allows to adjust for zero output.

Analysis of the circuit in Figure 6 is simplified by applying Thevenin's theorem[1] to the Wheatstone bridge portion of the circuit. The Thevenin voltage and Thevenin resistance are

$$V_1 = \frac{R_b}{R_b + R}5V \quad R_1 = \frac{R_b R}{R_b + R} \text{ (left half)} \quad \text{and} \quad V_2 = \frac{R_b}{R_b + R'}5V \quad R_2 = \frac{R_b R'}{R_b + R'} \text{ (right half)}$$

The Thevenin voltages are labeled V_1 and V_2 and the Thevenin resistances are labeled R_1 and R_2 and the circuit in Figure 6 reduces to that shown in Figure 7.

The strain gauges are bonded onto opposite faces of the metal strip so that when one changes by an amount $+\Delta R$ the other changes by $-\Delta R$. Accordingly, we can write $R = R_S + \Delta R$ and $R = R_S - \Delta R$ where R_S is the resistance of the unstrained gauge. Because $\Delta R << R_S$ then $R_1 \cong R_2$. With these approximations the output voltage may be written

$$V_o = -2000\frac{\Delta R}{R_b + R_s}5V \tag{7}$$

showing that the output voltage is directly proportional to the change in resistance of the strain gauge, ΔR.

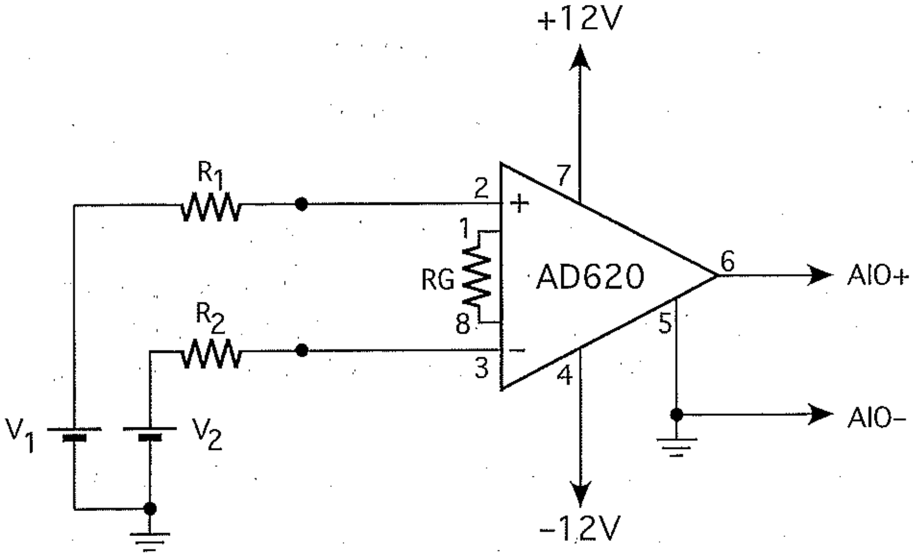


Figure 7: Thevenin's equivalent to the circuit of Figure 6.

Procedure

First build the circuit as shown in Figure 6. The Wheatstone bridge with the strain gauges, the two 1-kΩ resistors and the 100-Ω pot are built on the small breadboard next to the steel strip. The remaining part of the circuit is build on the main breadboard. The micrometer of the bending jig should be adjusted so that the strip is reasonably straight and the micrometer scale recorded as the zero reading to be subtracted from subsequent readings.

The following is a recommended procedure for constructing and testing the amplifier.

1. Measure the resistance of each strain gauge. The resistances should be very close and each about 120 Ω.
2. Connect the bridge to the excitation voltage (+5 V). Measure the voltage across the bridge (A and B) and adjust the 100 Ω variable resistor until the voltage is nearly zero.
3. Connect the bridge to the instrumentation amplifier and measure the output voltage
4. with a voltmeter. Adjust the 100 Ω variable resistor until the output voltage is about zero.
5. After making the electrical connections to the DAQ system (described below) the system is ready for taking measurements.

Having made the preliminary adjustments, the barrel of the micrometer should be moved until it barely flexes the steel strip bearing the strain gauges.

Next construct a VI to record the output and standard deviation of strain gauge amplifier.

On the Block Diagram for a new program, select Functions, Input and drag DAQ Assistant to the diagram.

A screen appears for setting up the DAQ. Select Analog Input, Voltage, ai0, Finish.

Following Finish, a Screen Analog Input Voltage Task screen appears. Select *Differential* and *Acquire N Samples.* The number of samples should be 100 and *Acquire Continuously* should be 10 Hz. This operation will allow you to record 100 values of the amplifier output at a rate of 10 per second.

Right click to get the Functions menu. Select Analysis and drag Statistics to the diagram.

When a Statistics screen appears select Arithmetic Mean and Standard Deviation and press OK.

Wire the Data connection from the DAQ to Signals on the Statistics icon. Right click on Arithmetic Mean and Create a Numeric Indicator. Right click on Standard Deviation and Create a Numeric Indicator.

The VI is now configured to take 100 measurements of the output voltage, compute the arithmetic mean and standard deviation of the 100 measurements, and repeat the process every time you run the program.

Electrical Connections to the Data Acquisition (DAQ) system

Your first connection should always be from the common (or ground 'GND') line on your protoboard to connector number 28 ('AIGND') on the DAQ system. That guarantees that your circuit and the DAQ system both have the same reference voltage for zero volts. Now you want to measure the voltage difference across your 0-5 V source. Connect the Common (Ground) connection to connector number 16 ('AI0-') and the Output connection to connector number 15(AI0+') on the National Instruments DAQ system.

LabVIEW controlling the NI DAQ will record the mean output voltage and standard deviation for selected micrometer readings. Enter each voltage and micrometer reading into an EXCEL spreadsheet and prepare a graph of the data as you proceed.

Your report should include:

1. The spreadsheet of your recorded data with the results of the linear regression.

2. A graph of the measurements and the fitted straight line using the slope and intercept as well as their uncertainties from the regression analysis.

3. A discussion of a strain gauge and the electronic circuitry.

References

Electronics texts that present discussions of Thevenin's theorem include

Electronics with Digital and Analog Integrated Circuits, Richard J. Higgins, Prentice-Hall (1983)

Analog Electronics for Scientific Applications, Dennis Barnaal, Waveland Press, Inc., Prospect Heights, IL 60070 (1989)

The Art of Electronics, Paul Horowitz and Winfield Hill, Cambridge University Press, 2nd ed. (1989)

Introductory Electronics for Scientists and Engineers, Robert E. Simpson, Allyn and Bacon, 2nd ed. (1987)

An Introduction to Modern Electronics, William L. Faissler, Wiley, (1991)

INTEGRATING MAGNETOMETER

Objective

To understand the characteristics of the integrating amplifier by building and testing a magnetometer.

Important Note - For the first time, this lab exercise requires students to measure two different voltages simultaneously using the data acquisition (DAQ) system. One voltage represents the magnetometer response and will be plotted on the y-axis. The other voltage is proportional to the current in the coil and will be plotted on the x-axis.

Background

In most of the exercises thus far, you have used the operational amplifier as a voltage amplifier, a device that converts a small voltage at its input terminals to a larger voltage at its output terminals. In this exercise, you use an operational amplifier to perform a time integral of a voltage. The integrating circuit (Figure 1)[1] is very much like the inverting amplifier except that a capacitor replaces a resistor in the feedback circuit.

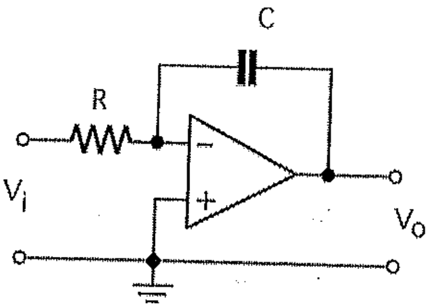


Figure 1: Op-amp configured as integrating amplifier.

Application of Kirchhoff's rules of circuit analysis to the input and output loops of this circuit shows that the current in the resistor is

$$I = \frac{V_i}{R} \tag{1}$$

and that the voltage across the capacitor is

$$V_C = -V_O = \frac{q}{C}. \tag{2}$$

Using the definition of current, $I = \frac{dq}{dt}$, and combining these equations, we find

$$V_o = -\frac{1}{RC}\int_{t_1}^{t_2} V_i \, dt \qquad (3)$$

Equation 3 reveals that the circuit produces an output voltage that is proportional to the time integral of the input voltage. This can be used with a variety of transducers that produce a voltage characterized as the derivative of some physical quantity. In this experiment, the transducer connected to the input of the circuit is a coil containing N turns of wire wound on a disk-shaped form enclosing a cross-sectional area A. If the coil is placed in a magnetic field (B), Faraday's Law requires that a voltage appear across the ends of the coil given by

$$V = -NA\frac{dB}{dt}\cos\Theta. \qquad (4)$$

where Θ is the angle between the direction of the magnetic field and the normal to the plane of the coil. If you place the coil in a magnetic field in such a way that the plane of the coil is perpendicular to the field and connect the ends of the coil to the input terminals of the integrator, the input voltage is

$$V_i = -NA\frac{dB}{dt} \qquad (5)$$

and the output voltage is

$$V_o = \frac{NA}{RC}\int_{B_1}^{B_2} \frac{dB}{dt}\,dt = \frac{NA}{RC}(B_2 - B_1) \qquad (6)$$

If the initial field, B_1, is zero, the output voltage is directly proportional to the strength of the magnetic field and the circuit comprises a calibrated magnetometer.

$$B = \frac{RC}{NA}V_o \qquad (7)$$

A non-inverting amplifier must be added to the circuit of Figure 1 in order to make a practical magnetometer (Figure 2). The non-inverting amplifier is needed to boost the voltage induced in the sense coil. The offset null is needed to compensate for current due to the finite input resistance of an operational amplifier. This current is integrated along with the current from the transducer and cannot be ignored. The offset null in the preamplifier (Figure 2) can be adjusted so that there is no drift in the output when there is no signal present at the input.

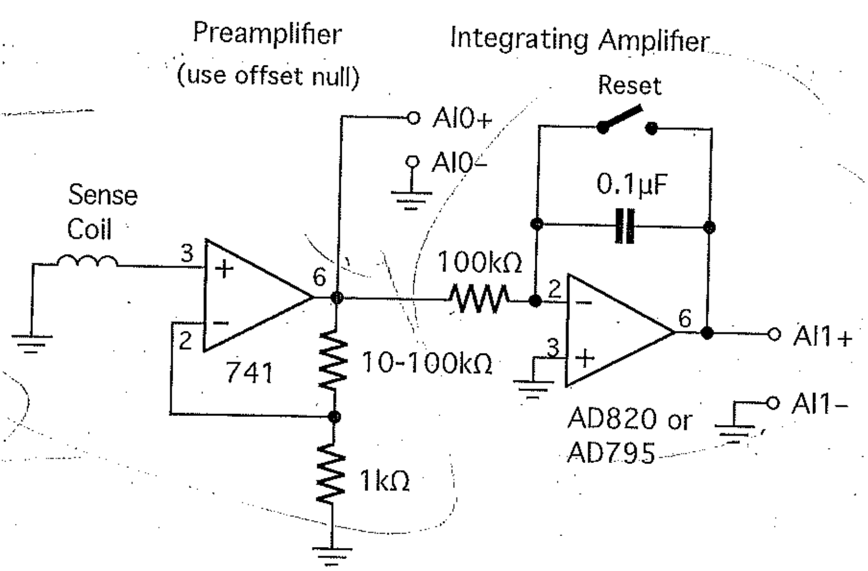


Figure 2: The practical magnetometer circuit. A 741 chip is used for the preamp circuit and an AD820 or AD795 FET-input precision op-amp is used for the integrator.

The reset switch shown in Figure 2 is a momentary contact switch connected so that it may be used to temporarily short the capacitor and force the output to zero regardless of the input. Since most applications require the instrument to begin integrating from zero, this switch may be used to set the output to zero in preparation for a measurement.

Procedure

The circuit shown in Figure 2 should be constructed on the breadboard using values of R and C selected to yield convenient readings with the available magnetic fields. The AD820 and AD795 have the same pin assignments as the 741. Do not forget to connect the power supply and the voltage offset null, neither of which is shown in the diagram. It is sufficient to perform the offset-null correction on the preamplifier circuit only. The sense coil consists of 250 turns of copper wire wound around a cyclindrical tube that serves as a guide for the magnet rod (Figure 3).

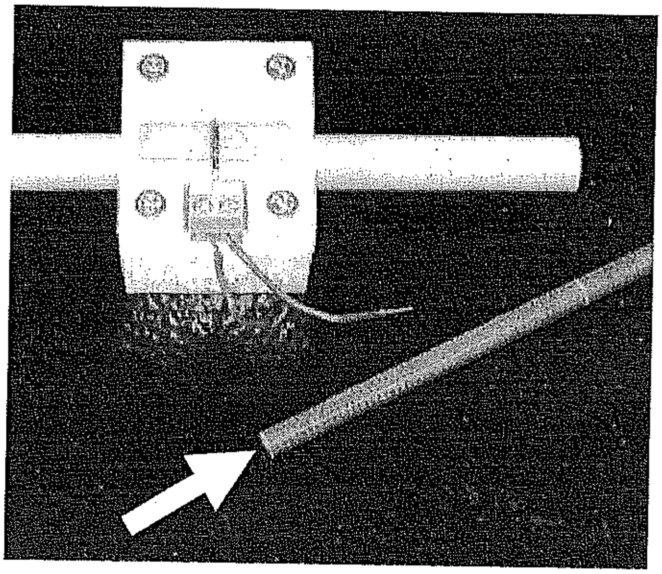


Figure 3: The sense coil (N= 250 turns) with rod and magnet (arrow). The two wires connect the coil to the magnetometer circuit.

First verify that the preamplifer works properly. Connect the multimeter to the output of the 741 (pin 6) and push the magnet through the sense coil. You should observe an output of order 100 mV. It might be helpful to set the multimeter to Min/Max. in order to caputure the extreme values.

Next connect the multimeter to the integrator output. With the reset switch open (i.e. not pushed down), adjust the offset null potentiometer so that the output drifts as little as possible; the reset switch may be used periodically during this adjustment process to prevent the output meter from drifting off scale. This adjustement is a bit awkward with a digital meter and can be done a lot easier with an analog meter. Therefore build a simple VI with a bipolar analog meter (±10 V) to use for offset-nulling the integrator drift. With this adjustment completed, the instrument is ready to use.

Electrical Connections to the Data Acquisition (DAQ) system and VI

The usual ground connections are in order. We will be using 2 analog input chanels, thus AI0- (terminal 16), AI1- (terminal 18), and AI GND (terminal 28) are connected to the ground bus on the breadboard. Then connect AI0+ and AI1+ to pin 6 of 741 and pin 6 of AD795 or AD820, respectively.

Put together a small LabVIEW VI that displays the output of the preamplifier and the integrator simultaneously as two separate XY-graphs. Setup the DAQ to sample the signals at a rate of 100-1000 Hz. Make sure the VI runs sufficiently long to capture the entire thrust motion of the magnet through the coil. You can setup the DAQ to collect a given number of N samples at a given sampling rate so that it runs for 1 or 2 seconds, long enough to allow you to push the magnet through the loop. Be sure you record the time between data points for each run. Once everything works experiment with different thrust velocities. Before you leave be sure to use a Gauss meter to measure the magnetic field of the magnet on the rod.

Your report should include:

1. The preamplifier signal and the integrator signal in the same graph. Be sure to label the axes correctly, including units.

2. A discussion of the electronics and the physics behind the shape of the preamplifier signal.

3. Convince yourself and the reader of your report that the integrator output is, indeed, the integral of the preamp output signal. You could accomplish that by integrating the preamplifier signal numerically in your spreadsheet and comparing it the the integrator output.

4. Do the max. and min. values of the preamplifier signal and the peak value of the integrator signal depend on the thrust velocity? Explain!

5. Using Eq. 7, estimate the peak field of the magnet that I thrust through the loop.

6. Discuss whether your estimate above errs on high or low side, of if there is little to no error.

References

1. The integrating amplifier is discussed in most electronics texts. Examples include:

Electronics with Digital and Analog Integrated Circuits, R. J. Higgins, Prentice-Hall (1983)

Analog Electronics for Scientific Applications, Dennis Barnaal, Waveland Press, Inc., Prospect Heights, IL 60070 (1989)

The Art of Electronics, Paul Horowitz and Winfield Hill, Cambridge Univ. Press, 2nd ed. (1989)

Introductory Electronics for Scientists and Engineers, Robert E. Simpson, Allyn and Bacon, 2nd ed. (1987)

LARGE GAIN AMPLIFIER AND LOCK-IN DETECTION

Objectives

1. To measure a resistance of the order of 0.001 Ω.

2. To gain experience with a high-quality programmable instrumentation amplifier.

3. To learn about synchronous demodulation, or "Lock-In" amplification.

Background

There are many experimental situations of interest where the voltage level of desired signals is of the order of 1 μV. Amplifying these signals to the order of 1 V where they can be used with a data acquisition system, such as LabVIEW, is plagued by background noise. It is relatively easy to amplify microvolt signals using amplifiers like you have constructed before, but the output is masked by noise. Fortunately there is a way out of this using high quality instrumentation amplifiers, filter circuits, and a lock-in amplifier. A block diagram of the scheme for the exercise at hand is shown in Figure 1.

Figure 1: Block diagram of the system.

Part 1: Programmable Instrumentation Amplifier

Three laboratory sessions will be used to complete the project of measuring a resistance of the order of 0.001 Ω. In the first lab session you will (i) construct a circuit to produce an AC signal with a peak voltage of the order of 1 mV and (ii) assemble an instrumentation amplifier having a gain of 1000 using the AD620 programmable instrumentation amplifier. You already met AD620 when you performed the strain gauge experiment (see Fig. 5 of that experiment for a block diagram of AD620). Figure 3 shows the circuit for the first part.

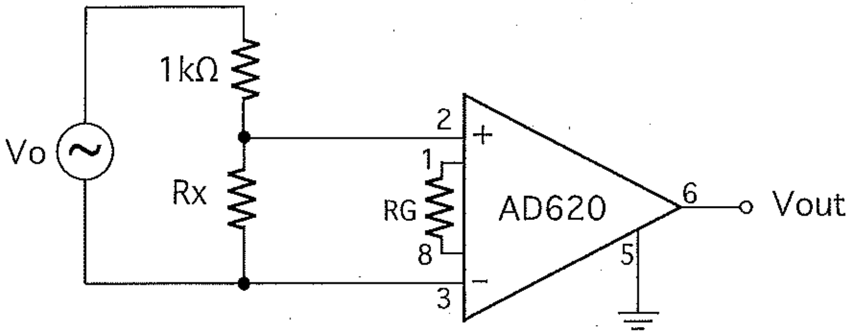


Figure 2: $R_x = 1\Omega$ and the $1k\Omega$ resistor form a 1:1000 voltage divider. The instrumentation amplifier is wired for gain = 1000 (select R_G appropriately)

Procedure Part 1

Construct the voltage divider consisting of the 1 kΩ resistor and $R_x = 1\ \Omega$ at the top left side of your breadboard (near the function generator). Be careful with your space allocation. By the end of this project you will have a total of 6 chips and various resistors and capacitors on your breadboard. Before assembling the voltage divider measure the resistance of the 1 kΩ resistor as accurately as you can. Once the divider is assembled and the ac signal generator is attached, measure the voltage across the 1 kΩ resistor using LabVIEW (your lab instructor will show you how to do this). Insert the AD620 chip close to the voltage divider, wire it for a gain of 1000, and connect the voltage across R_x to the input of the amplifier. Notice that this is a differential input; no part of the input circuit is connected to ground. Examine the voltage across the R_x and the output of the amplifier using an oscilloscope. Note carefully the noise levels and make sketches of the voltage patterns observed on the oscilloscope. Measure the peak voltages as carefully as you can. You should also measure these voltages using LabVIEW as detailed in the next section.

LabVIEW VI

Using the DAQ Assistant, create an express VI to measure two voltages at the same time, using analog input channels AI0 and AI1. Because your signal has a frequency of around 100 Hz and it is necessary to measure about 100 cycles to obtain a good amplitude measurement, it should measure for a duration of 1 second. Because we need about 100 data points per cycle for adequate resolution of a sine wave, the express VI should be set to measure 10,000 (10k) points. If the express VI is configured to measure at a rate of 10,000 (10k) points per second, then it will have the correct total measuement duration of 1 s.

It will be necessary to split the data signals into two separate channels for separate measurements of the amplitudes. Use the Split Signals VI (Express -> Sig Manip -> Split Signals VI). Right-click and drag the center of the Split Signals VI to increase its size vertically until it shows the signal splitting into two.

Next place a Tone Measurement VI on the block diagram. It is found by selecting the Signal Processing palette, then the Wfm Measure subpallette, where the Tone Measurement VI is located.

The Tone Measurement VI opens up a configuration window, choose the default setting, measuring amplitude. Hold the control key while you drag the Tone Measurement VI to create another copy. Wire your diagram as shown in Figure 3.

Connect the signals to the data acquisition system in the usual way. Measure the signal amplitude and the amplitude of the sine wave excitation from the function generator. Repeat for several values of the function generator output amplitude.

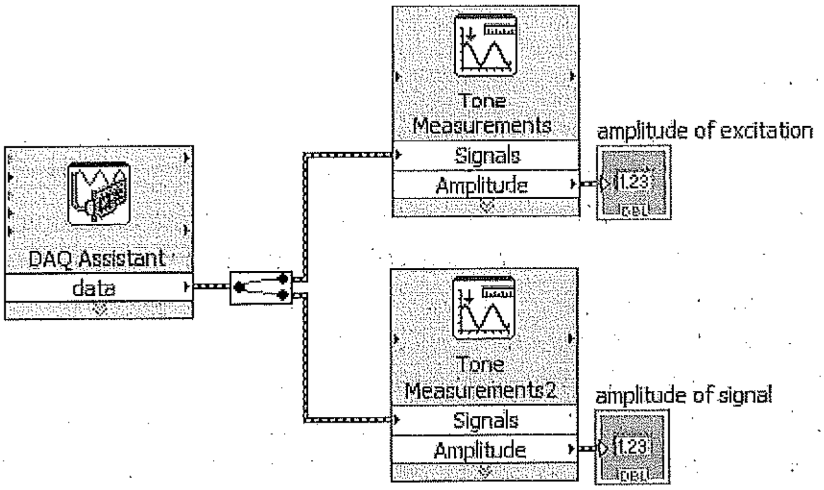


Figure 3: Important components of the LabVIEW VI for accurate measurement of the AC signals and the value of R_x.

Measuring the Exact Value of R_x

Make a graph showing the amplitude of amplified signal, V_{out}, on the y-axis and the amplitude of the original excitation, V_o, from the function generator on the x-axis. If R_1 is the value of large resistor (1 kΩ) then

$$R_x = \frac{slope}{1000} R_1$$

with *slope* being the slope of the V_{out} *vs.* V_o graph.

Your Part-1 Report should include

1. An explanation of why the current is essentially the same in both the R_1 and R_x.

2. A reasonably accurate determination of the resistance of the 1 Ω resistor. Make sure you include the uncertainty of your result.

3. A determination of the signal-to-noise ratio of the AC source.

Part 2: 2-Stage Instrumentation Amplifier

In Part 1 of this exercise you constructed and tested circuits for the first two parts of the total system, i.e. the AC signal and the first instrumentation amplifier. In Part 2 of the exercise you will build and test the next two components, i.e. the filter circuit and the second instrumentation amplifier. The circuit for Part 2 is shown in Figure 4.

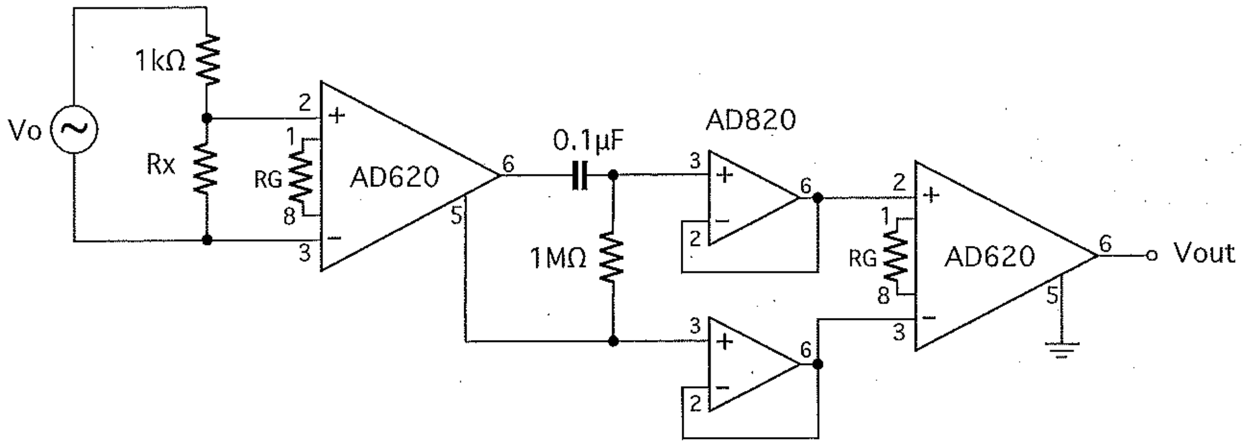


Figure 4: Circuit for AC signal, first instrumentation amplifier, high-pass filter, and second instrumentation amplifier. The second amplifier is wired for gain = 100, resulting in a total gain of 100,000. Note that the first amplifier stage and the filter are "ground-free", but the second amplifier output is referenced to ground.

Procedure Part 2

IMPORTANT: Leave the 1-Ω resistor from Part 1 in place on your breadboard. Only at the last part of today's laboratory exercise will we replace it with the smaller 0.001-Ω resistor.

ALSO IMPORTANT: Pay close attention to the recommended order of steps below. Following the recommended procedure below will allow you to check your circuit at each stage of development.

First wire up the resistor and capacitor combination to the output of the first AD524. These form a high pass filter that will block any DC signal from going to the next amplification stage. Note that pin 6 from the first amplifier is no longer connected to ground as it was for Part 1. Wire up the two AD820 op amps as shown. (Alternatively an AD795 chip can be used in place of the AD820).

Before you go on to connect the second amplifier, stop and check your circuit. The voltage between the outputs of the two AD820 amplifiers should look just like the final output of Part 1 of this exercise. Use an oscilloscope or LabVIEW to verify this.

Next, wire up the second AD620 but at first omit the gain resistor R_G, so that the AD620 has a gain of one. Consequently, the output of the second AD620 should look like the previous laboratory. Use an oscilloscope to verify the output of the second AD620.

Replace the 1-Ω resistor used in Part 1 with a 0.001-Ω resistor provided by your laboratory instructor. Wire the second instrumentation amplifier for a gain of 100 placing the appropriate gain resistor between pins 1 & 8. Examine the voltage across the 0.001-Ω resistor and the output of the amplifier using an oscilloscope. Note carefully the noise levels and make sketches of the voltage patterns on the oscilloscope. Measure the peak voltages as carefully as you can using the oscilloscope. An important part of your report will be your oscilloscope measurements of the signal amplitude and noise.

You also need to measure these voltages using LabVIEW. Use the same VI and and measurement procedure that was employed in Part 1 of this laboratory exercise to make accurate measurements of the excitation voltage amplitude from the function generator and the amplified voltage amplitude from the 0.001-Ω resistor. Plot the amplified signal amplitude on the y-axis as a function of the excitation amplitude on the x-axis. From the slope of the resulting line, find the resistance of R_x (the '0.001-Ω' resistor):

$$R_x = \frac{slope}{100{,}000} R_1$$

where R_1 is the measured value of the large resistor.

Your Part-2 Report should include

1. A determination of the resistance of the 0.001 Ω resistor. Make sure you include the uncertainty of your result.

2. A serious and detailed explanation of the purpose of the resistor and capacitor in this circuit.

3. A serious and detailed explanation of the purpose of the two AD820 operational amplifiers.

4. Your graph of the amplitude of the amplified voltage from the small resistor as a function of the amplitude of the excitation signal from the function generator.

Part 3: Synchronous Demodulation, or "Lock-In" Amplification

In this part you will use an AD630 chip configured as a synchronous demodulator, more widely known as "Lock-In" amplification. Lock-in amplification works by limiting the bandwidth of the signal, thereby minimizing the amplified noise. In order for the scheme to work the frequency and phase of the signal must be known. Therefore a lock-in amplifier has a signal input and a reference input. The reference signal must have the same frequency and phase as the small signal to be recovered from the noise. Figure 6 shows a block diagram of the AD630 configured as a lock-in amplifier. The reference signal has the same frequency and phase as the signal and is used to operate the "demodulation switch" A-B. The action of this switch is to reverse the sign of the input signal half-way through the period of the reference signal.

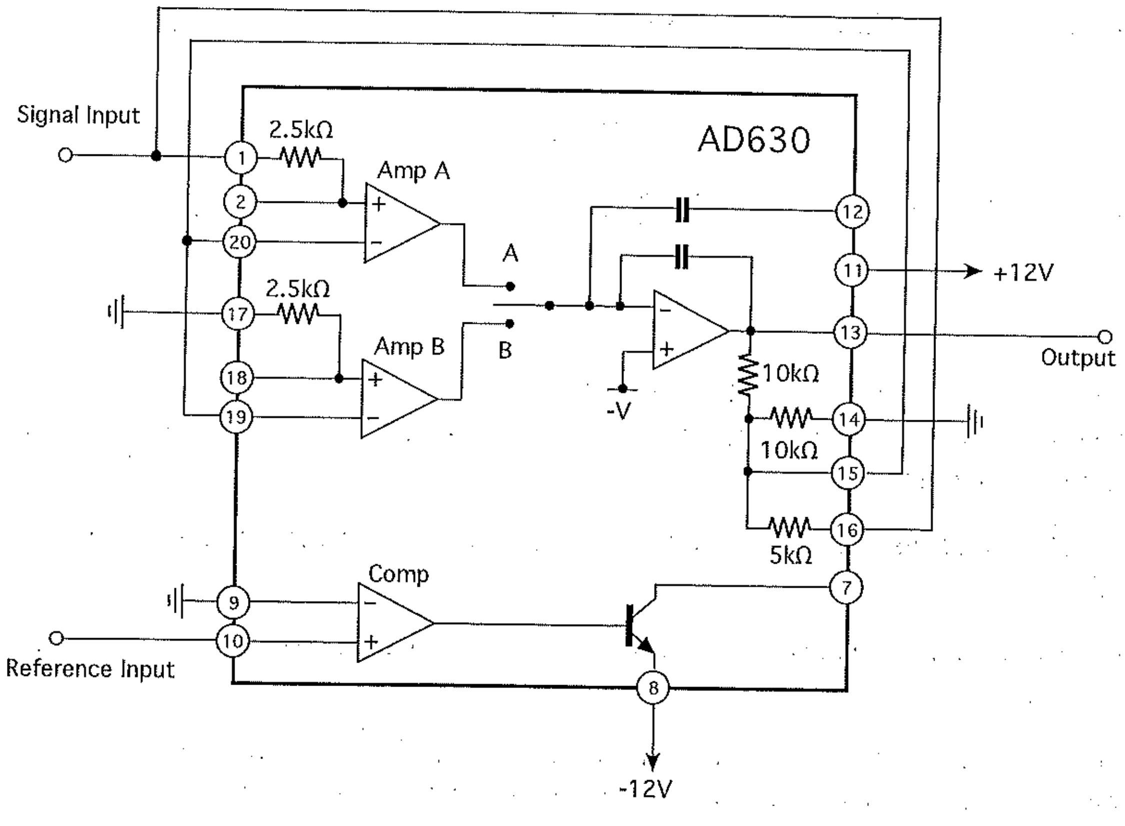


Figure 5: Block diagram of the AD630 wired as a lock-in amplifier.

Procedure Part 3

First wire up the AD630 separately from your Part 2 amplification circuit. Put a signal of approximately 1 V amplitude into both the input of the AD630 and also into the REFERENCE voltage input. Figure 6 shows the circuit of the AD630 wired up as a lock-in amplifier with both signal input (pins 1 and 16) and the reference input (pin 10) connected to the function generator. Please be aware that in this configuration the lock-in amplifier has an internal gain of 2. This will be important later when we will calculate the overall gain of the system.

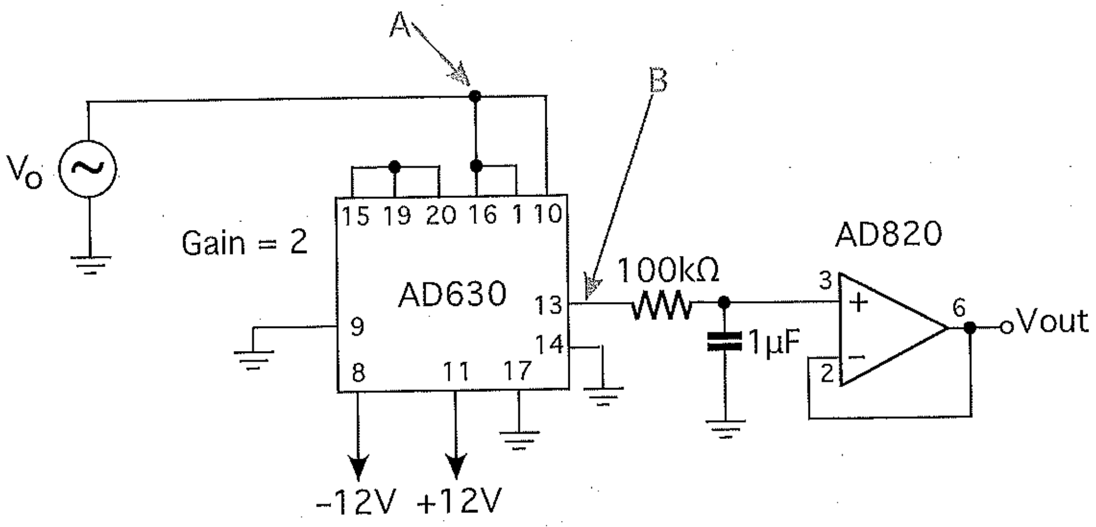


Figure 6: Lock-in amplifier with the AD630. Both signal and reference inputs are conntected to the function generator. Adjust the amplitude of V_o to 1 V.

Inspect the waveforms at point *A* (i.e., the signal at the lock-in input, pin 1/16) and point *B* (the output of the lock-in, pin 13) using an oscilloscope or LabVIEW. At *A* you should see an ordinary sine waveform, and at *B* you should see a rectified sine wave. Make a drawing of the waveforms or capture the screen. At the output of this circuit (pin 6 of the AD820) you should see a DC-voltage (with a little ripple).

Put together a VI to measure the amplitude of your 1-V input and the DC-output signal using the *Tone Measurement* Express VI. The DC-output signal V_{out} is related to the input voltage amplitude V_o as $V_{out} = 2\frac{2}{\pi}V_o$. The first "2" is due to the internal gain = 2 of the AD630, while the $\frac{2}{\pi}V_o$ corresponds to the average of the rectified input signal V_o. Verify this by recording V_{out} for several values of V_o and plotting V_{out} vs. V_o and fitting a straight line to the points. Is the slope of your graph equal to $\frac{4}{\pi}$?

Now use the AD630 to accurately measure the amplified voltage from the 1-mΩ resistor. To accomplish this you will connect the lock-in's signal imput to the output of the 2-stage instrumentation amplifier, and the reference input to the function generator. The complete circuit is shown in Figure 7. Measure the DC voltage V_{out} as a function of the input excitation amplitude V_o and determine the slope. The relation between V_{out} and V_o is

$$V_{out} = \frac{4V_o}{\pi}\frac{R_x}{1k\Omega}Gain$$

with *Gain* = 100,000. You should replace $1k\Omega$ with the measured value of that resistor. Then use the slope to determine the resistance of the R_x as accurately as possible.

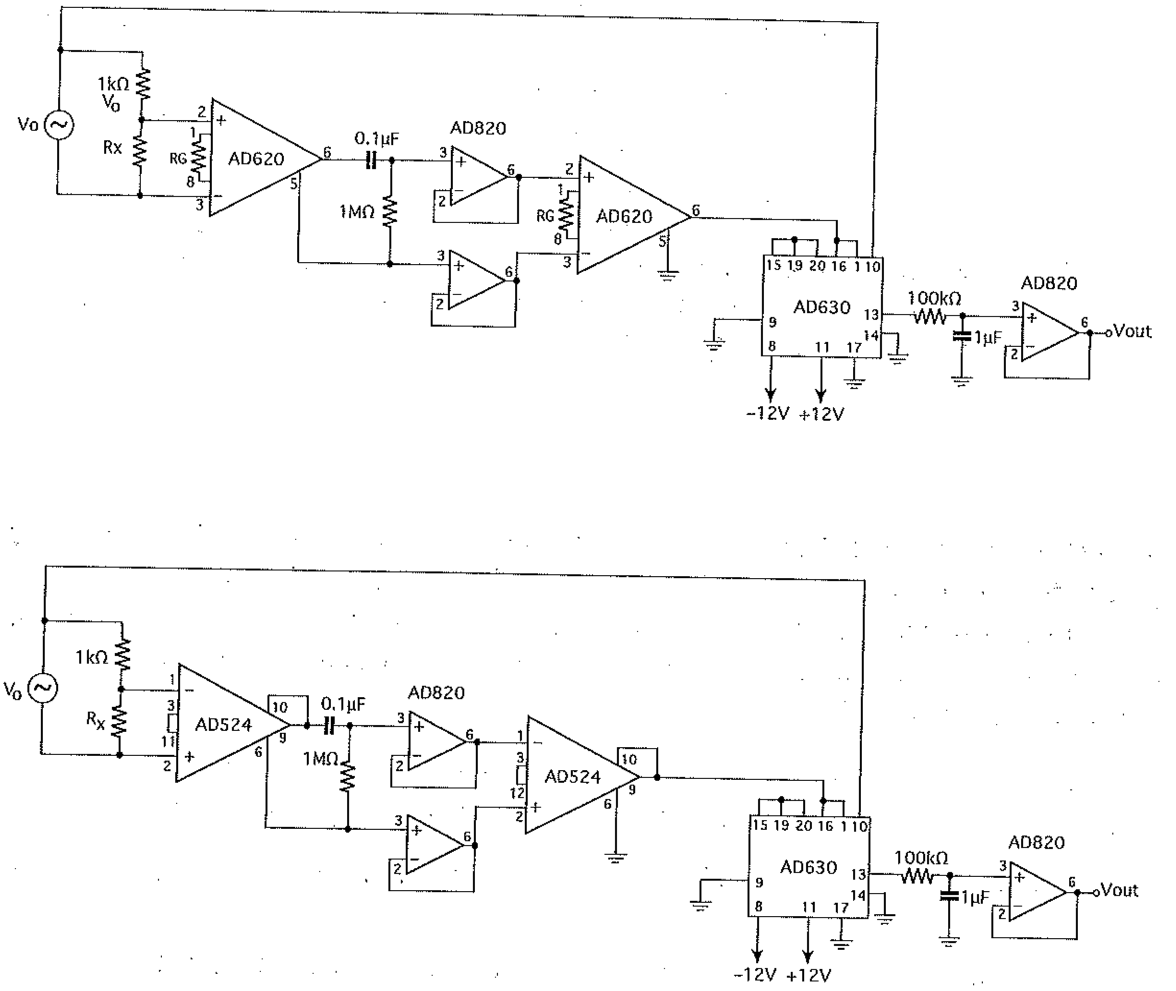
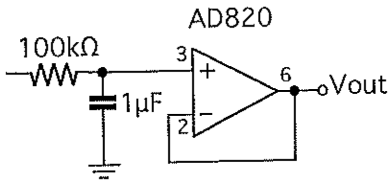


Figure 7: Complete circuit of the 2-stage instrumentation amplifier connected to the lock-in amplifier using the AD620 instrumentation amplifier (top) and the AD524 instrumentation amplifier (bottom).

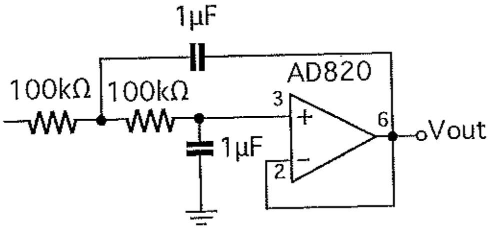
If there is time

There are a couple of things to try it time allows.

1) Replace the first order low-pass filter



by a second order filter



2) Use the excitation signal from the function generator to trigger the analog-to-digital conversion. That way you are doing synchronous conversion, and you eliminate the last remaining bit of ripple from the DC voltage.

Your Part-3 Report should include

1. A determination of the resistance of the 0.001 Ω resistor. Make sure you include the uncertainty of your result.

2. Your graph of the amplitude of the amplified voltage from the small resistor as a function of the amplitude of the excitation signal from the function generator.

3. A discussion how the lock-in technique improves the result as compared to the 2-stage instrumentation amplifier approach from Part 2.

PHYSICAL PENDULUM

Objectives

Part 1: To record the angular displacement of a damped physical pendulum as a function of time.

Part 2: Establish the validity of the small-angle approximation from an analysis of the oscillation period as a function of the amplitude.Analyze the energy loss of the pendulum and deduce the underlying damping mechanism.

Background

In this exercise you will record the oscillation of a damped physical pendulum and study the damping mechanism. Figure 1 shows the geometry of the pendulum. A 1/8" diameter aluminum rod with a 2" diameter and ½" thick aluminum disk as the bob makes up the pendulum. The rod is mounted on the shaft of a variable resistor, and mount the resistor so that the swinging pendulum rotates the shaft.

A potentiometer arrangement (Figure 2) produces a voltage that can be calibrated to exhibit the angular position of the pendulum. Sampling the potentiometer output voltage at regular intervals of time yields relative measurements of angular displacement and time. The internal clock of the computer is used to record the time when each voltage is sampled. These measurements are saved for analysis.

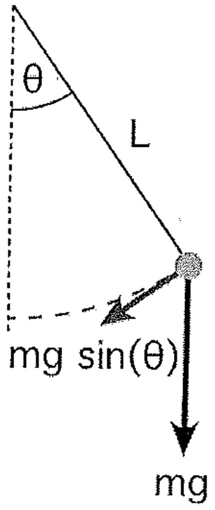


Figure 1: The geometry of the pendulum. The tangential component of the pendulum bob's weight, $mg\sin(\theta)$, acts as the restoring torque.

The tangential component of the weight acts as the restoring torque that tends to restore the oscillating pendulum to its equilibrium position. The frictional torques cause the pendulum to lose energy and the amplitude of the oscillations decreases with time.

If we take the gravitational force as acting at the center of gravity and assume that the frictional torque is proportional to the angular velocity of the pendulum, the differential equation describing the motion takes the form

$$\frac{d^2\theta}{dt^2}+\frac{2}{\tau}\frac{d\theta}{dt}+\omega_o^2\sin\theta=0 \tag{1}$$

with τ is the damping constant and $\omega_o=\sqrt{\frac{Mgx}{I}}=\frac{2\pi}{T}$ where T is the oscillation period, I is the moment of inertia of the pendulum about the axis of rotation, M is the mass of the pendulum, g is the acceleration due to gravity, and x is the distance from the center of gravity to the axis of rotation.

If we limit oscillations to small angles, i.e. $\theta < 10°$, then $\sin\theta\approx\theta$ and the equation of motion (Eq. 1) becomes

$$\frac{d^2\theta}{dt^2}+\frac{2}{\tau}\frac{d\theta}{dt}+\omega_o^2\theta=0 \tag{2}$$

and can readily be solved. The angular position as function of time is given by

$$\theta=\theta_o e^{-t/\tau}\cos(\omega t) \tag{3}$$

with $\omega=\omega_o\sqrt{1-\frac{1}{\omega_o^2\tau^2}}$. Note that $\omega\approx\omega_o$ for low damping, i.e. $\tau\to\infty$ or $\omega_o\tau\gg 1$.

Because M, I, and g are constant the period T is independent of the initial angle θ_o in the small-angle approximation $\sin\theta\approx\theta$. Take a little time to substitute Eq. 3 into the equation of motion, Eq. 2, to demonstrate that $\theta=\theta_o e^{-t/\tau}\cos(\omega t)$ is a solution.

If the small-angle approximation is no longer justified, the equation of motion (Eq. 1) can be solved by numerical techniques but there is no solution that can be found similar to Eq. 3 when the approximation $\sin\theta\approx\theta$ is valid. A detailed analysis of Eq. 1 shows that the period can be determined from

$$\frac{T}{T_o}=\frac{2}{\pi}F(k) \tag{4}$$

where T_O is the period for small angular displacements, $k=\sin\left(\frac{\theta_o}{2}\right)$, i.e. for an angle of release for which the small-angle approximation holds, and $F(k)$ is a function of k defined as

$$F(k)=\int_0^1\frac{dz}{\sqrt{(1-z^2)(1-k^2z^2)}} \qquad \text{with } z=\frac{1}{k}\sin\left(\frac{\theta}{2}\right) \tag{5}$$

Table 1: Calculated values of the ratio T/T₀ from Eq. 4. The values were computed by numerical integration of Eq. 5 using Simpson's rule.

Angle (Degrees)	Calculated Ratio T/T_0
0	1.000
15	1.004
30	1.017
45	1.040
60	1.073
75	1.119
90	1.181
105	1.262
120	1.373
135	1.528
150	1.762
165	2.185

A theoretical value for the period ratio for a series of release angles is shown in Table 1. In the course of this experiment you will determine the period of oscillation as a function of the release angle. Table 1 serves as a reference to compare your experimental results.

Damping Mechanism

Our pendulum shows considerabe damping and typically stops oscillating after 2–3 min. The usual description of damping assumes a retarding force proportional to the velocity of the pendulum, as in Eqs. 1 and 2. This force gives rise to a torque causing an exponential decay of the amplitude as described by the decay time | in Eq. 3. However, other mechanisms can give rise to a decease of the oscillation amplitude, and as part of this experiment we will study the nature of the damping mechanism present in our pendulum.

In particular we will consider 3 types of frictional forces of the following form.

$$\begin{aligned} \vec{F}_o &= f\ \hat{v} \\ \vec{F}_1 &= \gamma_1 v\ \hat{v} \\ \vec{F}_2 &= \gamma_2 v^2\ \hat{v} \end{aligned} \qquad (6)$$

where $\hat{v}$ is a unit vector pointing in the direction *opposite* of the velocity of motion. $\vec{F}_o$ is a velocity-independent frictional force such as might be present at the shaft of the potentiometer. $\vec{F}_1$ is the often-present velocity-proportional air drag, and $\vec{F}_2$ is a force proportional to the square of the velocity; that typically shows up as air drag of object moving at high velocity. In our measurements we will examine the loss of energy per cycle of oscillation. Thus we need to express the energy loss integrated over a oscillation period. The instantaneous loss of energy is the work done by the frictional force over a distance dx, $W = F\,dx$. The rate of instantaneous energy loss is then $\frac{dW}{dt} = \vec{F}\cdot\frac{d\vec{x}}{dt} = \vec{F}\cdot\vec{v}$. Integrating over one cycle yields the energy loss per cycle $\Delta E = \int_t^{t+T} \vec{F}\cdot\vec{v}\,dt'$. For a pendulum dropped from an initial angle A^1 at time $t = 0$ we have $x(t) = A\cos(\omega t)$ and thus $v(t) = -\omega A\sin(\omega t)$, with $\omega = \frac{2\pi}{T}$. We must think of these expressions as describing the motion of one particular cycle with A, 7and T representing average values of amplitude, frequency, and period, respectively, over that cycle.

For velocity-independent friction, $\vec{F}_o = f\,\hat{v}$, and calling the beginning of the cycle over which we integrate $t = 0$, this results in

$$|\Delta E| = f_o\omega A\int_0^T |\sin(\omega t)|dt = 2f_o\omega A\int_0^{T/2} \sin(\omega t)dt = 4Af_o \qquad (7)$$

The average energy of the pendulum during this cycle is

$$E = \frac{1}{2}m\omega^2 A^2, \qquad (8)$$

therefore $A = const.\sqrt{E}$ and Eq. 7 may be written as $|\Delta E| = const.\sqrt{E}$, and thus $\frac{|\Delta E|}{\sqrt{E}} = const.$

Similarly one can show that for frictional forces of form $\vec{F}_1 = \gamma_1 v\,\hat{v}$ and $\vec{F}_1 = \gamma_2 v^2\,\hat{v}$ the energy loss can be written as $\frac{|\Delta E|}{\sqrt{E}} = const.\cdot\sqrt{E}$ and $\frac{|\Delta E|}{\sqrt{E}} = const.\cdot E$, respectively. Figure 2 summarizes these findings.

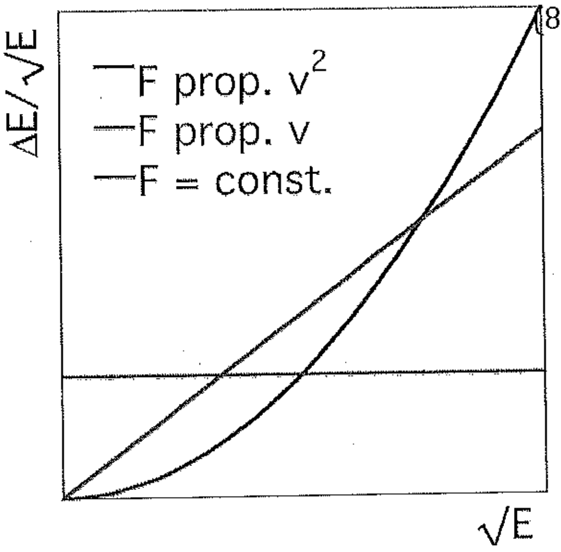


Figure 2: Energy loss curves for various frictional mechanisms.

---

[1] This treatment as presented here is limited to the small-angle approximation.

The energy loss curve $\frac{|\Delta E|}{\sqrt{E}}$ *vs.* $\sqrt{E}$ is thus indicative of the mechanism responsible for damping of the pendulum. We now need to relate the energy loss $\frac{|\Delta E|}{\sqrt{E}} = f(\sqrt{E})$ to an observable that is easily measured. From Eq. 8 we see that $E \propto A^2$, therefore we can write $\frac{|\Delta E|}{\sqrt{E}} = \frac{|\Delta A^2|}{A}$. Since $|\Delta A^2| = 2A|\Delta A|$ we have $\frac{|\Delta A^2|}{A} = \frac{2A|\Delta A|}{A} = 2|\Delta A|$. Therefore plotting $|\Delta A|$ *vs.* A is equivalent to $\frac{|\Delta E|}{\sqrt{E}}$ *vs.* $\sqrt{E}$. Thus plotting $|\Delta A|$ *vs.* A will reveal the damping mechanism.

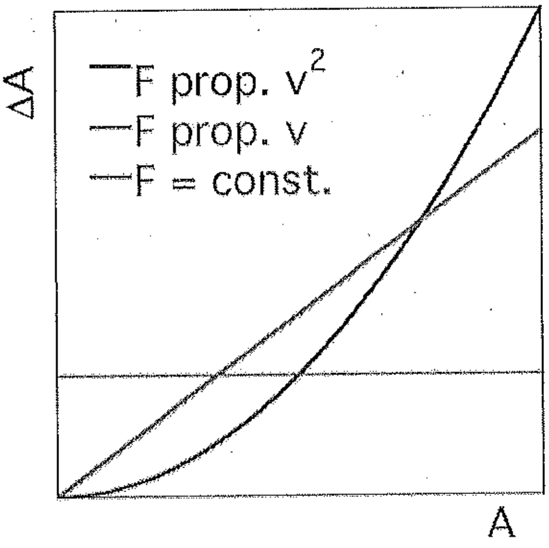


Figure 3: Plotting $|\Delta A|$ *vs.* A is equivalent to the energy loss curves $\frac{|\Delta E|}{\sqrt{E}}$ *vs.* $\sqrt{E}$

Experimental Setup

The transducer that tracks the pendulum is a 1-turn potentiometer with linear taper and without a mechanial stop, i.e. its shafts rotates continuously. The potentiometer is connected to +5 V and ground as a voltage divider (Figure 4). The potentiometer is adjusted so the voltage divider output produces about 2.5 V when the pendulum is in its equilibrium position. When the pendulum is moved from the equilibrium position, the output of the variable voltage divider will show a voltage greater or less than 2.5 V, depending on which side the pendulm is moved. There is a 1-k∧ series resistor at the output of the voltage divider to limit the current if the pot wiper is on the "upper end" of the scale.

The pendulum transducer is connected to channel 0 of the analog input of the DAQ. A LabVIEW vi will sample the voltage and convert it to an angular reading after a simple 3-point calibration. The vi will be set up to sample the voltage until the pendulum stops oscillating. The sampled voltage will be saved to a LabVIEW measurement file that allows convenient analysis of the pendulum's osciallation waveform at a later time.

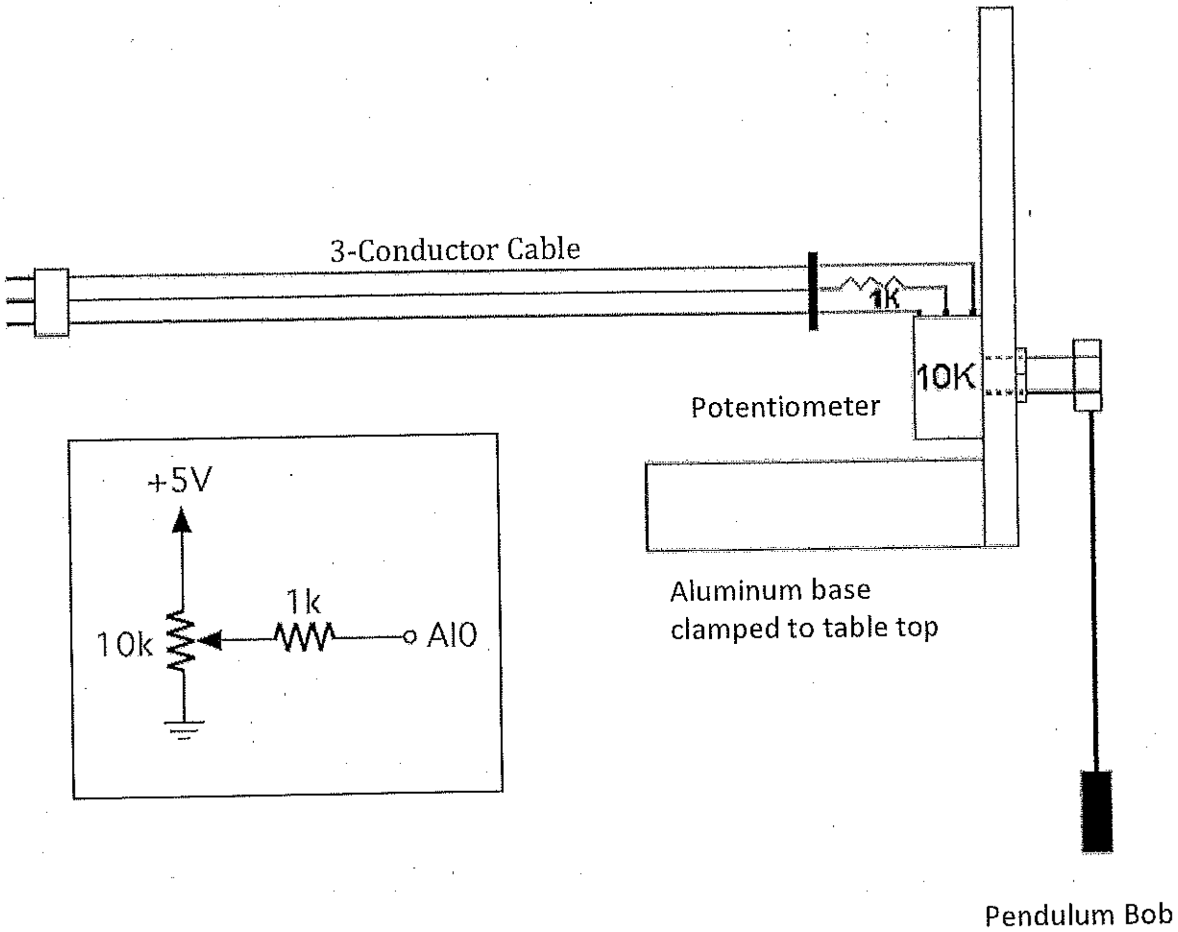


Figure 4: The pendulum and its mount. The inset shows how the 10k-pot is connected. The 1-k resistor limits the current when the pot wiper is on the "upper end" of the scale.

Procedure Part 1

Connect the pendulum potentiometer to the analog input channel 0 and +5V and ground as shown in Fig. 4. Use the Fluke multimeter to measure the potentiometer voltage and adjust the pot shaft so that the voltage is near 2.5 V. We want to record the oscillations of the pendulum as a function of time. For convenience the voltage from the pendulum pot must be converted to angular readings. For this we need to move the pendulum to known angles and record the voltage reading for that angle. Since the pot is very linear it is sufficient to perform 2 readings, however, you may want to use more than two readings to confirm the linearity of the pot readings. Calibration may be done manually by recording voltages and angles in a spreadsheet, then performing a linear regression to obtain the parameters *m* and *b* for the calibration equation $Angle = mVoltage + b$. Please note that the calibration is good as long as the pendulum rod remains attached to the pot; however, once the pendulum has been removed from the pot's shaft, the calibration must be repeated. For the convenience of easily repeating a calibration, a LabVIEW vi similar to the one shown in Figure 5 may be used. This vi requires the pendulum to be moved to angular positions -90°, 0°, and +90°, where it records the voltage and then performs a linear regression resulting in the slope *m* and intercept *b* of the calibration equation.

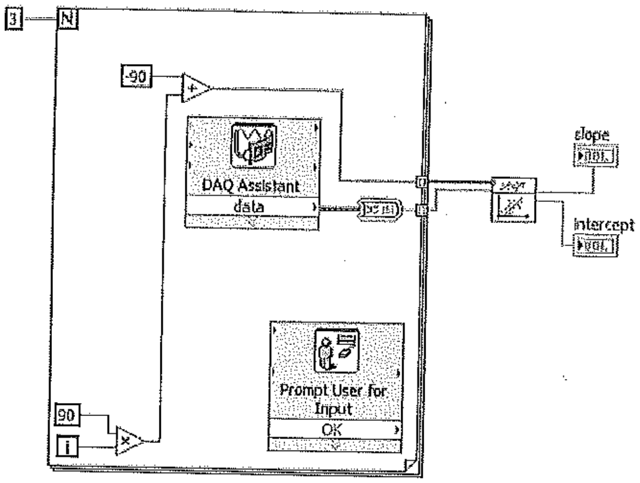
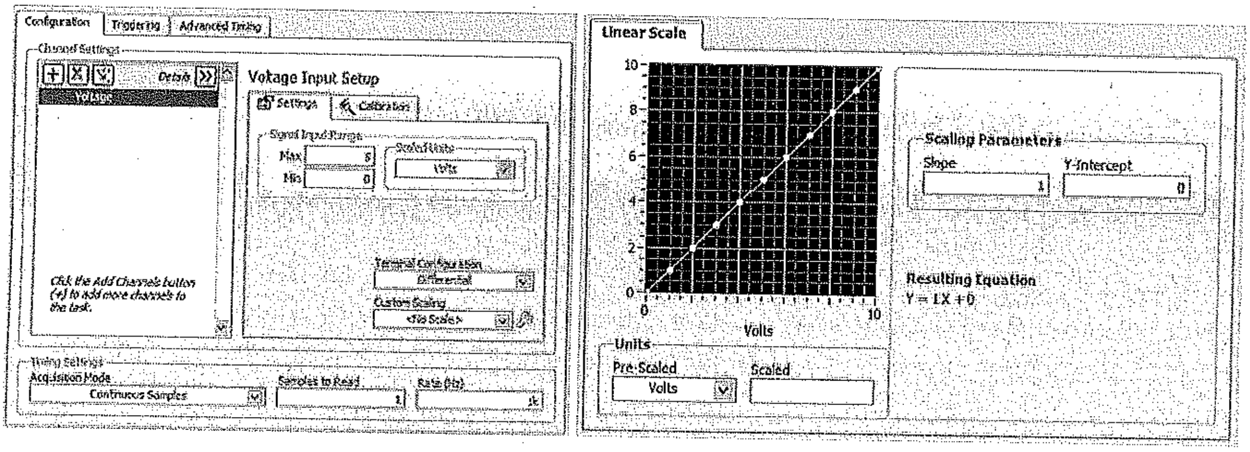


Figure 5: A LabVIEW calibration vi

In principle we could measure the voltage corresponding to the pendulum position, then run it through the calibration equation to obtain the angular position. A more convenient way to perform the voltage-to-angle conversion can be done during the configuration of the DAQ assistant. In the configuration page (Figure 6, left) click on the *Custom Scaling* drop down window and select *Create New...* followed by *Linear*. On the following page enter a name, e.g. "Voltage-to-Degrees", then click *Finish*. In the window that opens next enter *slope* and *intercept* then click OK (Figure 6, right) . Now the data acquisition system is calibrated to return angular positions of the pendulum directly.

Next assemble the vi that records the pendulum position data. Set up analog input channel 0 to read a given number of samples at a specified rate. Make it so that the sampling rate and the time period over which data will be recorded can be entered on the



Max 5

Min 0

Scaled Units Volts

Terminal Configuration Differential

Custom Scaling <No Scale>

Acquisition Mode Continuous Samples

Samples to Read 1

Rate (Hz) 1k

Slope 1

Y-Intercept 0

Resulting Equation
Y = 1X + 0

Pre-Scaled Volts

Scaled

Figure 6: The configuration panel of the DAQ Assistant allows custom scaling of the raw input voltage (left). If linear scale is selected, the slope and intercept parameter must be entered to complete the calibration (right).

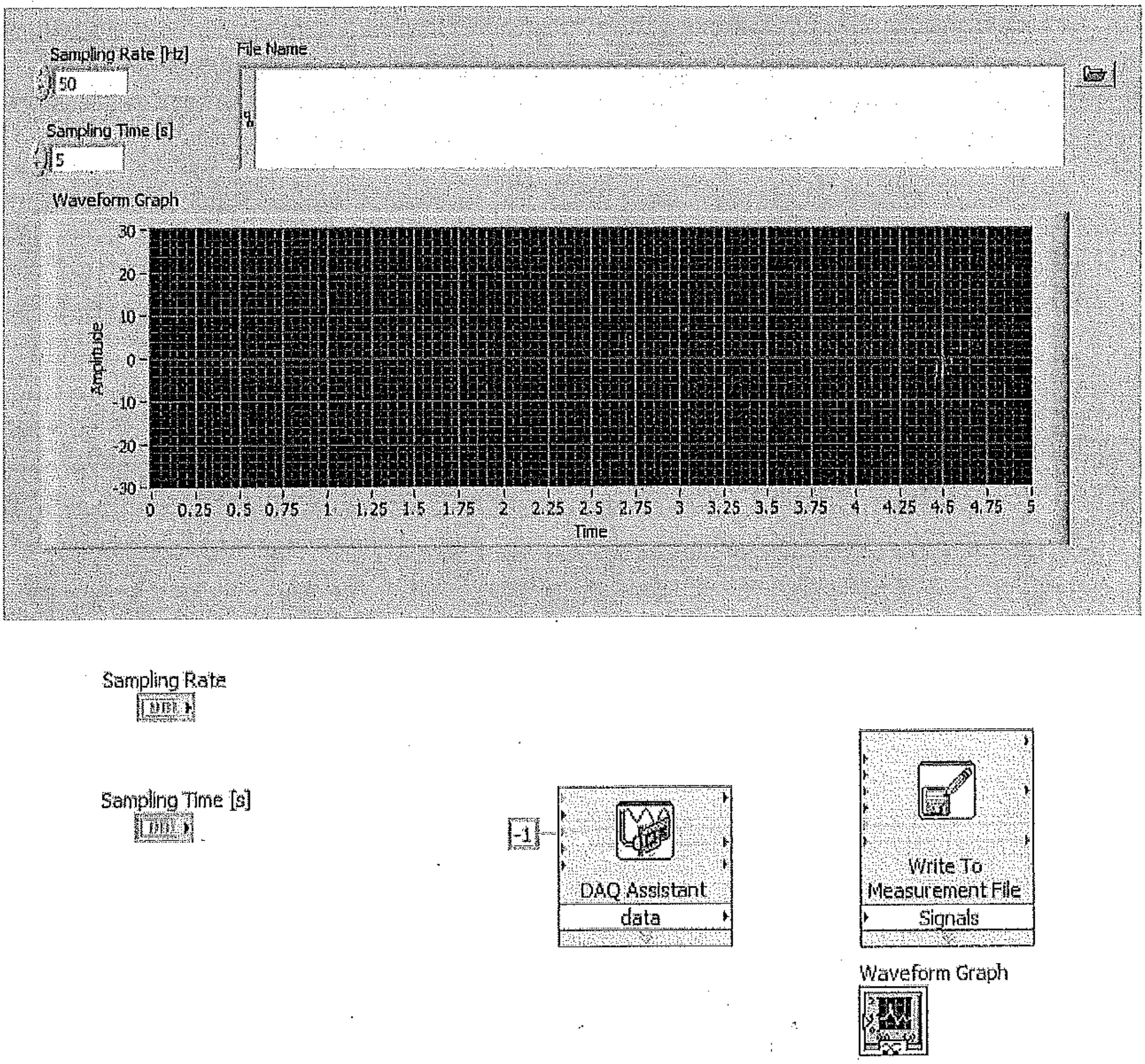


Figure 7: Elements of the pendulum data acquisition system. The "−1" parameter forces the DAQ assistant to wait until the data accumulation is complete.

front panel. At the end of the data accumulation period write the result to a *Measurement File* (use the *Write to Measurement File* express vi). This file format preserves all timing information and makes it possible to read back the file for later analysis. All elements needed to accomplish this are shown in Figure 7. Just connect the components to complete the vi.

With the completed vi record the oscillations of your pendulum for a variety of initial angles until the pendulum stops completely (typically 2–3 min.). Experiment with sampling rates from 10 to 100 Hz and save all files, keeping careful notes so that you will be able to select the "right" file later.

Procedure Part 2

Next we want to find how the oscillation period changes as a function of the amplitude and compare that with expectations from Table 1. To do this we read the recorded oscillation signal from the measurements file (*Read From Measurement File* from the *Express -> Input* palette) and extract a small portion of the entire waveform (*Extract Portion of Signal* from the *Express -> Sig Manip* palette). Using the *Tone Measurements* express vi from the *Express -> Signal Analysis* palette you can extract the amplitude and the frequency, and from that the period, of the small section of the signal. Repeating this analysis for another section of the signal, you can step through the entire oscillation record and obtain a plot of the period vs. amplitude of the entire signal. At the end save the data in a spreadsheet file. Figure 8 shows an implementation of this scheme.

Import the Period vs. Amplitude data file into a spreadsheet construct a column of $|\Delta A|$ values by subtraction two adjacent amplitude values and plot $|\Delta A|$ $vs.$ A. Compare your plot with that of Figure 3 to establish which damping mechanism prevails for which range of amplitudes.

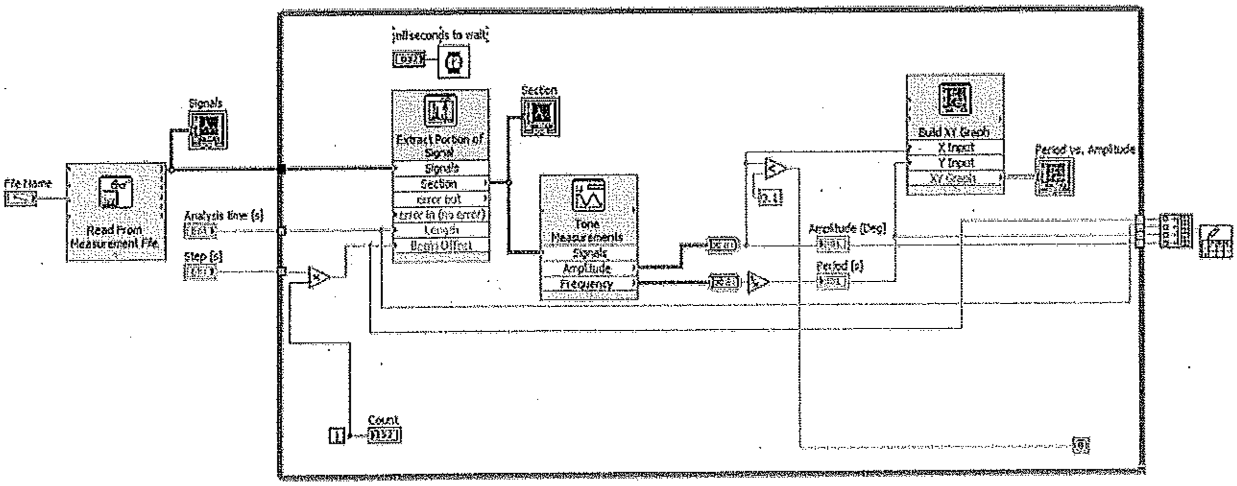


Figure 8: vi for the analysis of the period as a function of the oscillation amplitude

Report Guidelines

The "Physical Pendulum" experiment is our midterm experiment, and the report for this follows a different format than the ones you did previously. The report for the midterm experiment will be done in the format of a journal paper similar to how you would prepare a paper that you submit to a journal for publication.

Here is some of the legwork that goes into this. Begin by looking at a few papers published in Physical Review, Physical Review Letters, or other physics journal. You can do that from any computer on campus via "Web of Science" or "INSPEC", both of which you can access via the MU Library website. Let your instuctor know if you need help with this.

In most cases papers have the following sections:

Introduction – Experimental Details – Results – Discussion –Summary – References

In some cases (longer papers) the sections are marked by section headers, in others (shorter papers, rapid communications, letters) the section headers are omitted, but the letter still has sections that correspond to this type of structure.

The content of the various sections follows this pattern.

Introduction:

What is known about this topic and the context of your work. What is new that you are doing that others have not done yet. Typically you would have these 3 paragraphs:

1st paragraph:	Wide view of topic
2nd paragraph:	Zoom in a bit closer to problem at hand
3rd paragraph:	Say what you are doing in this paper

Experimental setup:

Details as to how you did your measurements or experiment. Put in sufficient detail that someone else could reproduce your experiment. You want to have a block diagram, photograph, circuit diagram, or some other visual of the setup, or if nothing else, a reference to another publication that describes your setup in detail. This is often done for complicated experimental techniques that have been published earlier and are used like that over and over, e.g. Mössbauer spectroscopy – most setups are alike, no need to describe it, just say "we used a standard Mössbauer spectrometer as described by [X]". Our experiment is simple enough that you will not want to do that.

Results:

List your measurements and present your results in a suitable manner without judging or discussion. "Suitable" may be a graph or a table, something that makes is easy to see what you have. Table is only useful if you have a few numbers. A series of spreadsheets would not be "suitable". In our case a graph would be suitable (θ vs. t, T vs. amplitude, ΔAmplitude vs. Amplitude, and such).

Discussion:

This is where you "think aloud" what it all means. Weigh the pro and cons, draw conclusions, indicate further experiments that could be done, etc. This is where the meat of the papers is.

Summary:

That is the section for those who don't have the time to read the entire paper. Typically people read the abstract and the summary first, then look at the figures and decide if they want to read the rest of the paper. Thus you will want to reiterate what was done and the most important results in the summary.

References:

List of papers and books that you mention in the body of the paper.

Here some tips how to proceed.

You will not find much on pendulums in Physical Review Letters. Best place to look for things you can write in your introduction is the *American Journal of Physics.*

Limit yourself to 5 figures, then prepare those figures first, write a caption, and arrange the figures so they "tell your story". Then use your words to guide the reader through your work.

Write in passive voice, because it does not matter who did what, but what was done!

Follow the Style Manual handed out separately.

References

1. The damped physical pendulum is analyzed in most calculus-based university physics texts. See, for example,

University Physics, Hugh D. Young, Addison-Wesley, 8th ed. (1992) or

Physics for Scientists and Engineers, Raymond A. Serway, Saunders, 3rd ed. (1990).

2. The development of the equations presented here for the period of a physical pendulum for arbitrary angular displacements can be found in

Classical Dynamics, Jerry B. Marion and Stephen T. Thornton, Harcourt Brace Jovanovich (1988).

3. Simpson's rule is discussed in most elementary calculus books. See, for example,

Calculus and Analytic Geometry, George B. Thomas and Ross L. Finney, Addison-Wesley (1988).

INTRODUCTION TO MICROCONTROLLERS: MEET ARDUINO

Objectives

1. Introduce the Arduino microcontroller.

2. Get familiar with simple optical and acoustic signal outputs.

Arduino is a single board microcontroller based on the ATMEL ATmega328P chip. The chip contains the processor, 32kbytes of flash memory to store programs and variables, as well as a small amount of EEPROM and RAM memory. So in effect, it is a micro computer on a chip; this is what is commonly refered to as a microcontroller. The Arduino board contains additional circuitry that allows connection of the ATmega328-chip to a computer via USB interface. This enables programming and exchange of data of the Arduino with a computer. The Arduino is "open-source", i.e. everything in and about the Arduino is in the public domain and accessible for all. A good source of all there is to know about Arduino and a great place to start is www.arduino.cc. A step-by-step tutorial to Arduino you'll find at www.ladyada.net/learn/arduino/index.html.

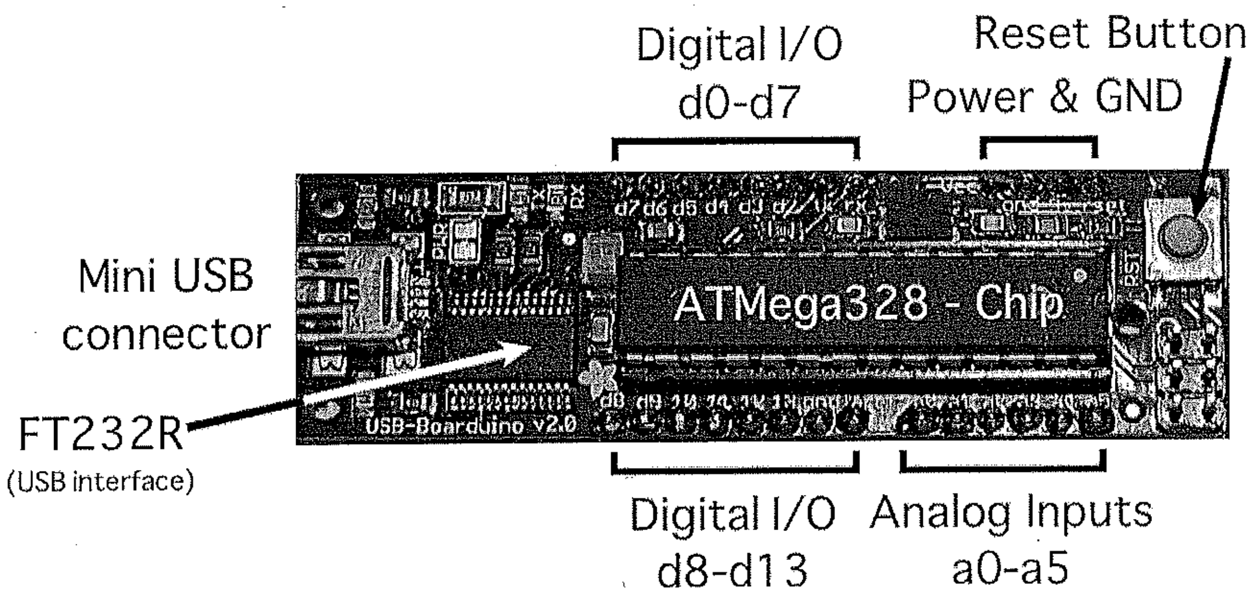


Figure 1: The Arduino of the *Boarduino* variety. The ATMega328 chip contains the processor and flash memory for storing program code and data. The smaller FT232R chip handles the connection to the host computer via USB. The pins for the analog and digital I/O lines are spaced such that the Arduino can be plugged into a solderless breadboard.

Programming the Arduino is done most conveniently via an integrated development environment (IDE). This is software that runs on your Windows, Macintosh, or Linux computer and lets you develop so-called sketches (this is what programs for the Arduino are called). The language used to program the Arduino is not unlike C or Python and is not very hard to learn. Once a sketch is written, the IDE compiles the sketch into machine code and uploads it to the microcontroller chip on the Arduino board.

The Arduino perceives its environment via sensors and transducers that can be connected to a number of analog and digital input and output ports. For example, a phototransistor can be used to measure light intensity, and a sketch that runs on the Arduino can then turn on a light should the light intensity fall below a defined value. The Arduino is also able to communicate with the host computer via USB and can send back numerical data that it computes as the sketch executes. The Arduino can also act as a stand-alone device without being connected to a host computer once the program is loaded. Should the Arduino lose power, it resets itself once power is restored and begins execution of the loaded sketch from the beginning. This is possible because the flash memory does not "forget" when power is disconnected.

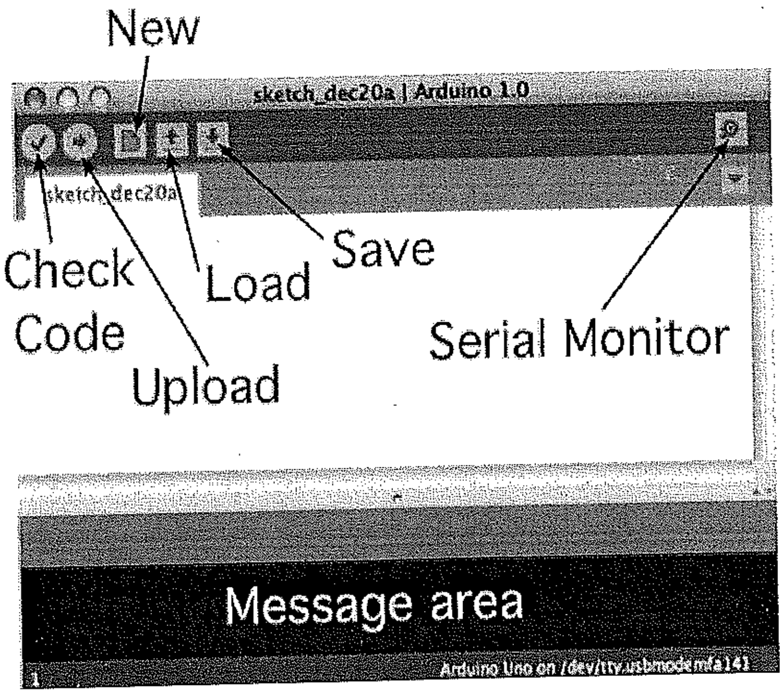
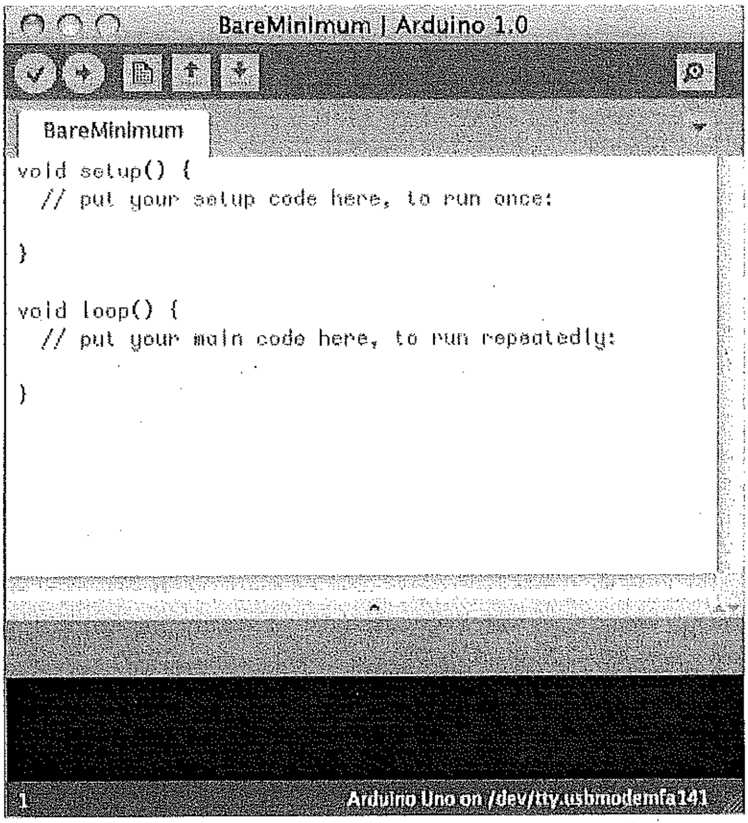


Figure 2: An empty sketch window opens when you first start the Arduino IDE. The buttons on the top are for managing and uploading your sketch to the Arduino.

Arduinos comes in different sizes and flavors. The particular model we are using in PHY 294 is known as the *Boarduino* (Figure 1). It has all the features of the regular Arduino but has a formfactor that allows it to be used with a breadboard of the type we are using in this lab. For simplicity we will continue to refer to the *Boarduino* as Arduino. Let's tinker a bit. Start the IDE (click on the *Arduino* icon on your desktop), and you will see an empty sketch window (Fig. 2). There are a few buttons on the top of the window, and a message area along the bottom. The second button from the left is the "upload" button; it lets you send your sketch to the memory on the ATMega328-chip. Other buttons are for starting a new sketch, or loading and saving sketches. The first button on the left is the "check code" or "compile" button. It will compile a sketch and tell you if there are any errors in the message area at the bottom of the window.

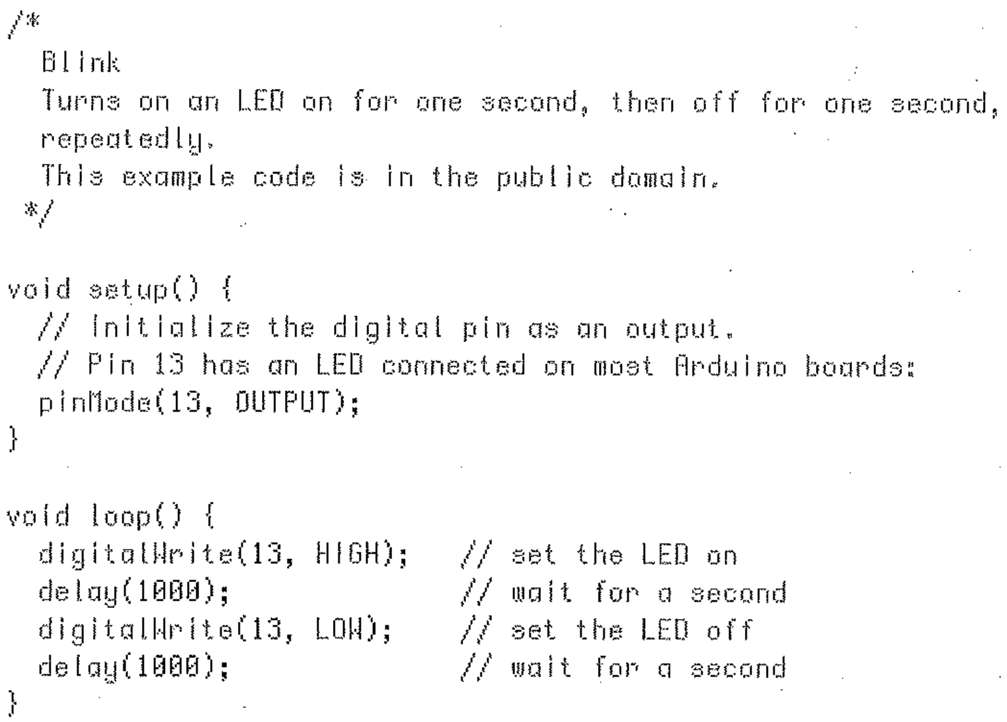
One valuable feature with the Arduino IDE is that it comes pre-loaded with a number of examples that will help you learn how to program the microcontroller. Click on File/Examples/1.Basics and select *BareMinimum* from the list. This shows the structure of an Arduino sketch (Fig. 3). There are two parts: (i) the setup() function and (ii) the loop() function. The setup() function is executed once and will contain all the definitions and initializations that need to happen so the sketch can run. Once the code of the setup() function has been executed, the processor begins to run the code of the loop() function. As the name implies, this code is repeated over and over, until the Arduino is turned off.



```
void setup() {
  // put your setup code here, to run once:

}

void loop() {
  // put your main code here, to run repeatedly:

}
```

Figure 3: The bare minimum sketch does not do anything, but it shows the sturcute of an Arduino sketch.

Let's look at a code that actually does something noticeable. Click on File/Examples/1.Basics and select *Blink* from the list. This is one of most basic Arduino codes you will find. It makes an LED blink once a second. The code appearing in the IDE window should look something like this:



```
/*
  Blink
  Turns on an LED on for one second, then off for one second,
  repeatedly.
  This example code is in the public domain.
 */

void setup() {
  // initialize the digital pin as an output.
  // Pin 13 has an LED connected on most Arduino boards:
  pinMode(13, OUTPUT);
}

void loop() {
  digitalWrite(13, HIGH);   // set the LED on
  delay(1000);              // wait for a second
  digitalWrite(13, LOW);    // set the LED off
  delay(1000);              // wait for a second
}
```

Anything in between /* and */ is ignored by the compiler; also, anything following // until the end of the line is ignored as well. So we can use these to annotate the sketch to help us understand what the code does. The setup() function has one line of code: `pinMode(13, OUTPUT);` here we define pin 13 as an digital output pin. It is digital, because the voltage on this pin will be either 5V or 0. 5V is called `HIGH` and 0 is called `LOW`. This is the only setup coded required, so now we go on to the loop() function. The first line, `digitalWrite(13, HIGH);` sets the voltage of pin 13 to +5V. Note that at the end of the command there is always a ";", that is how we signal the IDE the end of the command. The next command is `delay(1000);` and tells Arduino to do nothing for 1000 ms. The next two commands is setting pin 13 to 0 and waits for 1 s. Since we are at the end of the loop() function, the sketch continues to excecute at the beginning. The effect is that as long as the Arduino is powered, it keeps blinking the LED connected to pin 13. Notice that we did not have to worry actually to connect an LED to pin 13 as there is an LED on the Arduino board which is already connected to pin 13. If you wanted to connect a second LED to, say, pin 10, then you would have to actually do this as there is no such element on the board.

Once you got your Arduino blinking, spend some time playing with this code. Try to make it blink faster or slower. Maybe just a short blink and a long pause, or vice versa. How about connecting a second LED to a different pin and then have both LEDs blink in some way. Come up with your own variation, be creative!

If you are not satisfied by just "ordinary" blinking, let's get a bit fancier. From the basic examples, select the *Fade* sketch and examine it. Predict what will happen when the sketch runs, then run it. This is an example of a digital pin used as an analog output pin. Not all of the 14 digital outputs can be used this way, only pins 3, 5, 6, 9, 10, and 11. The analog output is achieved by switching the pin rapidly between 0 and 5V while varying the on-off time ration, a.k.a. the duty cycle. This results in an rms-value between 0 and 5V, effectively turning the digital output pin in a analog output.

Blinking a light is one way Arduino can get your attention. Even though it is a simple action, it can be used to convey quite a bit of information. Think about Morse code. Another way to convey information is via sound. The Arduino commands `tone()` and `noTone()` can be used to produce a square-wave voltage with a defined frequency at a specified output pin. If a small speaker or a piezo crystal is connected to the pin, you can hear a tone. The command syntax

```
tone(pin, freq)
```

sounds a tone of frequency `freq` until the command `noTone()` is encountered. Alternatively, the command

```
tone(pin, freq, dur)
```

sounds a tone of frequency `freq` for duration of `dur` milliseconds. Connect the provided piezo buzzer to pin 5, designate this pin as an OUTPUT and write an Arduino sketch that plays the "Big Ben" chime.

Your report should include:

1. A description of at least three variations of the *Blink*-sketch. Include listings of your sketches.

2. Briefly summarize the *Fade* sketch, explaining how the "fading"-effect is achieved.

3. Your annotated sketch that produces the Big Ben chime. Describe how you went about coming up witht the notes and name your sources.

CURRENT-TO-VOLTAGE CONVERTER

(BE-Diode of an NPN Transistor)

Objectives

1. To investigate the electrical properties of the BE-diode of an NPN transistor.

2. To design, construct, and test a current-to-voltage converter using an operational amplifier.

Background

A semiconductor diode consists of a sharp junction between a p-doped and n-doped piece of semiconductor. The pn-junciton acts as a current valve that lets current pass only in one direction. The current-voltage characterisitic of a semiconductor diode is given by the Schockley equation

$$I = I_R\left(e^{\frac{eV}{nkT}} - 1\right) \tag{1}$$

with V the potential difference between the anode (p-side) and cathode (n-side) of the diode, I_R the reverese saturation current, T the absolute temperature, k the Boltzmann constant, and n an empirical factor between 1 and 2.

An npn-junction transistor consists of a p-doped semiconductor (the base) sandwiched between two n-doped layers (collector and emitter). Thus the base-emitter (BE) junction behaves just like a regular pn-junction diode. In this experiment we will record the current-voltage characteristic of the BE-diode of an npn junction transistor. Measuring current directly is not very easy as most am-meters do have a finite resistance which would distort the current that is to be measured. Therefore the approach is to turn the current into a voltage and measure the voltage instead.

Figure 1 shows the circuit of the I-to-V converter.[1] Its output voltage V_{out} is proportional to the current I flowing through the BE-diode of the transistor. As long as we treat the Op-Amp as ideal, the current I flowing through the transistor will also flow through the 1-kΩ feedback resistor. According to Kirchoff's rules, we have

$$V_{in} - V_{BE} - IR - V_{out} = 0 \tag{2}$$

Since there is no current flowing into the Op-Amp, the inverting input is at the same potential as the non-inverting input, which is at ground potential. Thus

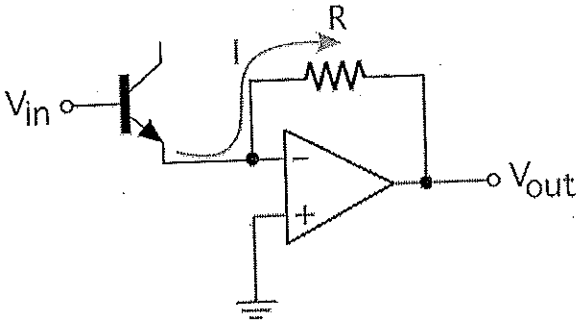


Fig. 1: The circuit of the current to voltage converter. The output voltage V_{out} is proportional to the current flowing through the BE-diode of the tansistor.

$$V_{in} - V_{BE} = 0 \tag{3}$$

Combining Eqs. 2 and 3 we have

$$V_{out} = -IR \tag{4}$$

Thus measuring V_{out} and knowing the feedback resistor R allows to find the current through the transistor

$$I = -\frac{V_{out}}{R} \tag{4}$$

Procedure

Set up the circuit shown in Fig. 2. Do not forget to connect both Op-Amps to the ±12V supply. The analog output channel AO-0 provides V_{EB} for the transistor. A voltage follower is used to avoid loading down the analog output circuit of the DAQ board. Use a feedback resistor between 1 and 10 kΩ. Be sure to use an Ohm-meter to measure the feedback resistor before you connect it in your circuit.

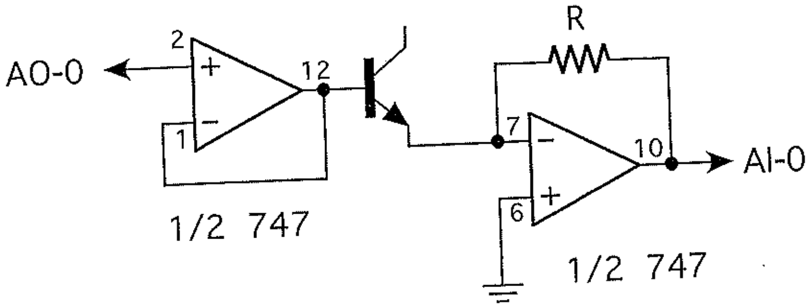


Fig. 2: I-to-V converter circuit with buffer between the transistor and the AO-0 output.

The LabVIEW VI

Begin by dragging a For loop from the *Programming -> Stuctures* functions palette to the block diagram.

Use the DAQ assistant (*Express -> Input; Signal acquisition; voltage; ao0; Finish*) to place an analog output (channel AO-0) inside the loop. On the setup screen select *Acquire 1 Sample (on demand)*, the click *OK.*

Use the DAQ assistant again, this time to place analog input (channel AI-0) inside the loop; also select *Acquire 1 Sample (on demand)* before clicking *OK.*

Drag an express XY-graph on the front panel, then move it inside the for loop on the block diagram.

Outside the loop, drag in the Write to Spreadsheet icon from the *Programming -> File I/O* functions palette.

Now your block diagram should look similar to Fig. 3. Make the appropriate connections to step the analog output channel 0 (AO-0) from 0 to 0.7 V in steps of 0.01 V. Read the analog input channel 0 (AI-0) for each of these steps, calculate the transistor base current, I_B, and plot it as a function of V_{BE} using an express XY graph. When the loop is done, save the data in a spreadsheet file for analysis at a later time.

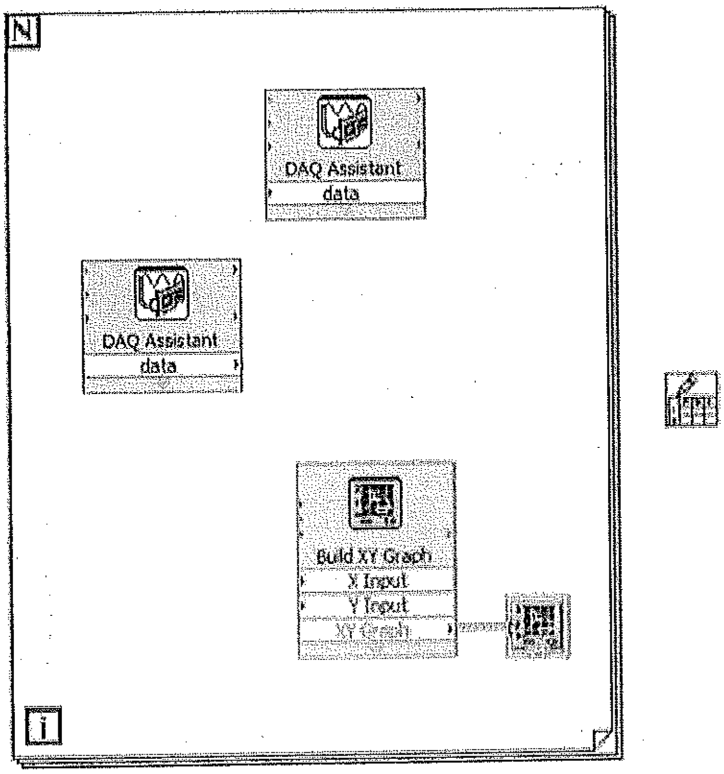


Fig. 3: Skeleton of the VI to determine the I-V characteristic of the BE-diode of an npn transistor.

Electrical Connections to the DAQ System

Connect AO GND, AI GND, and AI0– (terminals 14, 28, and 16, respectively) to the ground bus line on your breadboard. You can daisy-chain from terminal 14 to 16 to 28 to the ground line. Then connect AO-0 (terminal 12) to pin 2 of the buffer Op-Amp, and AI-0+ (terminal 15) to pin 10 of the I-to-V converter Op-Amp (Fig. 2).

Your report should include

1. A discussion of the current-to-voltage converter.

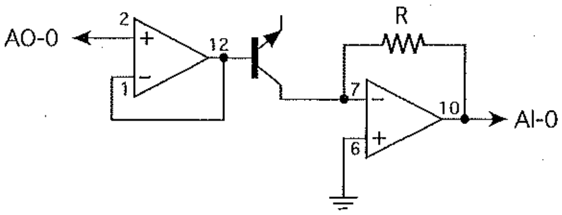
2. A graph of I_B *vs.* V_{BE}.

3. Note that for $e^{\frac{eV}{nkT}} \gg 1$ Eq. 1 simplifies to $I = I_R e^{\frac{eV}{nkT}}$. Thus plotting *ln I vs. V* will have the form of a straight line. Plot *ln* I_B *vs.* V_{BE} and identify the region for which $e^{\frac{eV}{nkT}} \gg 1$ is valid. Perform a linear regression to find the slope and from that determine the empirical constant *n*. Is your value in the range of 1...2?

4. The spreadsheet showing the measurements and linear regression analysis that goes with step 3.

If there is time

Flip the transistor as shown in the diagram and measure I_B *vs.* V_{BC}. Compare the current characteristic with the first measurement. What you observe is a result of the emitter being doped at a higher concentration than the collector. This is useful if you have an unknown transistor and must identify the terminals.



References

Texts that discuss a current-to-voltage converter include

The Art of Electronics, Paul Horowitz and Winfield Hill, Cambridge University Press, 2nd edition (1989).

Introductory Electronics for Scientists and Engineers, Robert E. Simpson, Allyn and Bacon (1987).

Electronics with Digital and Analog Integrated Circuits, Richard J. Higgins, Prentice-Hall (1983).

Analog Electronics for Scientific Applications, Dennis Barnaal, Waveland Press, Inc., Prospect Heights, IL 60070 (1989).

BIPOLAR JUNCTION TRANSISTOR CHARACTERISTICS

Objective

To measure the electrical characteristics of an NPN junction transistor.

Background

Operational amplifiers are versatile devices for manipulating electrical signals. For example, they have great utility for tailoring voltages to suit the input of a data acquisition system (DAQ). However, an operational amplifier is inadequate if the output is to control a device such as a relay or light bulb that requires more than about 50 mA. The capabilities of an operational amplifier can still be used by letting it control a transistor which then controls the relay or light bulb.

The bipolar junction transistor (BJT) is a common and useful type of transistor[1]. There are two versions referred to as NPN and PNP. NPN and PNP transistors function on the same physical principles but the polarities for the voltage connections and current directions are opposite for the two types. A junction transistor has three parts that are termed base, emitter, and collector. The symbol for the transistor and the identification of the three parts are shown in Figure 1. The PNP symbol is the same except the arrow points in the opposite direction. A current into the base region controls a current in the collector region.

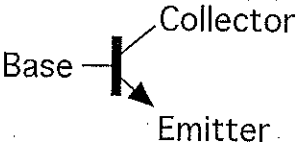


Figure 1: The symbol for an NPN junction

Accordingly, the current in a device connected in series with the collector terminal can be controlled by an appropriate current into the base terminal. It is easy to control tens of milliamperes of current in the collector by tens of microamperes of current in the base. In this experiment we want to determine the electrical characteristics of a junction transistor which can then be used in control circuits.

Three currents and three voltages are associated with the three terminals of the transistor (Figure 2).

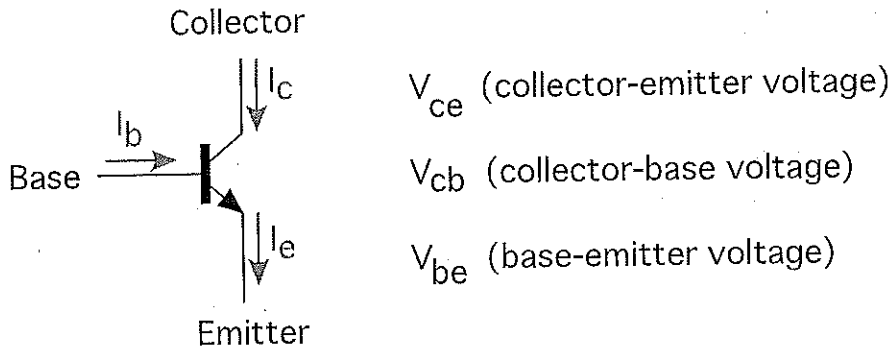


Figure 2: Currents and voltages associated with a junction transistor.

The currents and voltages are labeled

I_b	———>	current into the base
I_c	———>	current into the collector
I_e	———>	current out of the emitter
V_{be}	———>	voltage between the base and emitter
V_{cb}	———>	voltage between the collector and base
V_{ce}	———>	voltage between the collector and emitter

Performing an experiment to systematically keep track of six variables seems a formidable task. However, only two of the currents and two of the voltages are independent. If you know any two currents you can calculate the third, and if you know any two voltages you can calculate the third. It follows from Kirchhoff's current rule that the current entering the transistor must equal the current leaving. Therefore

$$I_e = I_c + I_b \tag{1}$$

It follows from Kirchhoff's voltage rule that the base-emitter voltage and the collector-base voltage must sum to the collector-emitter voltage. Therefore

$$V_{ce} = V_{be} + V_{cb} \tag{2}$$

The currents and voltage of most interest are I_c, I_b, and V_{ce}. To investigate systematically the relationship among these three variables I_b is maintained at some constant value and I_c measured as V_{ce} changes. The data are then displayed as a plot of I_c versus V_{ce} for the constant current I_b. The base current I_b is then changed and the procedure repeated. A set of plots is generated that corresponds to the chosen values of base current. This set is referred to as the characteristic curves. Characteristic curves for a junction transistor have the features shown in Figure 3.

In the flat part of two curves, we determine the change in collector current ΔI_c and the corresponding change in base current ΔI_b. The ratio $\Delta I_c/\Delta I_b$ is called the current gain and is given the symbol β. Knowing the current gain, you can determine the current change needed in the base to achieve a desired current change in the collector.

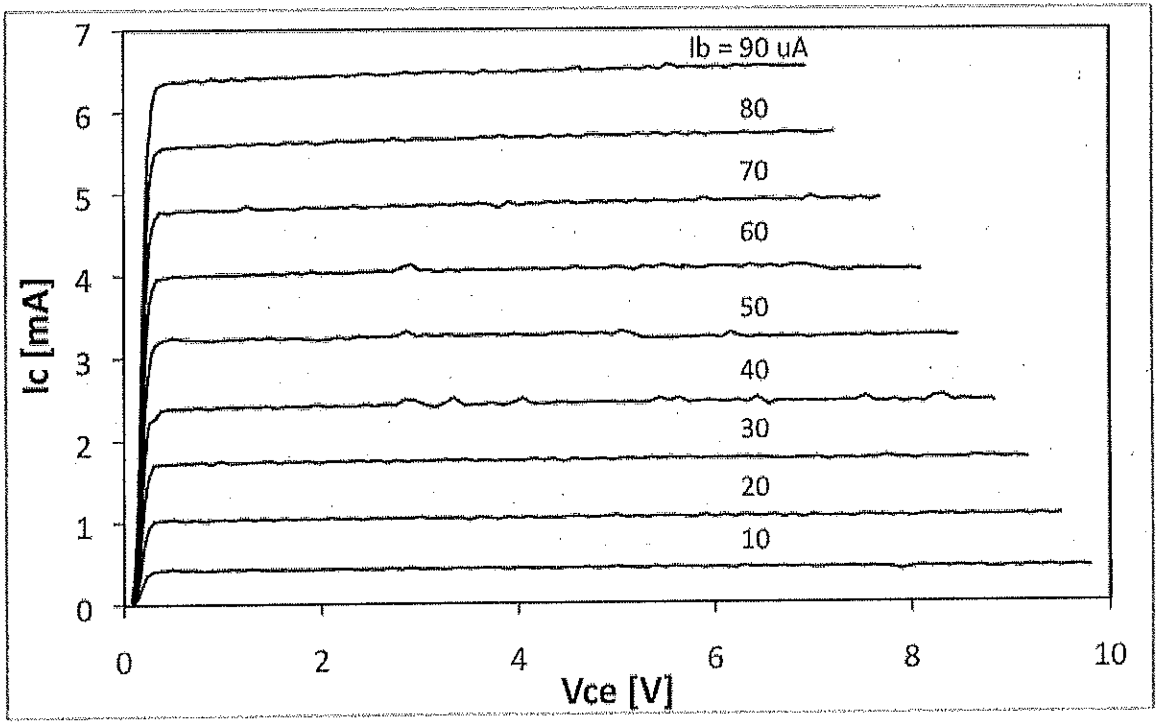


Figure 3: Typical characteristic curves for an NPN junction transistor.

The basic circuit for an NPN junction transistor is shown in Figure 4. If the type of transistor were PNP then the polarities of the power supplies would be reversed. To a reasonable approximation the base-emitter voltage is constant and is about 0.6 volt for a silicon transistor. Therefore, keeping the power supply voltage for the base (V_B) constant the current in the base will be constant. Varying the power supply for the collector (V_C) will cause the collector-emitter voltage and the collector current to change. Recording V_{ce} and I_C as the power supply voltage changes produces the data for constructing the characteristic curves.

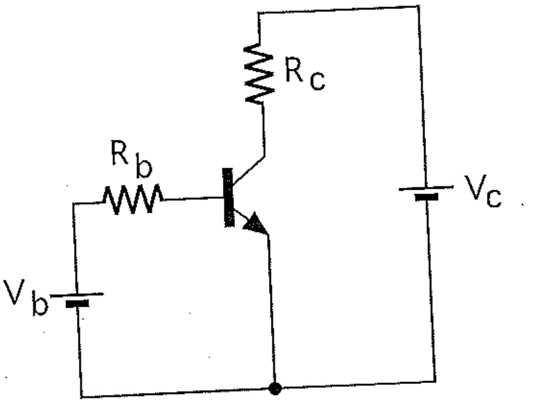


Figure 4: Appropriate electrical connections for an NPN junction transistor.

The collector-emitter voltage is measured by connecting one of the analog inputs to the collector terminal. To measure the collector current connect a second analog input to the upper end of the resistor (R_C) connected to the collector. Subtracting the voltages gives the voltage across the resistor. Dividing the voltage across the resistor by the resistance gives the collector current. Two other analog inputs are used to measure the voltage across a resistor in the base circuit. Knowing the resistance we can determine the base current.

Procedure

Three analog inputs to the DAQ will be used for this experiment. It is recommended that you start with the VI from the I-to-V converter experiment (pp. 95). Wire the transistor circuit as shown in Figure 5 and use the DAQ analog input channels AI0 through AI2. The resistor values are $R_C = 470\Omega$ and $R_b = 100\ k\Omega$. The base current is determined by the resistor $R_b = 100\ k\Omega$, the diode drop across base-emitter of the transistor, and applied voltage at AO-0. Use Kirchhoff's loop equation to determine I_b.

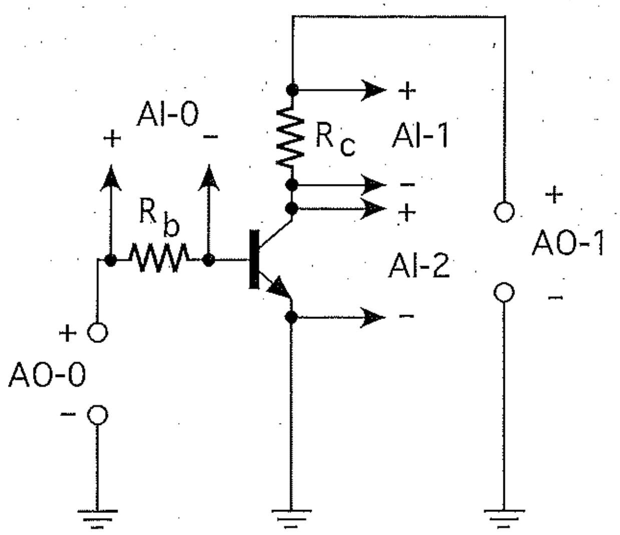


Figure 5: Circuit for measuring the characteristics of an NPN transistor.

For calculations, the base-emitter voltage may be taken as a constant 0.7 V. Start with $R_b = 100$ kΩ and $V_c = 0$, record the voltage across R_b, and begin taking measurements by executing your VI for several values of V_c in the range of 0 to 10 V. The computer will display a plot of collector current versus collector-emitter voltage. Make sure your VI will save the data of each run in a different file. Use four different values of R_b and save the data so that each file represents a different value of R_b.

Import these files into a spreadsheet and plot all the data on one graph. It should be similar to what is shown in Fig. 3. Calculate the base currents and current gain (β) for each run. Average the β-values for determining the current gain for the transistor being investigated.

Your report should include

1. A graph of the experimental data.

2. A hard copy of the spreadsheet showing the measurements and analysis.

3. A calculation of the current gain for each run and the corresponding average.

4. A printout of the program with comments about the statements.

5. A discussion of the results.

References

1. Examples of electronics texts having a discussion of junction transistors include

Electronics with Digital and Analog Integrated Circuits, Richard J. Higgins, Prentice-Hall (1983)

Analog Electronics for Scientific Applications, Dennis Barnaal, Waveland Press, Inc., Prospect Heights, IL 60070 (1989)

The Art of Electronics, Paul Horowitz and Winfield Hill, Cambridge University Press, 2nd ed. (1989)

Introductory Electronics for Scientists and Engineers, Robert E. Simpson, Allyn and Bacon, 2nd ed. (1987)

An Introduction to Modern Electronics, William L. Faissler, Wiley, (1991)