Experiment 1: Basic Electric Components/Measurements and Traditional Instruments

Tools/concepts emphasized: Digital Multimeter, Power Supply, Function Generator, Oscilloscope, Ohm's Law, Kirchhoff's Law.

1. Objectives

i) Familiarization with resistive electric components such as resistors and potentiometers.

ii) Familiarization with traditional instrumentation equipment such as the digital multimeter, function generator, and oscilloscope.

iii) Use of instruments to perform basic direct current (DC) and alternating current (AC) measurements.

2. Background

Most measurement systems contain electric circuits; hence, as engineers, it is crucial that we know how to build, design, and analyze systems containing electric components. The experimental analysis of electric circuits requires the generation and measurement of signals (*e.g.*, voltages and currents) via special purpose instruments. In the following, the various components and instruments to be used in this experiment will be briefly described. More detailed information about the subjects covered in this experiment can be found in references [1, 2, 3] and equipment manuals (the laboratory TA is responsible for having the manuals available if requested).

2.1. Electric Components

Resistor: A resistor is a dissipative element that converts electric energy into heat. The voltage-current relationship of an ideal resistor is given by

$$V = RI \tag{2.1}$$

where I is the current flowing through the resistor of resistance R when a voltage V is applied across its terminals. A resistor is conventionally drawn as shown in Figure 1. The unit of resistance is the *Ohm* (Ω).

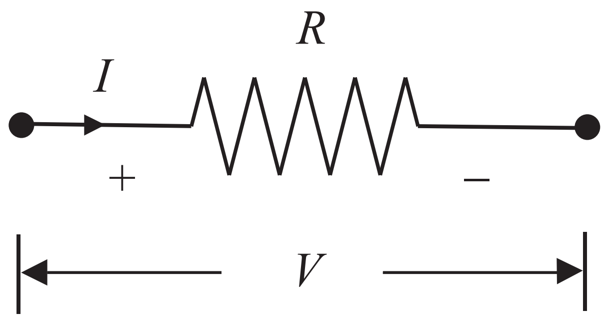

Figure 1: Schematic Representation of a Resistor

The most common resistor is made of carbon and manufactured as a cylinder with axial wire leads. Four colored bands painted around the cylinder body identify the resistance value and its tolerance. The international resistor color code is represented in Figure 2 and Table 1 where the resistance is given by

$$R = AB \times 10^{C} \pm D\%. \tag{2.2}$$

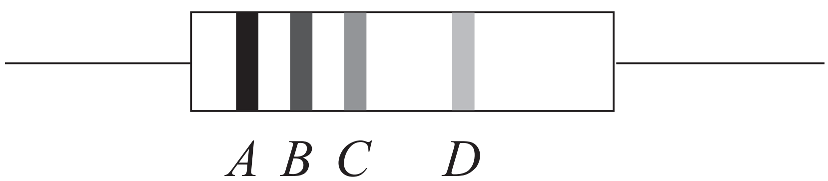

Figure 2: Resistor Color Code

For example, if the color bands from left to right on a resistor are red, violet, orange, and gold, then the resistance is

$$\begin{aligned} R &= \left(27 \times 10^{3} \pm 5\%\right)\Omega \\ &= (27,000 \pm 1,350)\,\Omega. \end{aligned} \tag{2.3}$$

The resistor power rating measures the amount of voltage or current the resistor can handle without being destroyed. Carbon resistors are supplied in $\frac{1}{8}$, $\frac{1}{4}$, $\frac{1}{2}$, 1, and 2 Watt sizes, and are identified by the dimension of their cylindrical body as illustrated in Figure 3. The maximum allowable voltage $V_{\max}$ under which a resistor can operate without damage is another important parameter to consider in the component selection process. For the available resistors in our laboratory $V_{\max} = 30$ Volts.

Table 1: Resistor Color Code

A, B, C bands				D band	
Black	0	Gold (C only)	-1	Gold	±5%
Brown	1	Silver (C only)	-2	Silver	±10%
Red	2			No band	±20%
Orange	3				
Yellow	4				
Green	5				
Blue	6				
Violet	7				
Gray	8				
White	9				

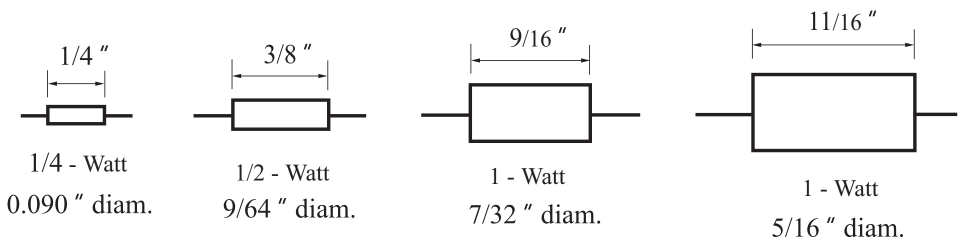

Figure 3: Carbon Resistor Dimensions and Power

Potentiometer: A potentiometer is a resistor whose value can be continuously adjusted. It consists of a carbon or wirewound resistor with a moving tap which can be positioned by either rotating a shaft or sliding a contact rule as shown in Figure 4. Potentiometers are characterized as either finite or infinite turn. The inherent characteristic of an infinite turn potentiometer is that its resistance varies periodically depending on the amount of rotation (typically one full rotation corresponds to the maximum resistance span). In the finite turn potentiometers, the amount of rotations required to expand the full scale of resistance limits the amount of revolutions of the shaft. Potentiometers are schematically represented by the one of the symbols shown in Figure 4.

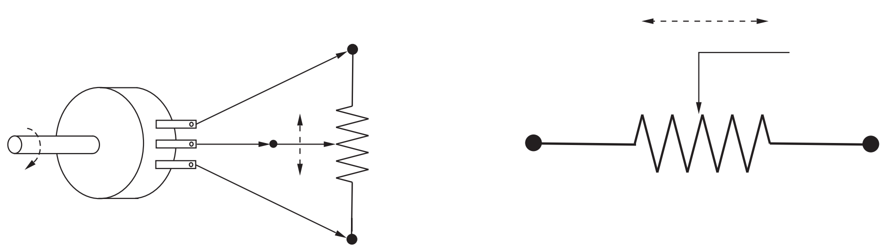

Figure 4: Circular and Sliding Potentiometers

Breadboard: Throughout your experiments whenever electric circuits have to be built, "breadboards" will be used for component mounting and creating artificial wired connections. A breadboard is comprised of a grid of small, contact squares which are internally connected (*i.e.*, short circuited) within a column or row (see Figure 5). Specifically, all contacts in the individual rows r_1, r_2, r_3, and r_4 are short circuited. In a similar manner, all contacts in the individual columns c_1^{top} through c_{29}^{top} and c_1^{bot} through c_{29}^{bot} are short circuited. The terminals of various components (resistors, capacitors, op-amps, *etc.*) are attached to the contacts through a plug–in procedure. For example, Figure 5 shows a resistor that has been connected in cascade with a capacitor. Although the resistor and capacitor terminals are not in direct contact, they share contacts within the same column c_{11}^{bot}. This implies that there is a wired junction between their terminals.

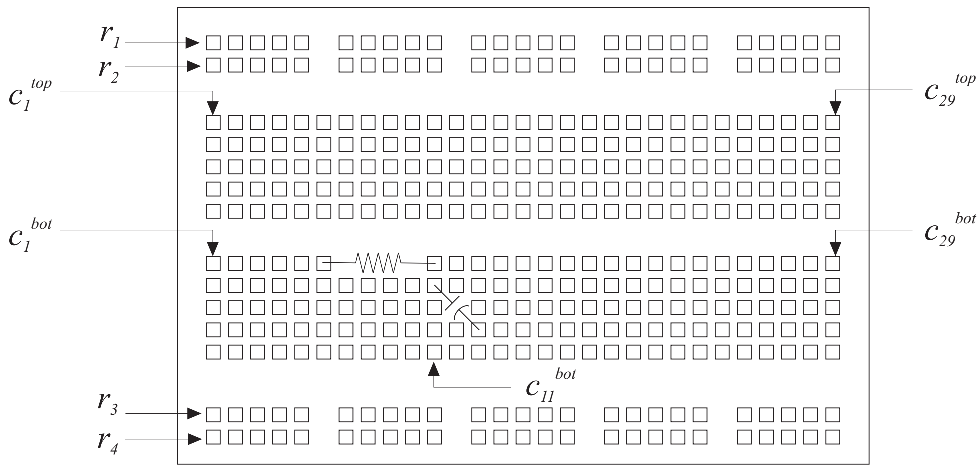

Figure 5: Schematic Representation of a Breadboard

2.2. Kirchhoff's Laws

Kirchhoff's laws are the basis for the analysis of electric circuits. They allow you to calculate voltages and currents anywhere in a circuit.

Current Law: Kirchhoff's current law states that the algebraic sum of all currents entering and leaving a node is zero. For example, at node A of Figure 6

$$I_1 + I_2 - I_3 = 0, \tag{2.4}$$

which can be rewritten as

$$I_3 = I_1 + I_2. \tag{2.5}$$

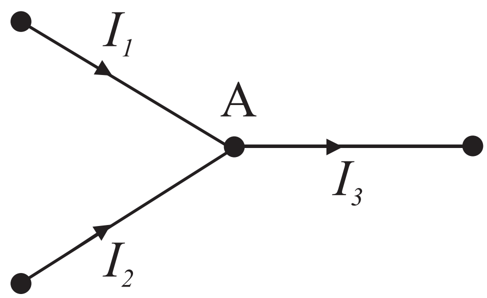

Figure 6: Example of Kirchhoff's Current Law

Voltage Law: Kirchhoff's voltage law states that the algebraic sum of voltages around a closed loop is zero. For example, in the loop of Figure 7

$$-V_1 + V_2 + V_3 + V_4 = 0 \tag{2.6}$$

where it is assumed that the voltage drops across each passive element in the direction of the current flow.

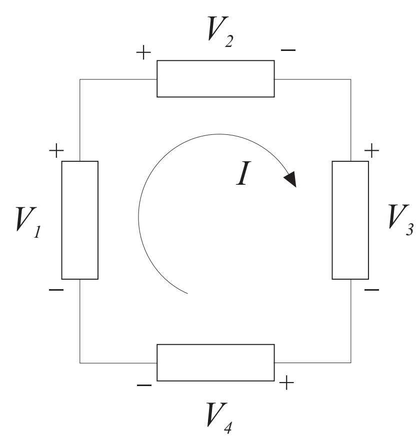

Figure 7: Example of Kirchhoff's Voltage Law

In this section, we provide a brief overview of the hardware and software environment to be used throughout this laboratory course.

2.3. Instruments

Multimeter: A multimeter is an instrument that can selectively measure multiple electrical quantities, such as for example, voltage (voltmeter), current (ammeter), and resistance (ohmmeter), upon the proper adjustment of its dials and buttons. In digital multimeters (to be used in this laboratory), the numerical value of the quantity being measured is directly shown in the instrument's display. The resolution of these devices is usually specified through the number of displayed digits (*e.g.*, $3\frac{1}{2}$ or $4\frac{1}{2}$ digits). When a multimeter is characterized as having a resolution of $X\frac{1}{2}$, it implies that it can provide an accurate measurement for the first X displayed digits within an individual scale. As an example, consider the case of a $3\frac{1}{2}$ digital voltmeter with a scale of 0 to 0.1 Volts, and display measurement presentation of $\boxed{.\text{xxx}}$. For this scale, the voltmeter is accurate within 0.0005 Volts. Note that this is not the voltmeter's resolution when the scale is different than the aforementioned.

Most of the meters used for measuring electric signals employ current detection as their basic indicating mode. An ammeter measures the current flowing between two terminals sharing the same voltage potential within an electric circuit. The ammeter is connected in between (*i.e.*, in *series* with) the two terminals as shown in Figure 8(a). This measurement can be achieved only if the current can flow through the ammeter. Therefore, the ideal ammeter should act as a short circuit (*i.e.*, $R_{in} = 0$ where R_{in} is the internal resistance of the meter) since it should not modify the circuit characteristics. The voltmeter measures the potential difference between a pair of nodes of an electric circuit. The voltmeter is connected in *parallel* with the portion of the circuit between the pair of nodes as illustrated in Figure 8(b). This measurement can be achieved only if *no* current flows through the voltmeter. Therefore, the ideal voltmeter should respond as an open circuit to the node pair ($R_{in} = \infty$). An ohmmeter is usually embedded within a voltmeter; hence, it is connected in parallel with the resistance to be measured.

The resistance R between two nodes can be calculated if the current flowing through the resistance and the applied voltage are known; *i.e.*, using (1), we have that $R = \frac{V}{I}$. An ohmmeter (usually embedded within a voltmeter) applies a known, reference voltage V_{ref} to the nodes and measures the current to then obtain the resistance by $R = \frac{V_{ref}}{I}$. To avoid damage to the multimeter, no other voltage should be applied to the nodes at the same time. The typical resistance range is 1 Ω to 32 MΩ for most multimeters.

Power Supply: A power supply is an electric device that provides a known DC voltage in place of a battery. A power supply is plugged into a wall outlet, and can typically deliver voltages ranging from 0 to 30 Volts. In addition, many power supplies provide separate, fixed ± 12 Volts and/or ± 5 Volts output terminals since these voltages are commonly used to power active electric components

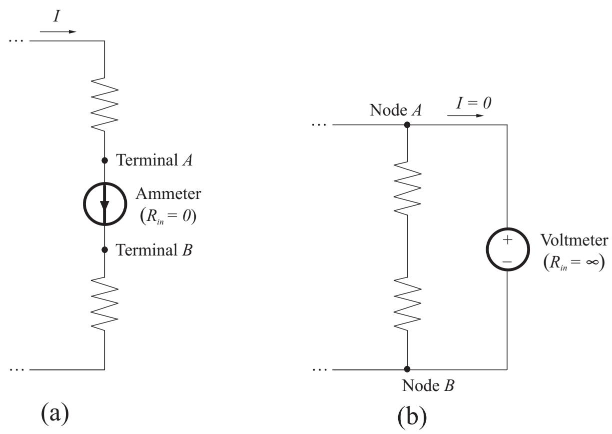

Figure 8: Ideal Ammeter and Voltmeter Measurements

such as op-amps and digital integrated circuits.

Function Generator: Whereas power supplies are used to provide DC voltages to a circuit, a function generator provides periodic AC voltages. This equipment allows you to vary not only the amplitude of the periodic AC signal, but also its waveform. Typically, sinusoidal, square, and triangular waveforms are included in most commercially available function generators.

Oscilloscope: Oscilloscopes are used for detailed monitoring of AC and DC electric signals in measurement systems. Whereas digital multimeters display the numerical value of a voltage or current signal, an oscilloscope displays signals as waveforms on a screen with high accuracy and precision. Dials or buttons on the equipment's front panel allow the modification of the amplitude and time scales of the signal being displayed. Typical oscilloscopes have two to four channels for signal monitoring. Signals are accessed via probes or clamps connected to oscilloscope channels.

3. Laboratory Practices

3.1. Safety Rules

While working in a laboratory environment, particular attention should be paid in the avoidance of electric shocks. Several important safety considerations must be observed while constructing the experiments.

The real measure of a shock's intensity is measured by the amount of current forced through the body. This implies that a voltage of 10 Volts can be as deadly as 5000 Volts. Any current amount over 10 mA can cause a painful shock. The lethal current range is between 100 to 200 mA, where ventricular fibrillation of the heart occurs. Above this range, the resulting muscular contractions are so severe that the ventricular fibrillation is prevented as shown in Figure[1] 9. If an electric shock occurs, **turn the power off and/or remove the victim** as quickly as possible without endangering yourself. The resistance of the victim decreases with time and the fatal current may be reached if action is delayed. If the victim has stopped breathing or is unconscious, start artificial respiration and call a medical authority.

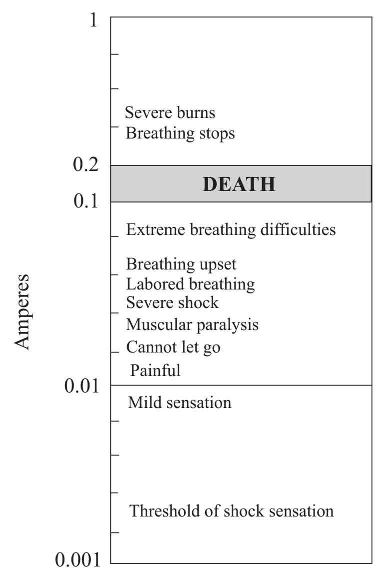

Figure 9: Physical Effects of Electric Shocks

3.2. Circuit Construction Rules

While laying out the circuits for testing, the following rules should be observed. First, maintain the same "ground" for *all* devices. Ground is considered to be the *zero–Volts*, voltage reference potential. In the laboratory, several instruments are tied to the Earth's potential for additional security. Power panels in the laboratory have terminals which are interconnected and tied to the Earth's ground. Instruments (power supplies, function generators, oscilloscopes, *etc.*) may connect to the AC line with a three–terminal plug. The round non current–carrying contact is normally connected to the instrument chassis (see Figure 10).

[1]Note that this figure is just a rough guideline of the effects of electric shocks in terms of the amount of current.

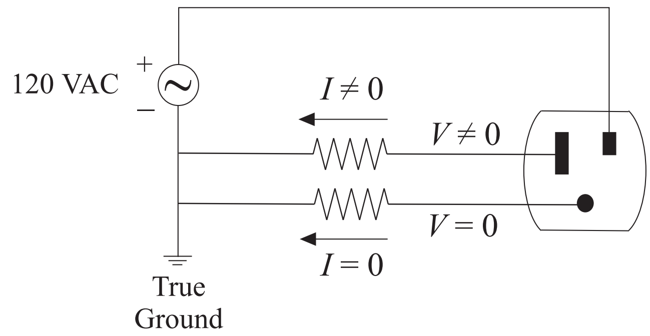

Figure 10: AC Three Terminal Plug

According to this, all instruments have their individual chassis interconnected through the grounding system of the power line. The individual instrument grounds should be wired appropriately, otherwise short circuits may occur. To avoid short circuits, several circuits have their terminals floating; *i.e.*, neither terminal is connected to the ground. This is mostly common in power supplies, where two terminals provide the appropriate output and the third one is connected to the instrument chassis. In this way, either terminal can be grounded, or both left floating as depicted in Figure 11.

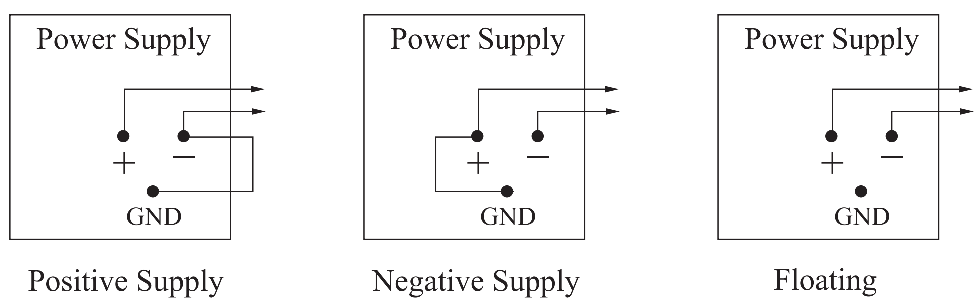

Figure 11: Terminal Connections for Power Supply Output

Secondly, neatness should be among the primary considerations when laying out your circuits. Avoid using very short or very long leads. Use the same colored wire for connecting the instruments' ground. Avoid having any loose wires in your setup and refrain from insecure connections. Use shielded wires for low–level signals. To eliminate the effects of magnetic fields induced by the 60-Hz power line and the high frequency signals from the computers, you may want to provide a magnetic shield (*e.g.*, aluminum foil enclosure) to your components.

4. Equipment List

i) Four known resistors ($\frac{1}{4}$ Watts, 5% uncertainty) $R_k = 10\ \Omega$, $100\ \Omega$, $1\ \mathrm{K}\Omega$, and $1\ \mathrm{M}\Omega$.

ii) Four unknown resistors R_{ui}, $i = 1, \ldots, 4$ ($\frac{1}{4}$ Watts, 5% uncertainty).

iii) One circular potentiometer ($\frac{1}{10}$ Watts, 20% uncertainty).

iv) Breadboard and a set of leads.

v) Digital multimeter, power supply, function generator, and oscilloscope.

vi) Set of BNC cables and function generator/oscilloscope probes.

5. Experimental Procedure

In this experiment, we will use a digital multimeter and an oscilloscope to measure resistance values in serial and parallel connection with DC and AC input signal, respectively.

5.1. Multimeter Measurements

i) Using the digital multimeter as an *ohmmeter*, measure the value of the unknown resistances R_{ui}.

ii) Connect any two of the known resistors in *series* on the breadboard. Measure the series resistance with the ohmmeter. Repeat the same procedure with the two resistors connected in *parallel*.

iii) Select an unknown resistor R_{ui} and a known resistor R_k of approximately the same order as the chosen R_{ui}. Connect the resistors in series and apply an appropriate voltage[2] to the series connection using the power supply. A schematic representation of this circuit is shown in Figure 12. **Before proceeding, you must request the laboratory TA to approve your electrical connections**. By using the digital multimeter as a *voltmeter*, devise a way of calculating the unknown resistance R_{ui} from some voltage measurement in the circuit. Call this new measurement of the unknown resistance, R^v_{ui}. Now, by using

[2]Be careful not to choose a large voltage. Take into consideration the power ratings of the resistors.

the multimeter as a *ammeter*, devise a way of calculating the unknown resistance R_{ui} from the current measurement. **Make sure that the ammeter has series connection in the circuit for current measurement. See Figure 8 for the proper ammeter connection**. Call this new measurement of the unknown resistance, R_{ui}^a.

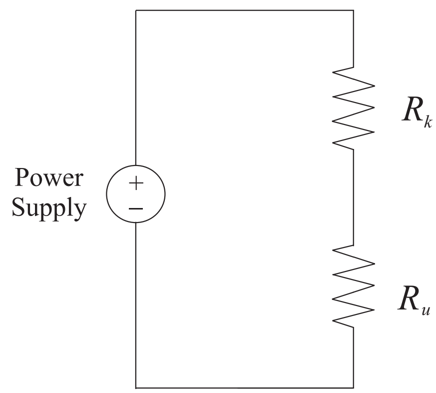

Figure 12: Series Connection of Resistors with Power Supply

5.2. Oscilloscope Measurements

i) Using a BNC cable, connect the output of the function generator to one of the oscilloscope channels. **Before proceeding, you must request the laboratory TA to approve your electrical connections**. Generate the sinusoidal signal shown in Figure 13 from the function generator for the following four cases $T = 10^{-3}$, 10^{-4}, 10^{-5}, and 10^{-6} sec (f=1kHz, 10kHz, 100kHz, and 1MHz, respectively), and view them on the oscilloscope.

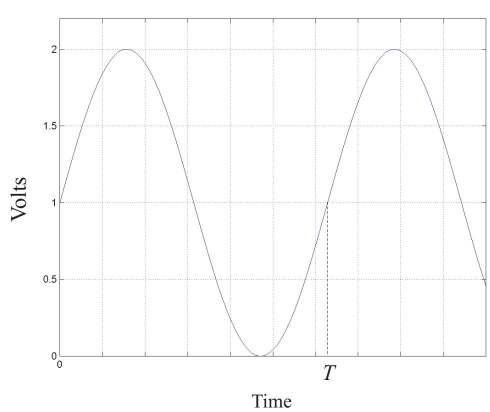

Figure 13: Sinusoidal Waveform

The amplitude scale, time scale, and trigger of the oscilloscope may have to be adjusted to properly view each case. **The TA should verify that the proper signal has been generated.** Adjust the oscilloscope so you *only* view the AC component of the signal. Now, adjust the oscilloscope so you view both the DC and AC components of the signal. Record the DC and AC amplitudes of the signals.

ii) Repeat the above procedure for the periodic signal shown in Figure 14.

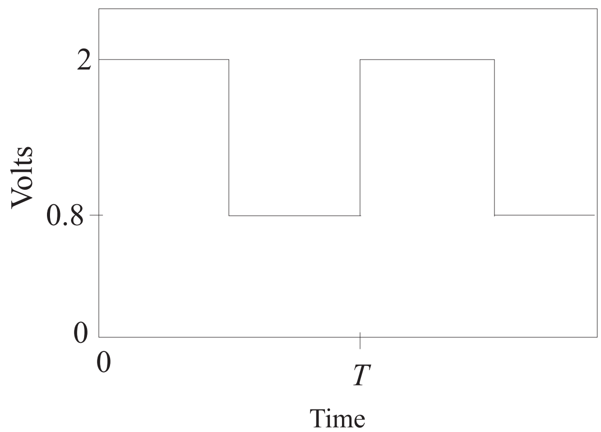

Figure 14: Periodic Square Waveform

iii) Connect any two of the known resistors R_k in series and apply any of the above sinusoidal voltage signals to the series connection using the function generator (*i.e.*, in the circuit of Figure 12, replace the voltage supply with the function generator). **Before proceeding, you must request the laboratory TA to approve your electrical connections**. Using the oscilloscope probes, connect one channel to the applied voltage and the other channel across the bottom resistor (*i.e.*, the one that is now in place of R_u in Figure 12). Make sure the *ground connectors* of the probes are connected to the circuit ground. Setup the oscilloscope so that both the input and output voltages are seen simultaneously on the screen.

iv) Repeat the above procedure for the periodic square voltage signal.

v) Assemble the circuit shown in Figure 15 on the breadboard. Make $V_i = 6\sin(2\pi \times 10^3 t)$ Volts and adjust the potentiometer such that $V_o = \frac{5}{6}V_i$. Now first turn off the function generator, then measure R_p using the ohmmeter.

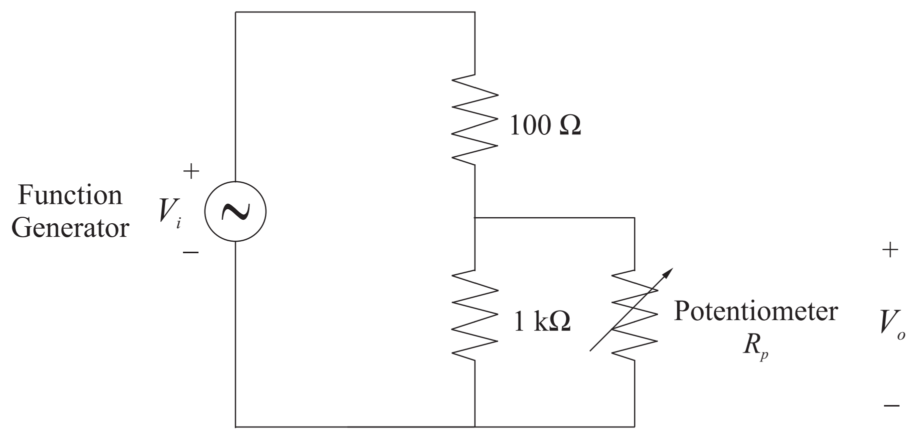

Figure 15: Circuit with Potentiometer

6. Analysis/Assignment

i) In step *i*) of Section 5.1, which do you expect to obtain more accurate measurements for high, medium, or low resistances? Compare the measured values with nominal values from the color code and commment.

ii) In step *ii*) of Section 5.1, what is the expected resistance of your series connection by calculation? Then compare the expected value of the series resistance calculated with the measuerd value with the ohmmeter. Repeat the same analysis with the two resistors connected in *parallel*.

iii) In step *iii*) of Section 5.1, comment on your measurements R_{ui}, R^v_{ui}, and R^a_{ui} obtained in the above experiments.

iv) In steps *i*)—*iii*) of Section 5.1, which method (among color code reading, ohmmeter, voltmeter, and ammeter) provided the most (least) accurate measure of the nominal resistance? Justify your answer.

v) In steps *i*) and *ii*) of Section 5.2, what are the DC and AC amplitudes of the signals?

vi) In steps *iii*) and *iv*) of Section 5.2, describe what you observe. Do your theoretical and experimental results agree? Explain any discrepancies.

vii) In step *v*) of Section 5.2, first calculate the theortical value R_{pcal} of the potentiometer.

Then compare and comment the measured potentiometer value R_p with the calculated value R_{pcal}. Explain any discrepancies.

References

1. W. Bolton, *Mechatronics: Electronic Control Systems in Mechanical and Electrical Engineering*, Addison-Wesley, New York, NY, 1999.

2. M.B. Histand and D.G. Alciatore, *Introduction to Mechatronics and Measurement Systems*, WCB/McGraw-Hill, Boston, MA, 1999.

3. J.W. Nilsson, *Electric Circuits*, Addison-Wesley, Reading, MA, 1996.

Experiment 2: Modern Data Acquisition and First-Order Systems

Tools/concepts emphasized: MATLAB, Simulink, Real-Time Workshop (RTW), QuaRC, Q4, data acquisition.

1. Objectives

i) Use of the computer-based data acquisition software, MATLAB and Simulink, as an instrumentation tool.

ii) Understand the time and frequency responses of first-order systems.

iii) Design passive filters.

2. Background

In Experiment 1, traditional equipments such as the digital multimeter, function generator, and oscilloscope were utilized for instrumentation. With the advent of cheap, high-performance computer technologies, computer-aided instrumentation systems have found their way into laboratories. This experiment introduces one such system, MATLAB and Simulink, which will be exploited throughout most of the remaining experiments in this course.

First-order systems and particularly filters are an integral part of many measurement systems. As such, it is important to understand and predict their time and frequency behaviors when subjected to known input signals. In the following, some pertinent theoretical concepts of first-order systems and passive filters are briefly described.

More detailed information about the subjects covered in this experiment can be found in references [1, 2, 3] and MATLAB manuals.

2.1. MATLAB and Simulink

In this section, we provide a brief overview of the hardware and software environment to be used throughout this laboratory course.

Q4 DACB: The Q4 is a general purpose DACB. It provides 4 single-ended ADCs, 4 DACs, 16 bits of digital inputs, 16 bits of digital outputs, 4 reconfigurable encoder counter/timers, and upto

4 encoder inputs. The Q4 DACB is accessed through the PC bus and is installed on an PCI bus internal to the laboratory PC. The aforementioned functions of the Q4 DACB can be accessed via an external terminal board (See Figure 1).

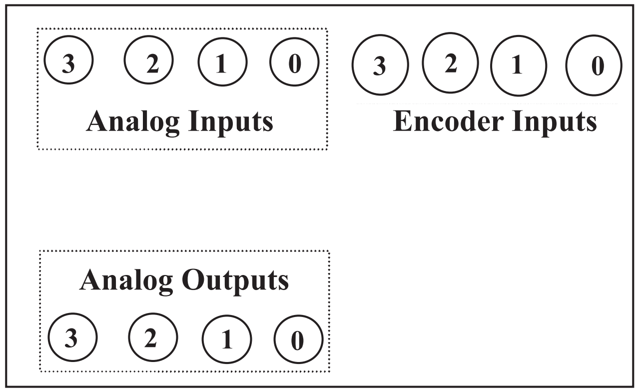

Figure 1: Q4 terminal Board

MATLAB-Simulink-RTW: This is the preferred software environment for the control laboratory. Students enrolled in this laboratory course were familiarized with the MATLAB software in ME 3413. Simulink is a graphical control-system simulation program. The RTW toolbox enables automated C code generation from user-designed Simulink control-system diagrams.

QuaRC_Library: This is a library of Quanser-supplied DACB drivers (e.g., Q4) compatible with Simulink (See Figure 2). Some commonly used blocks of the Q4 library are HIL Read Analog (ADC), HIL Write Analog (DAC), and HIL Read Encoder (See Figure 3).

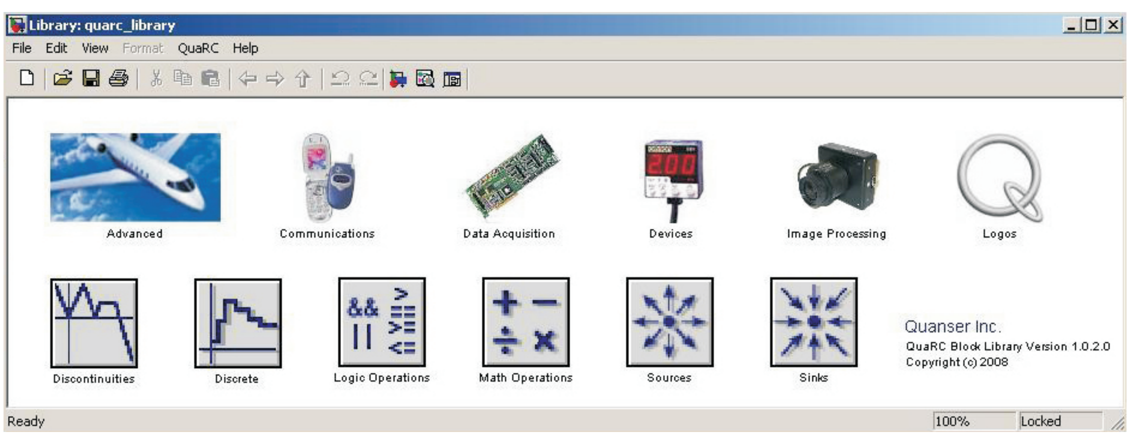

Figure 2: QuaRC_Library Block Library

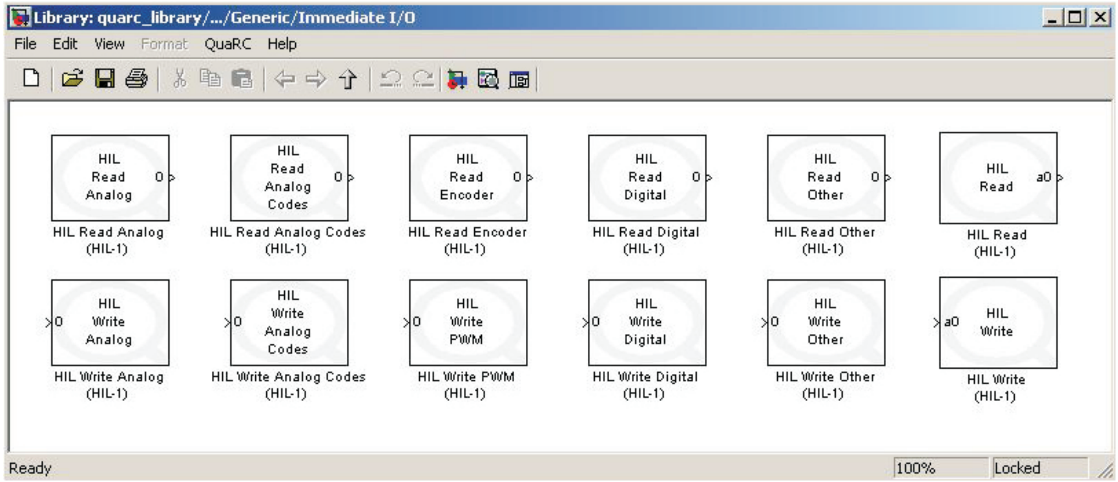

Figure 3: Immediate I/O Block Library

QuaRC: The QuaRC program interfaces the Simulink generated C code with the Q4 board in a seamless manner. The QuaRC program consists of two principal components, *viz.*, QuaRC client and QuaRC server. The QuaRC client is installed on the host computer with the Q4 DACB. The QuaRC server may be installed on the host or the remote computer. The user designs a Simulink control diagram and generates the C code on the remote computer. The C code from the remote computer is transferred to the host computer by the QuaRC server. The QuaRC client and host computer's processor communicate with the Q4 DACB for real-time data acquisition and control. The QuaRC client also relays the real-time data to the QuaRC server for plotting purposes.

2.2. First-Order Systems

Many components of measurement systems can be modeled as a linear differential equation with constant coefficients (*e.g.*, accelerometer, filter, *etc.*):

$$a_n \frac{d^n y(t)}{dt^n} + a_{n-1} \frac{d^{n-1} y(t)}{dt^{n-1}} + \cdots + a_o y(t) = b_m \frac{d^m u(t)}{dt^m} + b_{m-1} \frac{d^{m-1} u(t)}{dt^{m-1}} + \cdots + b_0 u(t), \qquad (2.1)$$

where $y(t)$ is the system output variable, $u(t)$ is the input variable, the a_i's and b_i's are the constant coefficients, and n defines the order of the system. When $n = 1$ and $m = 0$ or 1, the system is referred to as a *first-order system*. For example, for the case where $m = 0$ and $b_0 = 1$, (2.1) reduces to

$$a_1 \frac{dy(t)}{dt} + a_o y(t) = u(t). \qquad (2.2)$$

Applying the Laplace transform to (2.2), the input-output transfer function of (2.2) can be obtained as[1]

$$\frac{Y(s)}{U(s)} = \frac{\frac{1}{a_0}}{\tau s + 1}, \tag{2.3}$$

where $\tau = \frac{a_1}{a_0}$ is called the *time constant* of the system. For the case where $u(t)$ is a step input of amplitude A, the response of (2.2) is given by

$$y(t) = (y(0) - y_\infty)\, e^{-\frac{t}{\tau}} + y_\infty, \tag{2.4}$$

where the constant $y_\infty = \frac{A}{a_0}$ is called the *steady-state* value of $y(t)$. Figure 4 shows the step response of (2.4) for the case where $y(0) = 1$, $y_\infty = 4$, and $\tau = 1$ sec.

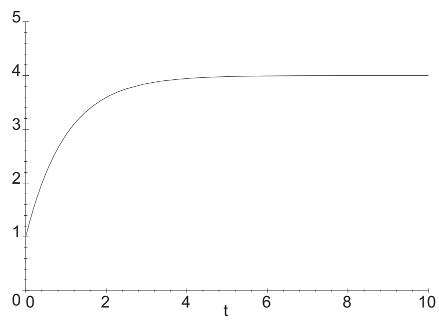

Figure 4: Typical Step Response of a First-Order System

For case where $u(t)$ is a harmonic signal, *i.e.*, $u(t) = A\sin(\omega t)$, the response of (2.2) is given by

$$y(t) = y(0)e^{-\frac{t}{\tau}} + \frac{A}{a_0\sqrt{1+\omega^2\tau^2}} \sin(\omega t + \phi), \qquad \phi = -\tan^{-1}(\omega\tau), \tag{2.5}$$

where ϕ denotes the phase shift in radians. Figure 5 shows the harmonic response of (2.5) for the case where $y(0) = 2$, $a_0 = 2$, $\tau = 0.5$ sec, $A = 1$, and $\omega = 1$ rad/sec.

2.3. Passive Filters

The term *filtering* is used to describe the process of removing certain frequency bands from an electric signal and allowing other frequencies to be transmitted. Filters are particularly useful in instrumentation since most sensors produce signals containing spurious noise which contaminate the measured quantity. Filters are then designed to extract the undesirable noise from the sensor

[1] Recall that the transfer function is obtained by setting the initial conditions to zero. For (2.2), this means $y(0) = 0$.

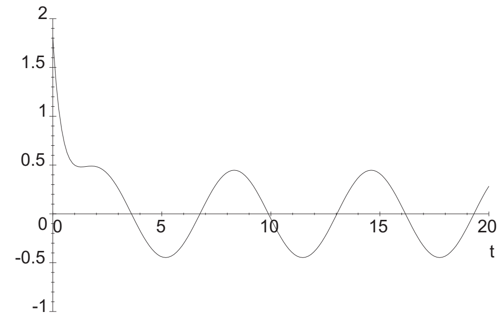

Figure 5: Typical Harmonic Response of a First-Order System

signal. The term *passive* is used to denote filters which are constructed using only passive electric components such as resistors, capacitors, and inductors[2]

A typical example of the first-order system of (2.2) is a passive, RC *low-pass* filter. This filter is implemented by the simple circuit shown in Figure 6. To see how the mathematical model of the circuit in Figure 6 is given by (2.2), one must first know the voltage-current relationship of a capacitor.

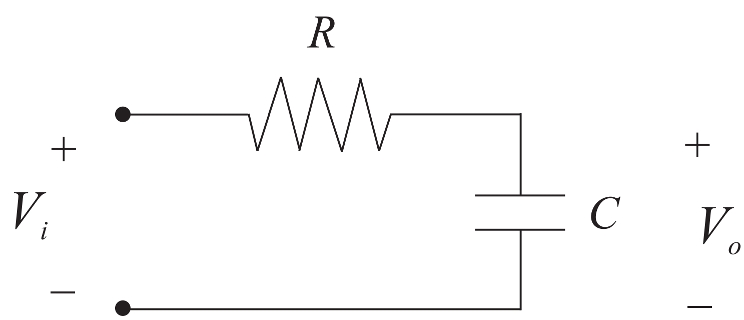

Figure 6: Passive, RC Low-Pass Filter

A capacitor is a passive element that stores energy in the form of an electric field. A capacitor is constructed by introducing a nonconducting medium within the gap between two conductors. The voltage-current relationship of a capacitor is given by

$$V = \frac{1}{C} \int I(t)dt, \tag{2.6}$$

where V is the voltage across the capacitor, I is the current arriving at the capacitor terminal, and C is the capacitance of the capacitor in *Farads* (F). Note that by differentiating (2.6) with respect

[2] *Active* filters (those which also involve the use of operational amplifiers) will be the topic of Experiment 3.

to time, the voltage-current relationship of a capacitor can also be written as

$$I = C\frac{dV}{dt}. \tag{2.7}$$

Capacitors are manufactured in a wide variety of shapes and materials. Electrolytic types are usually cylindrical with axial leads. Electrolytic capacitors require the same voltage polarity, and they typically carry large capacitance values (μF and in some cases mF). Tantalum capacitors denoted by their light blue coating color exhibit the highest capacitances per volume. Ceramic capacitors (to be used in this laboratory) are round or rectangular in shape.

Now, to show that the mathematical model for the low-pass filter is given by (2.2), one has to simply apply Kirchhoff's voltage law to the circuit of Figure 6 along with the relationship of (2.7). As a result, the following transfer function is obtained

$$\frac{V_o(s)}{V_i(s)} = \frac{1}{RCs+1}, \tag{2.8}$$

where the time constant $\tau = RC$. Figure 7 shows the frequency response of a first-order, low-pass filter (*i.e.*, Bode plot of (2.8)) for the case where $\tau = 10^{-2}$ sec. Note the magnitude attenuation for frequencies higher than the *cutoff* frequency $\omega_c = \frac{1}{\tau} = 100$ rad/sec.

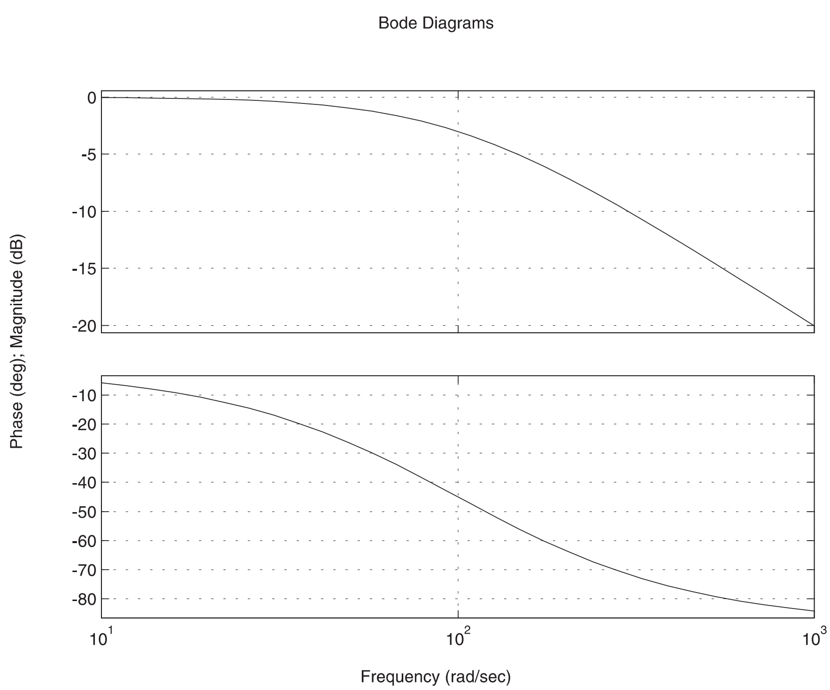

Figure 7: Typical Frequency Response of a First-Order, Low-Pass Filter

A first-order, *high-pass* filter is obtained if the output voltage V_o in the circuit of Figure 6 is now taken to be the voltage across the resistor. In this case, the transfer function will be

$$\frac{V_o(s)}{V_i(s)} = \frac{RCs}{RCs+1}. \tag{2.9}$$

Note that this transfer function is representative of a first-order system where $m = 1$ in (2.1). Figure 8 shows the frequency response of a first-order, high-pass filter (*i.e.*, Bode plot of (2.9)) for the case where $RC = 10^{-2}$ sec.

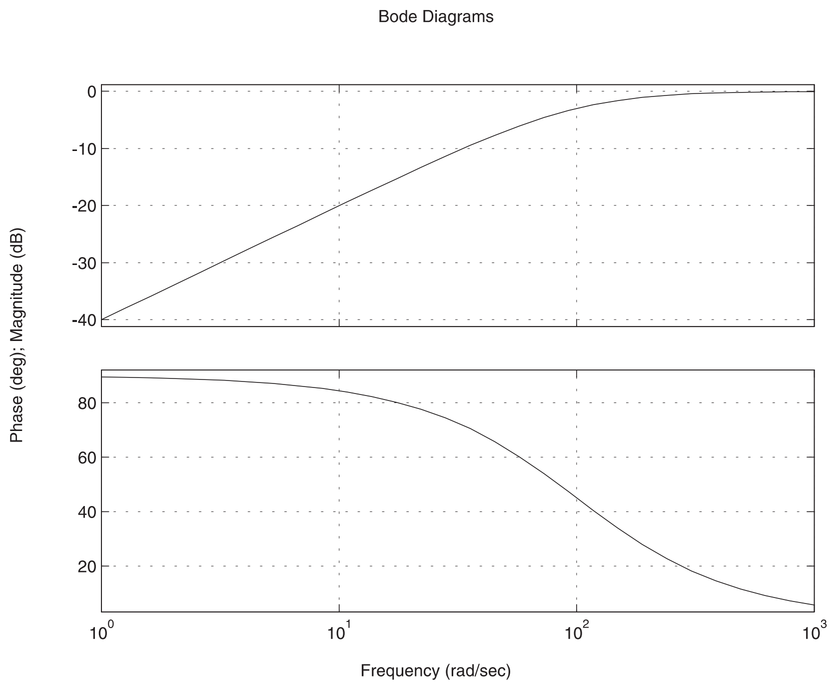

Figure 8: Typical Frequency Response of a First-Order, High-Pass Filter

3. Equipment List

i) PC with Q4 DACB and terminal board

ii) Software environment: Windows, MATLAB, Simulink, RTW, and QuaRC

iii) Set of known resistors and capacitors.

iv) An unknown capacitor C_u

v) Breadboard, a set of leads, and connectors

4. Experimental Procedure

In this experiment, we will design a first-order system and study system identification by using a PC-based data acquisition software.

4.1. Time Response of First-Order Systems

i) Assemble the circuit shown in Figure 9 on the breadboard using a known resistor and an unknown capacitor C_u. Using the Q4 terminal board and RCA–Alligator cables, connect the input of the circuit to the channel 0 of DAC (analog output) and the output to the channel 0 of ADC (analog input), as illustrated in Figure 1. **Before proceeding, you must request the laboratory TA to approve your electrical connections**.

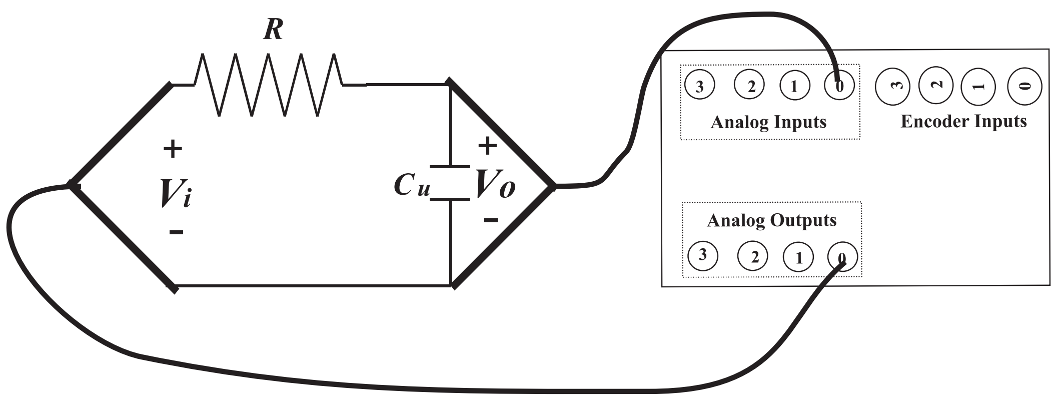

Figure 9: Simulink Block Diagram for the Loop-Back Experiment

ii) From the **Start** button of the Windows toolbar, select the option sequence **Programs–MATLAB–R2007b–MATLAB R2007b** to launch the MATLAB application.

iii) In the MATLAB window, choose "C:\MeasurementSystemsLab\Experiment2" from the Current Directory window. This directory path choice will change the directory from the default MATLAB directory to the working directory for Experiment 2.

iv) From the **File** menu of the MATLAB window, select the option **Open** to load the Simulink block-diagram "Experiment2A.mdl." shown in Figure 10 to your desktop. This will load the files for time response of first-order system experiment.

v) In the MATLAB window, apply a *unit* step input voltage to the circuit by setting the

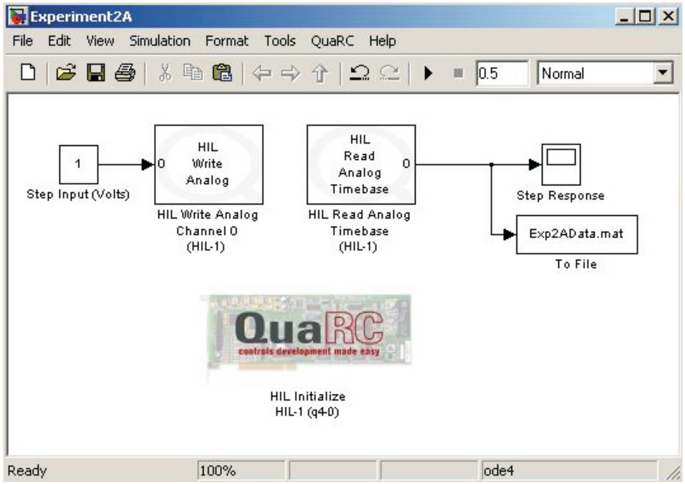

Figure 10: Simulink Block Diagram for Time Response of First-Order Systems Experiment

value of the constant in the Constant block to 1. click the black **Start** arrow button to acquire the real-time data for the experiment.

vi) In the Simulink block-diagram, the To File block creates "Exp2AData.mat" which the plot data is saved on. Plot the step response from the MATLAB window by executing the following commands: load Exp2AData and plot(data2A(1,:),data2A(2,:)).

4.2. Frequency Response of Passive Filters

i) Build a first-order, passive, low-pass filter that provides "good" attenuation of signals with frequencies $f \geq 15$ Hz (note that this frequency is given in Hz, not rad/sec). **Before proceeding, you must request the laboratory TA to approve your electrical connections**.

ii) From the **File** menu of the MATLAB window, select the option **Open** to load the Simulink block-diagram "Experiment2B.mdl." shown in Figure 11 to your desktop.

iii) Click the black **Start** arrow button to acquire the real-time data for the experiment. In the MATLAB window, apply various sinusoidal inputs (starting from 1Hz to 2kHz) to the circuit by changing the value of the frequency in the Signal Generator block. Collect the magnitude of output signal with respect to the input signal frequency.

iv) After sufficient experimentation, press the black **Stop** square button in the MATLAB

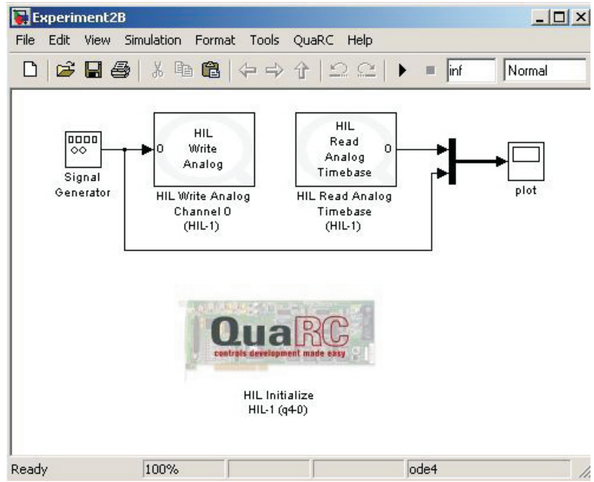

Figure 11: Simulink Block Diagram for Frequency Response of Passive Filters Experiment

window to stop execution of your program on the Q4 DACB.

v) Build a first-order, passive, high-pass filter that provides "good" attenuation of signals with frequencies $f \leq 40$ Hz. Repeat the same stpes *ii*)—*iv*) of Section 4.2.

vi) Build a circuit shown in Figure 12 by using the same values of resistors and capacitors used in the step *i*) of Section 4.2. Repeat the same stpes *ii*)—*iv*) of Section 4.2.

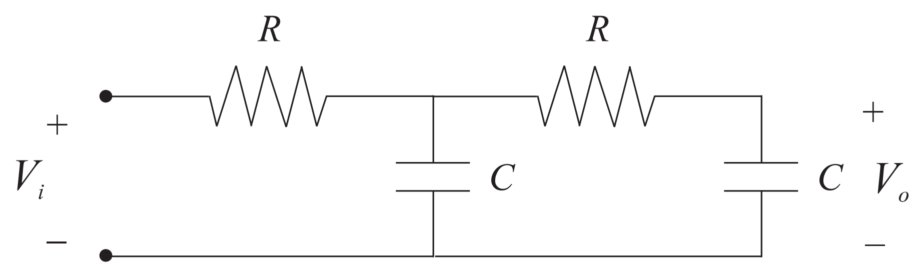

Figure 12: Passive, Second-order RC Low-Pass Filter

5. Analysis/Assignment

i) In step *i*) of Section 4.1, devise a way of identifying the unknown capacitance C_u (Hint: use equation (2.4) and the definitions of τ and y_∞). Compare the calculated value of C_u with its nominal value (use the capacitance meter on a multimeter to provide the nominal value). Note that the terminal board of the Q4 data acquisition card introduces a capacitance of 1 μF in parallel to the unknown capacitor C_u.

ii) In steps *i*)—*v*) of Section 4.2, plot your experimental results in a semi-log paper with the magnitude in decibels and frequency in rad/sec. What is the transfer function of the filter you built? Plot the theoretical frequency response using MATLAB and compare it to the experimental one.

iii) In step *vi*) of Section 4.2, derive the transfer function of the circuit shown in Figure 12. By using MATLAB, plot Bode plot with the transfer function you obtained. (MATLAB command for Bode plot is BODE. For more information, type "help bode" in MATLAB windows.) Plot your experimental results in a semi-log paper with the magnitude in decibels and frequency in rad/sec. Compare the theoretical magnitude frequency response with experimentally determined magnitude frequency response. Comment on the differences.

References

1. W. Bolton, *Mechatronics: Electronic Control Systems in Mechanical and Electrical Engineering*, Addison Wesley, New York, NY, 1999.

2. M.B. Histand and D.G. Alciatore, *Introduction to Mechatronics and Measurement Systems*, WCB/McGraw-Hill, Boston, MA, 1999.

3. J.W. Nilsson, *Electric Circuits*, Addison Wesley, Reading, MA, 1996.

Experiment 3: Operational Amplifier Circuits

Tools/concepts emphasized: Op-Amps, amplifiers, and active filters.

1. Objectives

i) Understand the main characteristics of operational amplifiers (op-amps).

ii) Analyze and implement amplifier circuits using op-amps, including inverting, non-inverting, summing, integrator, and differentiator amplifiers.

iii) Analyze and implement active filters using op-amps, including low-pass, high-pass, notch, and band-pass filters.

2. Background

The following sections provide some fundamental concepts behind op-amps, and their use in the design of amplifier circuits and active filters. More detailed information about the subjects covered in this laboratory can be found in references [1, 2, 3].

2.1. Op-Amps

The op-amp is a high-gain amplifier which uses feedback to control its performance characteristics. Internally, it consists of several serially-connected transistor amplifiers. Externally, it is represented by the symbol shown in Figure 1. Functionally, the op-amp contains one output terminal which is controlled by two input terminals. The two input terminals are labeled positive and negative or non-inverting and inverting, respectively. The positive input is in phase with the output, and the negative input is 180° out of phase with the output.

Two basic rules are used to analyze ideal op-amp circuits:

- There is no current flow through the op-amp input terminals ($I_{oa} = 0$).
- The voltage drop across the input terminals is zero ($V_{oa} = 0$).

These rules are adequate for most design work. In fact, circuits of moderate complexity can be analyzed with these rules.

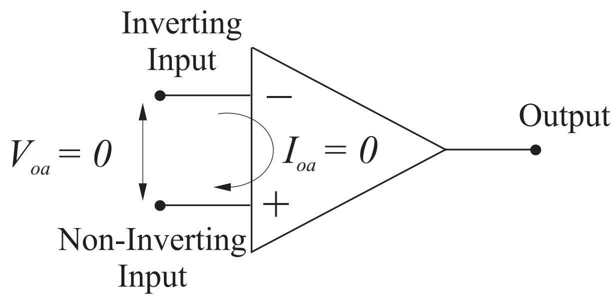

Figure 1: Schematic Representation of an Op-Amp

The op–amp that will be used in this laboratory is based on the National Instruments 741 op–amp which can operate with signals of up to 1.5 MHz. This op–amp needs two voltage supplies, $V^+ = 12$ Volts and $V^- = -12$ Volts. The LM348 component, which includes four 741 op–amps, will be used in this laboratory. The pinout diagram for the LM348 chip appears in Figure 2.

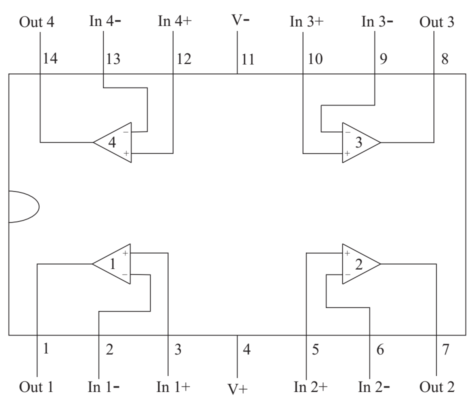

Figure 2: Pinout Diagram for the LM348 Op-Amp Chip

2.2. Amplifiers Circuits

Inverting Amplifier: The inverting amplifier is given by the circuit shown in Figure 3. The circuit amplifies and inverts the input voltage V_i. Its transfer function can be computed as

$$A = \frac{V_o}{V_i} = -\frac{R_f}{R_i} \tag{2.1}$$

where R_f and R_i correspond to the feedback and input resistances, respectively, and the "−" sign indicates that the output voltage V_o is 180° out of phase with the input voltage V_i. Note that the resistor R_p does not influence the transfer function; it is merely used to minimize the effects of the op-amp's input bias current. The optimal value for R_p is

$$R_p^{opt} = \frac{R_i R_f}{R_i + R_f}. \tag{2.2}$$

The optimal value for R_f can be found to be

$$R_f^{opt} = \sqrt{\frac{R_{id} R_o (1 + A)}{2}} \tag{2.3}$$

where R_{id} and R_o represent the op-amp's differential input and output resistances, respectively. For the 741 op-amp (to be used in this laboratory), $R_{id} = 6$ MΩ and $R_o = 70$ Ω. For most designs a value of $R_p = 0$ Ω is sufficient.

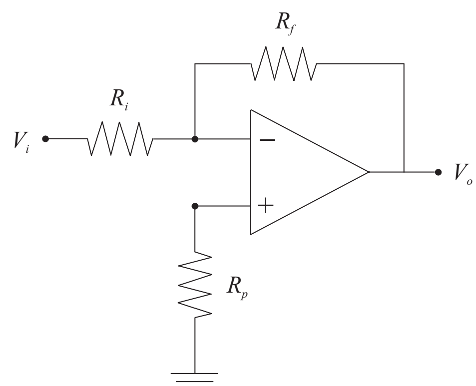

Figure 3: Inverting Amplifier Circuit

Non-Inverting Amplifier: The non-inverting amplifier is given by the circuit shown in Figure 4. The circuit amplifies the input voltage V_i without inverting the signal. Its transfer function can be computed as

$$A = \frac{V_o}{V_i} = 1 + \frac{R_f}{R_i} \ . \tag{2.4}$$

The effect of R_p is similar to the one for the inverting amplifier.

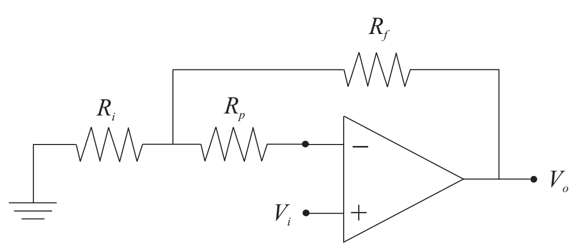

Figure 4: Non-Inverting Amplifier Circuit

Summing Amplifier: The summing amplifier is given by the circuit shown in Figure 5. This circuit gives the weighted, negative sum of the two input voltages V_{i1} and V_{i2}, *i.e.*,

$$V_o = -\frac{R_f}{R_1}V_{i1} - \frac{R_f}{R_2}V_{i2}\ . \tag{2.5}$$

Note that in the special case where $R_1 = R_2 = R_f$, the output voltage simply becomes

$$V_o = -\left(V_{i1} + V_{i2}\right)\ . \tag{2.6}$$

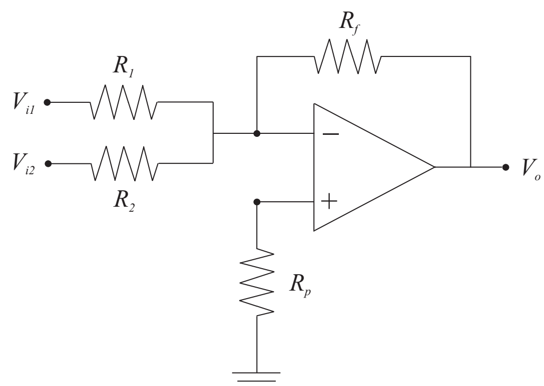

Figure 5: Summing Amplifier Circuit

Differentiator Amplifier: The differentiator amplifier is given by the circuit shown in Figure 6. This circuit produces an output voltage that is ideally proportional to the time derivative of the input voltage, *i.e.*,

$$V_o(t) = -C_i R_f \frac{dV_i}{dt}. \tag{2.7}$$

The application of the Laplace transform to (2.7) yields the following *ideal* transfer function for the differentiator amplifier

$$A_{ideal}(s) = \frac{V_o(s)}{V_i(s)} = -C_i R_f s. \tag{2.8}$$

Note that this transfer function is not casual since the magnitude frequency response tends to infinity as the frequency increases. In reality, the *actual* transfer function of the differentiator amplifier is

$$A_{actual}(s) = \frac{V_o(s)}{V_i(s)} = -\frac{f_{o1} f_{o2} C_i R_f s}{(s + f_{o1})(s + f_{o2})} \tag{2.9}$$

where the corner frequencies f_{o1} and f_{o2} depend on the internal structure of the op-amp. Typical values for the 741 op-amp are $f_{o1} = 20$ kHz and $f_{o2} = 100$ kHz. Also, note that R_p and C_p do not

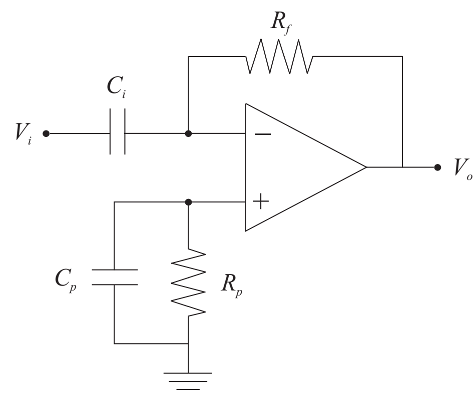

Figure 6: Differentiator Amplifier Circuit

influence the transfer function. While the role of R_p was previously described, C_p is used to bypass the thermal noise of R_p to ground. C_p is added only if $R_p > 100\ \mathrm{k}\Omega$.

Integrator Amplifier: The integrator amplifier is given by the circuit shown in Figure 7. This circuit produces an output voltage that is ideally proportional to the time integral of the input voltage, *i.e.*,

$$V_o(t) = -\frac{1}{R_i C_f}\int_{t_0}^{t} V_i(\tau)d\tau \ . \tag{2.10}$$

The application of the Laplace transform to (2.10) yields the following transfer function for the integrator amplifier

$$A(s) = \frac{V_o(s)}{V_i(s)} = -\frac{1}{R_i C_f s}. \tag{2.11}$$

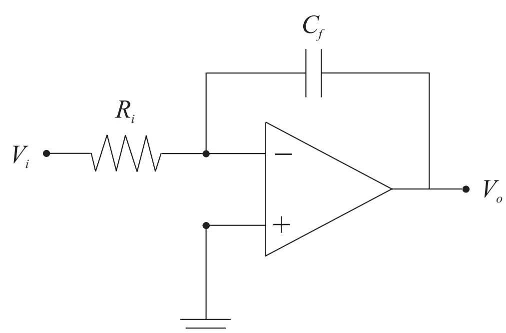

Figure 7: Integrator Amplifier Circuit

2.3. Active Filters

Active filters are used extensively in signal conditioning applications. Specifically, they are used for shaping the frequency responses of sensors. The frequency component of sensor signals can be emphasized or attenuated depending on the application. As opposed to passive filters, active filters utilize op–amps to achieve, if desired, large gains and improved circuit stability. Active filters can be classified into low-pass, high-pass, notch, and band-pass filters.

First-Order Low-Pass Filter: A first-order, low-pass filter is given by the circuit shown in Figure 8. Its transfer function can be computed as

$$\frac{V_o(s)}{V_i(s)} = -\frac{R_f}{R_i}\ \frac{1}{R_f C_f s + 1}. \tag{2.12}$$

It is interesting to compare (2.12) with the transfer function of the *passive*, low-pass filter given by Eq. (8) of Laboratory 2. Note that, whereas the passive filter provides no DC ($s = 0$) amplification, the active filter has a DC gain of $\frac{R_f}{R_i}$.

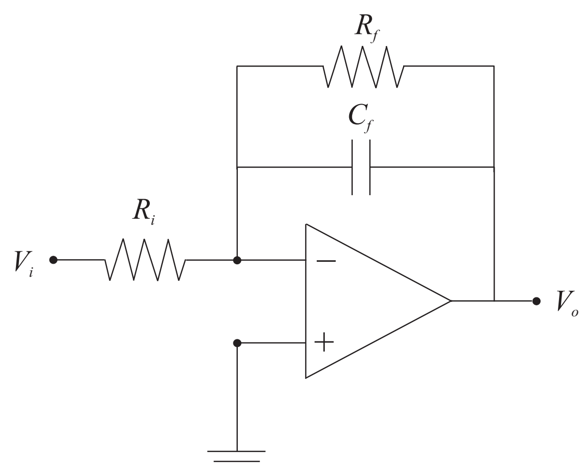

Figure 8: Active, First-Order, Low-Pass Filter

Second-Order Low-Pass Filter: A second-order, low-pass filter is implemented by the circuit shown in Figure 9. Its transfer function is given by

$$\frac{V_o(s)}{V_i(s)} = \frac{\frac{1}{R_1R_2C_1C_2}}{s^2 + \frac{R_1+R_2}{R_1R_2C_1}s + \frac{1}{R_1R_2C_1C_2}} = \frac{\omega_n^2}{s^2 + 2\zeta\omega_n s + \omega_n^2} \tag{2.13}$$

where ω_n is the natural frequency and ζ is the damping ratio, which are explicitly defined as

$$\omega_n = \sqrt{\frac{1}{R_1R_2C_1C_2}} \qquad\qquad \zeta = \frac{R_1+R_2}{2}\sqrt{\frac{C_2}{R_1R_2C_1}}. \tag{2.14}$$

From (2.14), it is clear that by a proper selection of the resistors and capacitors, the natural frequency and damping ratio of the filter can be specified. Figure 10 shows the frequency response of (2.13) for two cases: $\omega_n = 100$ rad/sec and $\zeta = 0.1$ (solid line), and $\omega_n = 100$ rad/sec and $\zeta = 1$ (dashed line).

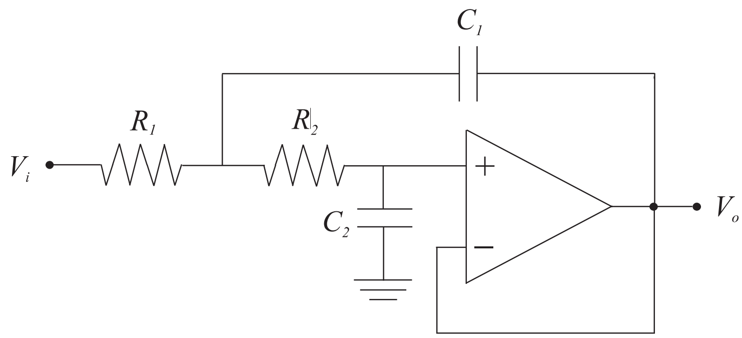

Figure 9: Active, Second-Order, Low-Pass Filter

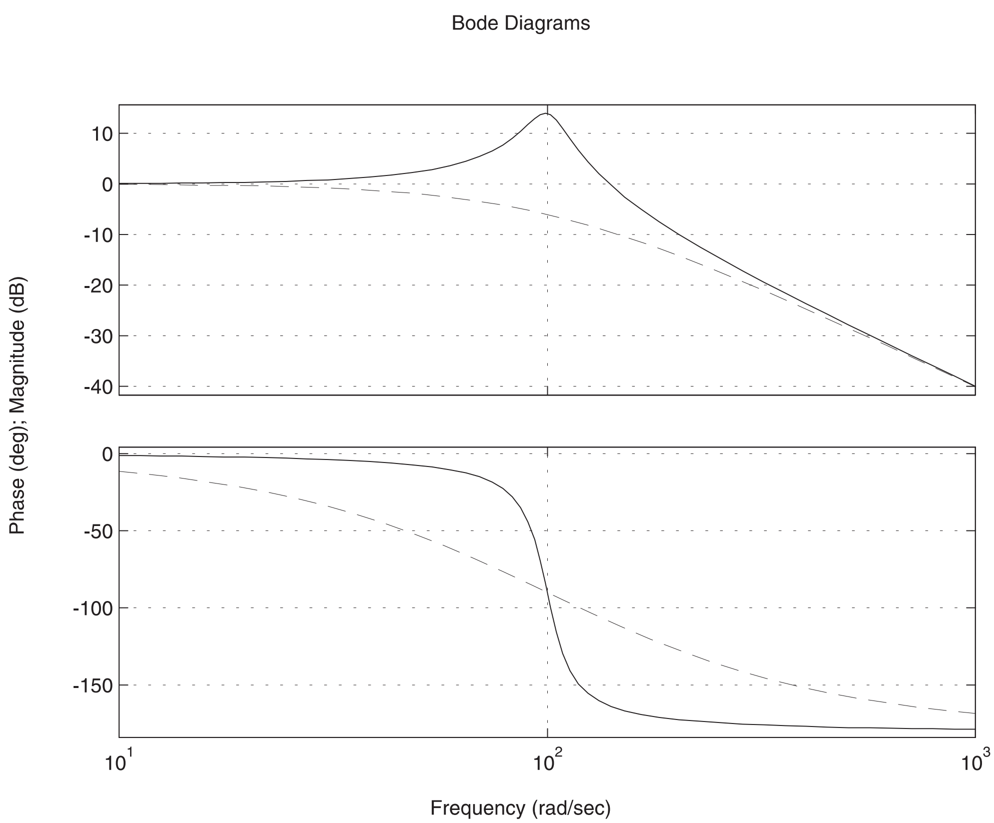

Figure 10: Typical Frequency Responses of a Second-Order, Low-Pass Filter

First-Order High-Pass Filter: A first-order, high-pass filter is given by the circuit shown in Figure 11. Its transfer function can be computed as

$$\frac{V_o(s)}{V_i(s)} = -\frac{sR_fC_i}{sR_fC_f + 1}. \tag{2.15}$$

It is interesting to compare (2.15) with the transfer function of the *passive*, high-pass filter given by Eq. (9) of Laboratory 2. Note that, whereas the passive filter provides no high frequency ($s \to \infty$) amplification, the active filter has a high frequency gain of $\frac{C_i}{C_f}$.

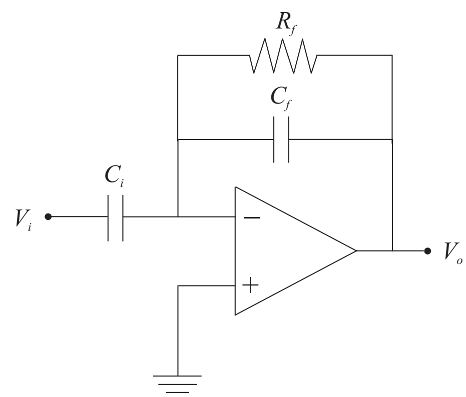

Figure 11: Active, First-Order, High-Pass Filter

Second-Order High-Pass Filter: A second-order, high-pass filter is shown in Figure 12.

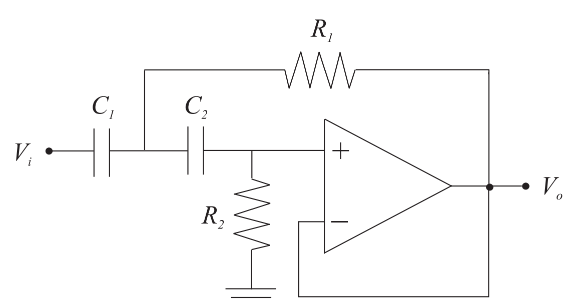

Figure 12: Active, Second-Order, High-Pass Filter

The transfer function for this filter is

$$\frac{V_o(s)}{V_i(s)} = \frac{\frac{1}{R_1R_2C_1C_2}s^2}{s^2 + \frac{C_1+C_2}{R_2C_1C_2}s + \frac{1}{R_1R_2C_1C_2}} = \frac{\omega_n^2 s^2}{s^2 + 2\zeta\omega_n s + \omega_n^2} \tag{2.16}$$

where ω_n is defined as in (2.14), but ζ is now given by

$$\zeta = \frac{C_1 + C_2}{2}\sqrt{\frac{R_1}{R_2C_1C_2}}. \tag{2.17}$$

Figure 13 shows the frequency response of (2.16) for two cases: $\omega_n = 100$ rad/sec and $\zeta = 0.1$ (solid line), and $\omega_n = 100$ rad/sec and $\zeta = 1$ (dashed line).

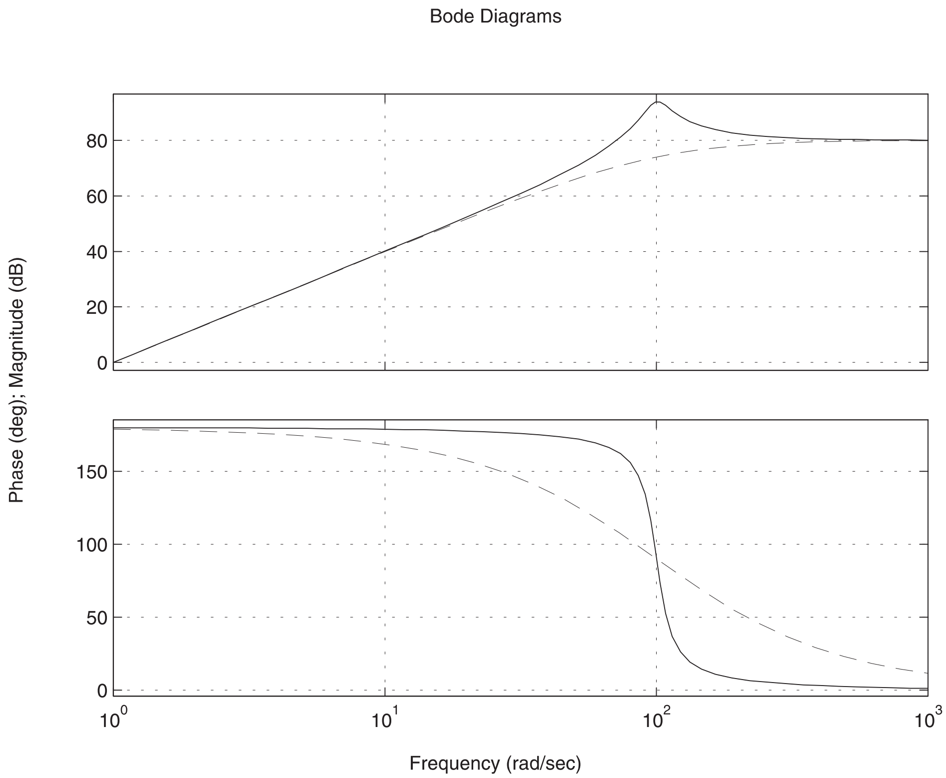

Figure 13: Typical Frequency Responses of a Second-Order, High-Pass Filter

Notch Filter: Notch (or band-stop) filters are extremely useful in measurement systems containing undesirable signals of fixed frequencies. In these cases, a notch filter can reduce the undesirable signal to a tolerance level. The op-amp implementation of this filter is shown in Figure 14, and its transfer function is given by

$$A_n(s) = \frac{V_o(s)}{V_i(s)} = \frac{s^2 + \omega_o^2}{s^2 + 2\omega_o s + \omega_o^2} \tag{2.18}$$

where the notch frequency ω_o is defined as

$$\omega_o = \frac{2}{RC}. \tag{2.19}$$

Figure 15 shows the frequency response of (2.18) for the case where the notch frequency is placed at 1 kHz (= 6283 rad/sec). Note that the magnitude frequency response has unity gain (0 dB) for all frequencies except at the notch frequency.

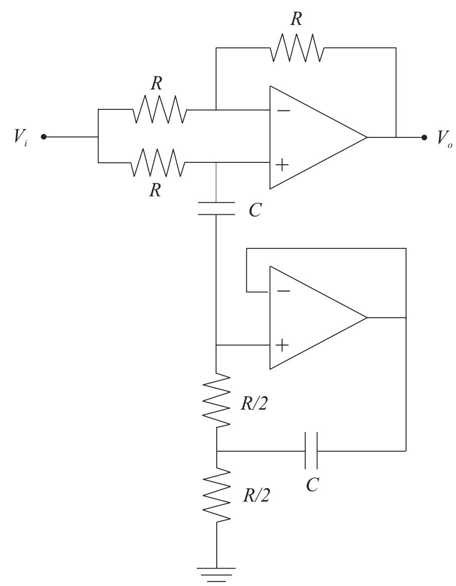

Figure 14: Active Notch Filter

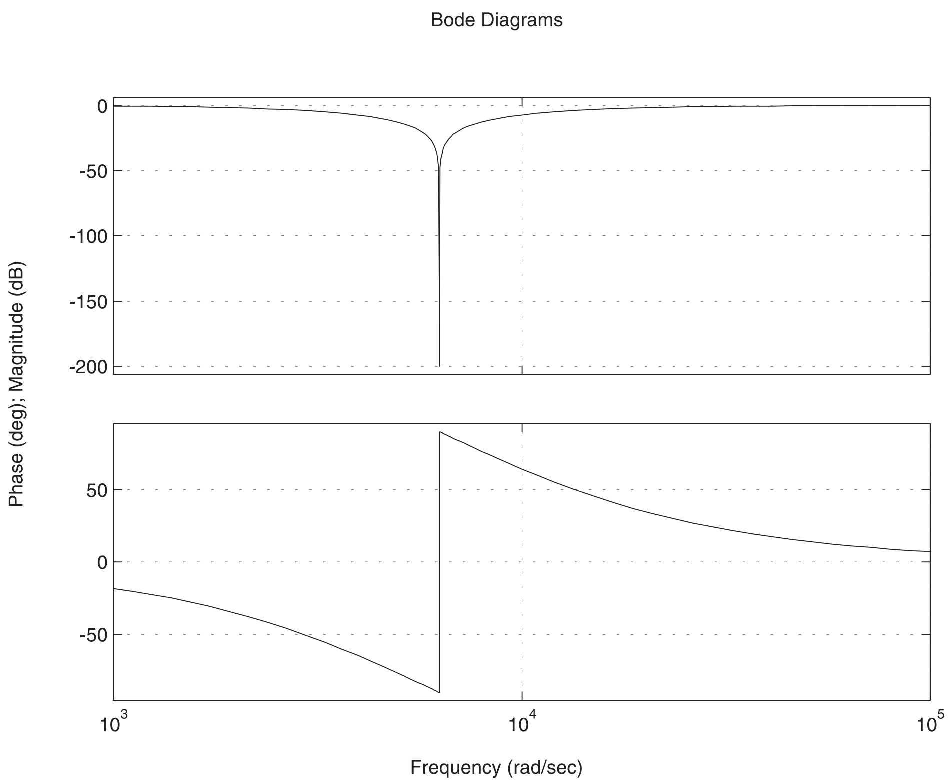

Figure 15: Typical Frequency Response of a Notch Filter

Band-Pass Filter: Band-pass filters are used when certain harmonics of a signal need to be emphasized while the rest be attenuated. Band-pass filters can be constructed by using a notch filter, inverting amplifier, and summing amplifier as shown in Figure 16. The filter's transfer function can be found as

$$\frac{V_o(s)}{V_i(s)} = 1 - A_n(s) \tag{2.20}$$

where $A_n(s)$ is the transfer function of the notch filter from (2.18).

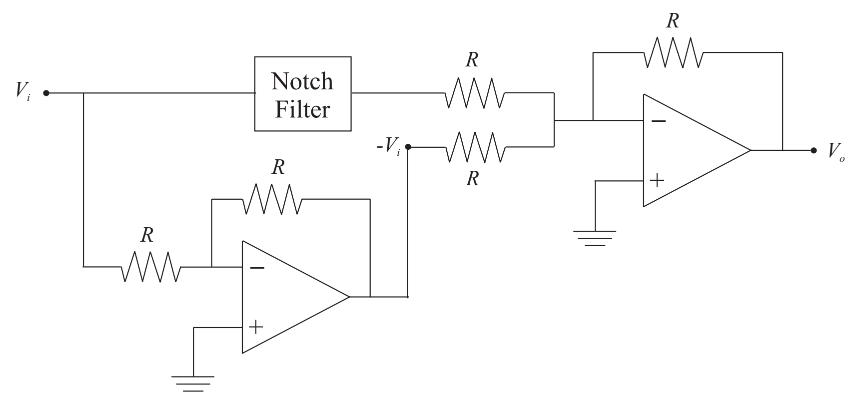

Figure 16: Active Band-Pass Filter

3. Equipment List

i) PC with Q4 DACB and terminal board

ii) Software environment: Windows, MATLAB, Simulink, RTW, and QuaRC

iii) One LM348 op-amps chip

iv) ±12 Volts, dual output power supply

v) Set of known resistors, capacitors, Breadboard, a set of leads, and connectors

4. Experimental Procedure

In this experiment, we will design amplifiers and active filters by using resistoers, capacitors, and LM348 op-amps.

4.1. Inverting Amplifier

i) Assemble the circuit of Figure 3 on the breadboard using resistors ($R_f = 2\ k\Omega$ and $R_i = 1\ k\Omega$) and a LM348 chip to build an inverting amplifier with a gain of approximately -2. Note that you can select $R_p = 0$. Since the LM348 operates with a dual power supply, make sure that you provide the appropriate wiring so that the $V^+ = +12$ Volts, GND, and $V^- = -12$ Volts outputs are available from your dual power supply. Using the Q4 terminal board and a double-ended RCA connector, connect the input of the circuit to the channel 0 of DAC (analog output) and the output to the channel 0 of ADC (analog input). **Before proceeding, you must request the laboratory TA to approve your electrical connections**.

ii) From the **Start** button of the Windows toolbar, select the option sequence **Programs–MATLAB–R2007b–MATLAB R2007b** to launch the MATLAB application.

iii) In the MATLAB window, choose "C:\MeasurementSystemsLab\Experiment3" from the Current Directory window. This directory path choice will change the directory from the default MATLAB directory to the working directory for Experiment 3.

iv) From the **File** menu of the MATLAB window, select the option **Open** to load the Simulink block-diagram "Experiment3A.mdl." to your desktop.

v) In the MATLAB window, apply a 1 Volt-amplitude, sinusoidal signal to V_i. click the black **Start** arrow button to acquire the real-time data for the experiment.

vi) In the Simulink block-diagram, the To File block creates "Exp3AData.mat" which the plot data is saved on. Plot the step response using MATLAB by executing the following commands: load Exp3AData and plot(data3A(1,:),data3A(2,:),data3A(1,:),data3A(3,:)).

4.2. Summing Amplifier

i) Assemble the circuit of Figure 5 on the breadboard using resistors ($R_f = 2\ k\Omega$, $R_1 = 2\ k\Omega$, and $R_2 = 2\ k\Omega$) and a LM348 chip to build a summing amplifier such that $V_o = -V_{i1} - V_{i2}$. Using the Q4 terminal board and a double-ended RCA connector, connect the input of the circuit to the channel 0 of DAC (analog output) and the output to the channel 0 of

ADC (analog input). **Before proceeding, you must request the laboratory TA to approve your electrical connections**.

ii) From the **File** menu of the MATLAB window, select the option **Open** to load the Simulink block-diagram "Experiment3B.mdl." to your desktop.

iii) In the MATLAB window, apply a 5 Volts DC–signal to V_{i1} and a 1 Volt-amplitude, sinusoidal signal to V_{i2}. click the black **Start** arrow button to acquire the real-time data for the experiment.

iv) In the Simulink block-diagram, the To File block creates "Exp3BData.mat" which the plot data is saved on. Plot the step response using MATLAB by executing the following commands: load Exp3BData and plot(data3B(1,:),data3B(2,:),data3B(1,:),data3B(3,:)).

4.3. Integrator Amplifier

i) Assemble the circuit of Figure 7 on the breadboard using a resistor ($R_i = 1\ M\Omega$), a capacitor ($C_f = 1\ \mu F$), and a LM348 chip to build an integrator amplifier such that $V_o \approx -\int_{t_0}^{t} V_i(\tau)d\tau$. Using the Q4 terminal board and a double-ended RCA connector, connect the input of the circuit to the channel 0 of DAC (analog output) and the output to the channel 0 of ADC (analog input). **Before proceeding, you must request the laboratory TA to approve your electrical connections**.

ii) From the **File** menu of the MATLAB window, select the option **Open** to load the Simulink block-diagram "Experiment3C.mdl." to your desktop.

iii) Apply a 1 Volt-amplitude, periodic square waveform and recording the output signal V_o. click the black **Start** arrow button to acquire the real-time data for the experiment.

iv) In the Simulink block-diagram, the To File block creates "Exp3CData.mat" which the plot data is saved on. Plot the step response using MATLAB by executing the following commands: load Exp3CData and plot(data3C(1,:),data3C(2,:),data3C(1,:),data3C(3,:)).

4.4. Active Second-Order, Low-Pass Filter

i) Assemble the circuit of Figure 9 on the breadboard using capacitors ($C_1 = 1\ \mu F$ and $C_2 = 1\ \mu F$), and resistors ($R_1 = 10\ k\Omega$ and $R_2 = 10\ k\Omega$) and a LM348 chip to implement the active filter. Using the Q4 terminal board and a double-ended RCA connector, connect the input of the circuit to the channel 0 of DAC (analog output) and the output to the channel 0 of ADC (analog input). **Before proceeding, you must request the laboratory TA to approve your electrical connections**.

ii) From the **File** menu of the MATLAB window, select the option **Open** to load the Simulink block-diagram "Experiment3D.mdl." to your desktop.

iii) Click the black **Start** arrow button to acquire the real-time data for the experiment. In the MATLAB window, apply various sinusoidal inputs (starting from 1 rad/sec to 2000 rad/sec) to the circuit by changing the value of the frequency in the Signal Generator block. Collect the magnitude of output signal with respect to the input signal frequency.

iv) After sufficient experimentation, press the black **Stop** square button in the MATLAB window to stop execution of your program on the Q4 DACB.

4.5. Active Notch Filter

i) Assemble the circuit of Figure 14 on the breadboard using capacitors ($C = 1\ \mu F$), and resistors ($R = 20\ k\Omega$) to implement the notch filter. Using the Q4 terminal board and a double-ended RCA connector, connect the input of the circuit to the channel 0 of DAC (analog output) and the output to the channel 0 of ADC (analog input). **Before proceeding, you must request the laboratory TA to approve your electrical connections**.

ii) From the **File** menu of the MATLAB window, select the option **Open** to load the Simulink block-diagram "Experiment3D.mdl." to your desktop.

iii) Click the black **Start** arrow button to acquire the real-time data for the experiment. In the MATLAB window, apply various sinusoidal inputs (starting from 1 rad/sec to 2000 rad/sec) to the circuit by changing the value of the frequency in the Signal Generator block. Collect the magnitude of output signal with respect to the input signal frequency.

iv) After sufficient experimentation, press the black **Stop** square button in the MATLAB window to stop execution of your program on the Q4 DACB.

5. Analysis/Assignment

i) In Section 4.1, derive the transfer function of the inverting amplifier you built. Simulate the transfer function using MATLAB/Simulink and compare the simulation with the experimental result.

ii) In Section 4.2, derive the transfer function of the summing amplifier you built. Simulate the transfer function using MATLAB/Simulink and compare the simulation with the experimental result.

iii) In Section 4.3, derive the transfer function of the integrator amplifier you built. Simulate the transfer function using MATLAB/Simulink and compare the simulation with the experimental result.

iv) In steps *i*)—*iv*) of Section 4.4, plot your experimental results in a semi-log paper with the magnitude in decibels and frequency in rad/sec. Derive the transfer function of the filter you built. Plot the theoretical frequency response using MATLAB and compare the simulation with the experimental result.

v) In steps *i*)—*iv*) of Section 4.5, plot your experimental results in a semi-log paper with the magnitude in decibels and frequency in rad/sec. Plot the theoretical frequency response of the transfer function given in (2.18) and (2.19) using MATLAB and compare the simulation with the experimental result.

References

1. W. Bolton, *Mechatronics: Electronic Control Systems in Mechanical and Electrical Engineering*, Addison Wesley, New York, NY, 1999.

2. M.B. Histand and D.G. Alciatore, *Introduction to Mechatronics and Measurement Systems*, WCB/McGraw-Hill, Boston, MA, 1999.

3. J.W. Nilsson, *Electric Circuits*, Addison Wesley, Reading, MA, 1996.

Experiment 4: Temperature Measurements

Concepts emphasized: Resistor temperature detector, thermistor, and thermocouple.

1. Objectives

i) Understand the main operating characteristics of temperature measurement devices such as resistor temperature detectors, thermistors, and thermocouples.

ii) Be able to experimentally determine temperature curves for temperature measurement devices.

2. Background

In many engineering systems temperature constitutes an important physical variable that needs to be monitored and controlled. For example, temperature sensors are present in buildings, chemical processing plants, engines, computers, vehicles, *etc.* Many physical phenomena (*e.g.*, pressure, volume, electrical resistance, expansion coefficients, *etc.*) can be related to temperature through the fundamental molecular structure. Temperature variations affect these quantities, and hence their changes can be used to indirectly measure temperature.

Temperature measurement devices can be classified as mechanically operative (mercury thermometer and bimetallic strip) or electrically operative (resistance temperature detector, thermistor, and thermocouple). In this laboratory, we will concentrate on electric-based temperature sensors due to their higher accuracy and ease in providing measurements for signal processing and computer acquisition. These sensors are based on the principle that electrical resistance or voltage of some materials changes in a reproducible manner with temperature. More detailed information about the subjects covered in this laboratory can be found in references [1, 2, 3].

2.1. Resistance Temperature Detector (RTD)

The variation of resistance R of an RTD with temperature T for most metallic materials can be expressed as

$$R(T) = R_0 \left(1 + \alpha_1 (T - T_0) + \alpha_2 (T - T_0)^2 + \ldots\right) \tag{2.1}$$

where T_0 is a reference temperature, R_0 is the resistance at the reference temperature, and the α_i's are some positive constant coefficients. The number of the terms included in (2.1) depends on the material and required accuracy. Typically, only α_1 is used since linearity can be achieved over a wide range of temperatures. Platinum, for example, is linear within ±0.4% over the range of −100° to +300° F. Typical resistance/temperature curves for some metals can be found in Figure 2.50 of reference [1].

2.2. Thermistor

Semiconductor resistance temperature sensors (thermistors) are more sensitive than RTDs. They have a very large negative coefficient, and a highly nonlinear characteristic. Their resistance/temperature relationship is

$$R = R_0 \exp\left(\beta\left(\frac{1}{\mathrm{T}} - \frac{1}{\mathrm{T}_0}\right)\right) \tag{2.2}$$

where T and T_0 are absolute temperatures in Kelvin, and R (R_0) is the resistance of the material at temperature T (T_0). The reference temperature T_0 is generally taken at 298° K. The constant coefficient β ranges from 3500° to 4600° K depending on the material, temperature, and individual construction for each sensor; therefore, it must be determined for each thermistor. Thermistors exhibit large resistance changes with temperature. A typical resistance/temperature curve for a thermistor appears in Figure 2.52 of reference [1].

2.3. Thermocouple

Thermocouples utilize the so-called *Seebeck* effect in order to transform a temperature difference to a voltage difference. A thermocouple consists of two electrical conductors that are made of dissimilar metallic materials and have at least one electrical connection. This electrical connection is referred to as a junction. A thermocouple junction may be created by welding, soldering, or by any method that provides good electrical contact between the two conductors, such as twisting the wires around one another. A typical thermocouple circuit with two junctions, T_0 reference and T measured, is shown in Figure 1. The output of a thermocouple circuit is a voltage which is related to the temperatures of the junctions that make up the circuit. For the circuit of Figure 1, the relationship for a wide range of temperatures is given by the following linear equation

$$V = \alpha_1 (T - T_0) + \alpha_2 (T - T_0)^2 + \dots \tag{2.3}$$

where the α_i's are some positive constant coefficients which are dependent on the thermocouple material. As was the case for the RTD, the number of the terms included in (2.3) depends on the material and required accuracy. Typically, only α_1 (often called the Seebeck coefficient) is used since linearity can be achieved over a certain temperature range.

Thermocouples are characterized according to the alloys that are used for their construction. The following four classes are the most popular ones: J–type (iron–constantan), K–type (chromel–alumel), E–type (chromel–constantan), and T–type (copper–constantan). Typical voltage/temperature curves for the above thermocouple types can be found in Figures 2.55 and 8.43 of references [1, 2], respectively.

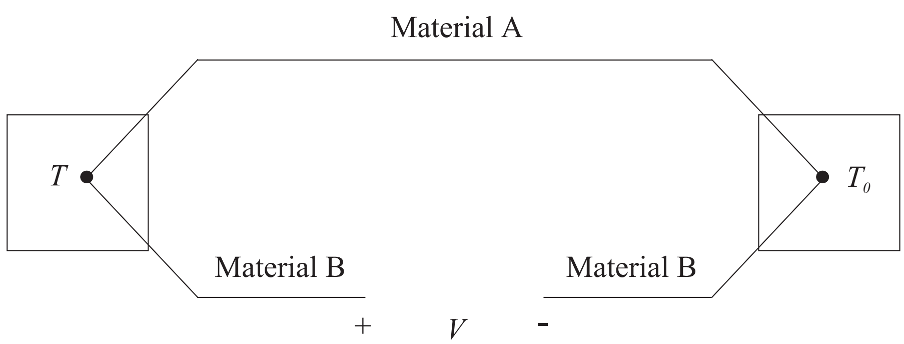

Figure 1: Basic Thermocouple Circuit

3. Laboratory Procedure

3.1. Equipment List

i) PC with Q4 data acquisition card and terminal board

ii) Software environment: Windows, MATLAB, Simulink, RTW, and QuaRC

iii) Thermal heater, Two water containers (one with ice water), and Mercury thermometer

iv) RTD, thermistor kit, and a pair of type-T thermocouples

v) Power supply, set of resistors, set of connectors, and a multimeter

4. Experimental Procedure

4.1. RTD Experiments

i) Using the set of leads, RTD, and the terminal board of the Q4 data acquisition card, complete the wiring diagram shown in Figure 2. **Before proceeding, you must request the laboratory TA to approve your electrical connections**. Fill the flask (water container) with less than 300 ml of water to prevent overflowing when water is boiling. **The heater is well able to boil the water at about its first incremental setting. Therefore, when adjusting the heater's setting, approach this point gradually and do not exceed it.** Note the existence of a series circuit comprised of the voltage supply (set to 5 Volts), a resistor $R = 100\ \Omega$, and the RTD.

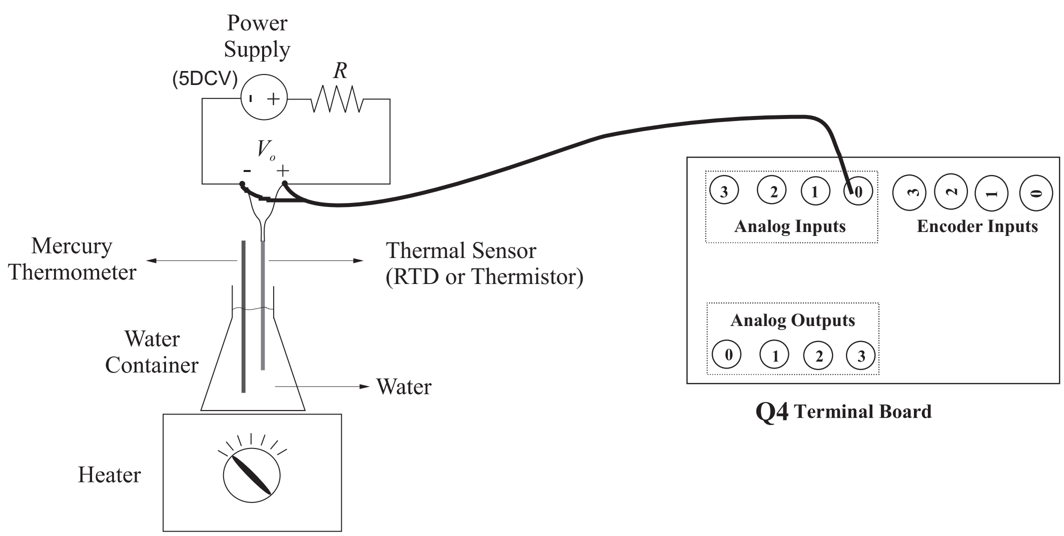

Figure 2: Temperature Measurement Testbed Setup for RTD and Thermistor

ii) Start MATLAB. Choose "C:\MeasurementSystemsLab\Experiment4" from the Current Directory window of the MATLAB. This directory path choice will change the directory from the default MATLAB directory to the directory where all files needed to perform Experiment 4 are stored.

iii) From the **File** menu of the MATLAB window, select the option **Open** to load the Simulink block-diagram "Experiment4A.mdl." shown in Figure 3 to your desktop. This will load the files for determining the voltage reading from the RTD.

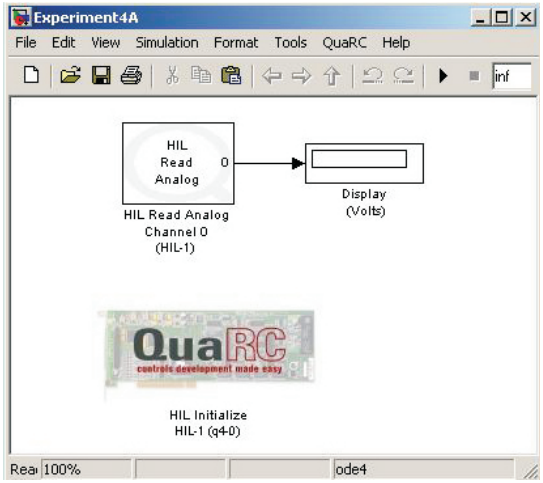

Figure 3: Simulink Block-Diagram for the Voltage Reading from the RTD

a) In the MATLAB window, click the black **Start** arrow button to acquire the sensor voltage.

b) Turn on the water heater and record the sensor voltage every 10 degree temperature increment (up to 80 degrees) by reading it from a mercury thermometer. Then turn off the heater. **Water is boiling. So during the experiment, you must pay extreme attention not to make any accident**.

c) In the MATLAB window, click the black **Stop** square button when you finish collecting the sensor voltage.

iv) From the **File** menu of the MATLAB window, select the option **Open** to load the Simulink block-diagram "Experiment4B.mdl." shown in Figure 4 to your desktop. This will load the files for determining the time constant of the RTD. The goal of this experiment is to estimate the sensor's time constant. The time constant will be defined as the time the sensor takes to reach 63.2% of the final value of its output voltage, when subjected to a step change of temperature. So one student gets ready for dipping the RTD from the cold water flask to the hot water flask. And another student gets ready for operating MATLAB.

a) In the MATLAB window, click the black **Start** arrow button to acquire the sensor voltage as soon as the RTD is dipped into hot water.

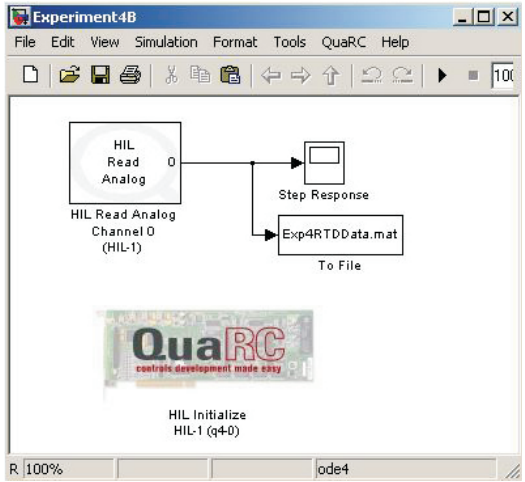

Figure 4: Simulink Block-Diagram for the Time Constant of the RTD

b) In the Simulink block-diagram, the To File block creates "Exp4RTDData.mat" which the plot data is saved on. Plot the step response from the MATLAB window by executing the following commands: load Exp4RTDData and plot(data4RTD(1,:),data4RTD(2,:)).

4.2. Thermistor Experiments

i) Repeat the steps *i*)—*iii*) of the above RTD experiments with the thermistor replacing the RTD. Note the existence of a series circuit comprised of the voltage supply (set to 5 Volts), a resistor $R = 15\ K\Omega$, and the thermistor.

ii) From the **File** menu of the MATLAB window, select the option **Open** to load the Simulink block-diagram "Experiment4C.mdl." shown in Figure 5 to your desktop. This will load the files for determining the time constant of the thermistor. The goal of this experiment is to estimate the sensor's time constant. So one student gets ready for dipping the thermistor from the hot water flask to the cold water flask. And another studednt gets ready for operating MATLAB.

a) In the MATLAB window, click the black **Start** arrow button to acquire the sensor voltage as soon as the RTD is dipped into hot water.

b) In the Simulink block-diagram, the To File block creates "Exp4ThermistorData.mat" which the plot data is saved on. Plot the step response from the MATLAB window by exe-

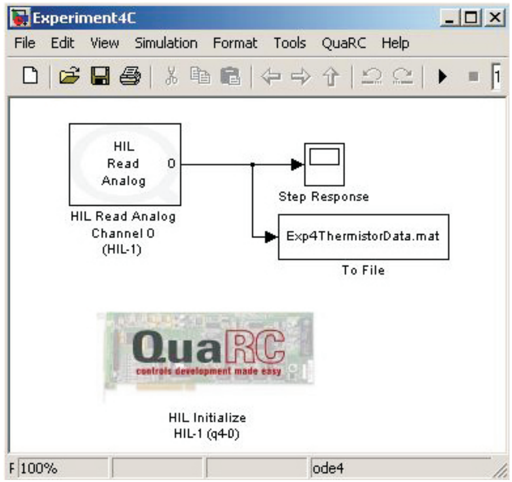

Figure 5: Simulink Block-Diagram for the Time Constant of the Thermistor

cuting the following commands: load Exp4ThermistorData and plot(data4Thermistor(1,:), data4Thermistor(2,:)).

4.3. Thermocouple Experiment

i) Setup the thermocouple circuit shown in Figure 6, and connect the circuit terminals to a voltmeter. The ice water container will provide a reference temperature for the thermocouple measurements. **Before proceeding, you must request the laboratory TA to approve your setup**.

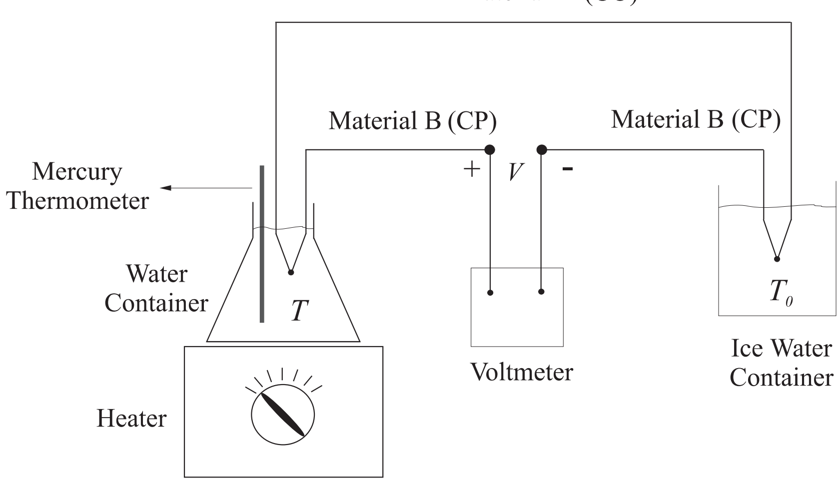

Figure 6: Experimental Setup for Thermocouple Measurements

ii) Turn on the water heater and record the sensor voltage every 10 degree temperature increament (up to 80 degrees) by a mercury thermometer. Then turn off the heater. **Water is boiling. So during the experiment, you must pay exteme attention not to make any accident**.

5. Analysis/Assignment

i) In step *iii*) of Section 4.1, based on the voltage collected at each 10 degree temperature increment, plot a voltage/temperature curve. By using voltage division, calculate the resistance of the RTD with repect to each temperautre increment and plot a resistance/temperature curve. Comment on the linearity of the curve you obtained.

ii) Based on the resistance/temperature curve obtained above, determine the RTD's positive constant coefficient α_1 from (2.1). Here we consider only the first order term of the equation, which is $R(T) = R_0(1 + \alpha_1(T - T_0))$.

iii) In Section 4.1, the resistance/temperature curve was indirectly obtained through voltage measurements. How can you *directly* measure the RTD resistance without the need for the above circuit?

iv) In step *iv*) of Section 4.1, calculate the time constant of the RTD.

v) In step *i*) of Section 4.2, based on the voltage collected at each 10 degree temperature increment, plot a voltage/temperature curve. By using voltage division, calculate the resistance of the thermistor with repect to each temperautre increment and plot a resistance/temperature curve. Comment on the linearity of the curve you obtained.

vi) Based on the resistance/temperature curve obtained above, determine the thermistor's positive constant coefficient β from (2.2).

vii) In step *ii*) of Section 4.2, calculate the time constant of the thermistor.

viii) In Section 4.3, experimentally determine the voltage/temperature curve for the thermocouple given by (2.3). Comment on the linearity of the curve you obtained.

ix) Based on the voltage/temperature curve obtained above, determine the thermocouple's positive constant coefficient α_1 from (2.3). Here we consider only the first order term of the equation, which is $V = \alpha_1(T - T_0)$.

x) Compare all three temperature sensors based on their experimental results and comment on the differences (e.g., linearity, time constant, characteristics, etc.).

References

1. W. Bolton, *Mechatronics: Electronic Control Systems in Mechanical and Electrical Engineering*, Addison Wesley, New York, NY, 1999.

2. M.B. Histand and D.G. Alciatore, *Introduction to Mechatronics and Measurement Systems*, WCB/McGraw-Hill, Boston, MA, 1999.

3. J.P. Holman, *Experimental Methods for Engineers*, McGraw-Hill, New York, NY, 1994.

Experiment 5: Position and Velocity Measurements

Concepts emphasized: Potentiometer, encoder, and tachometer.

1. Objectives

i) Understand the operation of motion-type sensors such as a potentiometer, encoder, and tachometer.

ii) Be able to calibrate motion sensors.

iii) Be able to perform data acquisition on motion sensors.

2. Background

In many mechatronic systems, we often need to know how various parts of the system are moving in order to control its behavior (*e.g.*, industrial robot arm, numerically-controlled lathe and mill axes, rotor, *etc.*). Thus, the most common measured physical quantities in mechatronic systems are position and velocity. Whether a mechatronic system undergoes rotary or linear motions, rotary motion sensors are more common than linear sensors since linear motion can often easily be converted to rotary motion. In the following, the basic principles of three of the most common motion sensors − potentiometer and encoder for position measurements, and tachometer for velocity measurements − are briefly described. More detailed information about the subjects covered in this laboratory can be found in references [1, 2, 3, 4].

2.1. Potentiometer

In Laboratory 1, we learned that a circular potentiometer is a resistor whose resistance varies with the movement of a rotating shaft. Mathematically, this fact is expressed by the following equation

$$R_p(\theta) = k_p\theta \tag{2.1}$$

where R_p is the potentiometer's variable output resistance, θ is the angular position of the shaft in degrees (or radians), and k_p is the proportionality constant in $\frac{\Omega}{\deg}$ (or $\frac{\Omega}{rad}$). shows a typical potentiometer circuit from which, as a result of the voltage division rule and (2.1), we have

$$V_o = \frac{R_{23}}{R_{13}}V = \frac{R_p}{R_{13}}V = \frac{k_p\theta}{R_{13}}V = \bar{k}_p\theta, \bar{k}_p \triangleq \frac{k_p}{R_{13}}V \tag{2.2}$$

where R_{23} is the resistance between terminals 2 and 3 (*i.e.*, the potentiometer's output resistance R_p), R_{13} is the resistance between terminals 1 and 3 (*i.e.*, the potentiometer's constant, total resistance), and $\bar{k}_p$ is a new proportionality constant in $\frac{Volts}{deg}$ (or $\frac{Volts}{rad}$). From (2.2) it is clear that, once the constant $\bar{k}_p$ is known, angular position can be calculated by measuring the potentiometer's output voltage V_o.

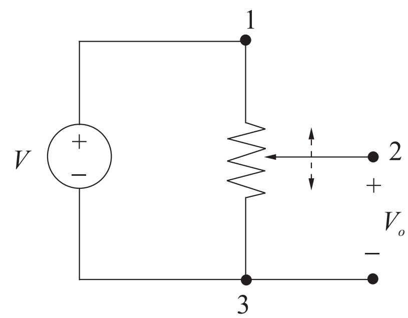

Figure 1: Potentiometer Circuit

2.2. Optical Encoder

A optical, rotary encoder is a device that converts angular position into a sequence of digital pulses. By decoding a set of bits, the pulses can be converted to relative or absolute position measurements. Encoders are manufactured into two forms: the *absolute* encoder, where a unique digital word corresponds to an absolute angular position, and the *incremental* encoder, which produces digital pulses as a shaft rotates, allowing measurement of relative angular position only.

Optical encoders are basically a circular disk containing a set of radial lines organized in tracks along with photoemitter-detector pairs. As the disk rotates, the light beam between the photoemitter-detector pair is interrupted by the radial lines, thus, producing digital pulses (see Figure 8.10 of [4]). This pulse train is then decoded and associated to an absolute or relative angular position. The disk of an absolute encoder is built to produce a digital word that distinguishes N different positions of the shaft. For example, if there are 8 tracks, the encoder will give $2^8 = 256$ distinct angular positions with a resolution of $1.406° = \frac{360°}{256}$. The incremental encoder is simpler in design than the absolute encoder. It consists of only two tracks and two photoemitter-detector pairs. As the disk rotates, pulse trains occur on the two channels at a frequency proportional to the disk speed. The pulse trains of the channels are out of phase with each other so that the direction of rotation can be determined by assessing which channel leads the other.

2.3. Tachometer

A tachometer is an electric generator used to measure angular velocity. A generator refers to a device which converts mechanical energy into electrical energy. As such, a tachometer generator is a machine which, when driven by a mechanical torque, produces an electric output voltage proportional to the speed of rotation, *i.e.*,

$$V_o = k_t\dot{\theta} \tag{2.3}$$

where k_t is the tachometer's proportionality constant in $\frac{Volts.sec}{deg}$ (or $\frac{Volts.sec}{rad}$) and $\dot{\theta}$ is the angular velocity in $\frac{deg}{sec}$ (or $\frac{rad}{sec}$). This measurement is accomplished by connecting the tachometer to the rotating shaft whose angular velocity is to be determined.

Tachometers are generally referred to as AC or DC tachometers. The DC tachometer is essentially a small DC generator which produces a moderate DC output voltage. It differs from a DC generator in that certain parts of the design are optimized to give greater accuracy as a velocity measuring instrument rather than the generation of electricity. For example, it is important that tachometers be as light as possible so their mass does not affect the velocity of the system being measured. Thus, they do not always need to be as robust as generators, and may incorporate lighter materials, such as fiberglass to reduce their total mass. DC tachometers usually have a high noise content on their output voltage; hence, requiring signal conditioning to remove the electrical noise.

3. Laboratory Procedure

Note: In this laboratory, as opposed to Labs 2 to 4, the computer-based data acquisition will not be done using the LabView environment. Instead, the *MATLAB* environment will be utilized since it already provides the data acquisition interface to the experimental hardware used in this laboratory.

3.1. Equipment List

i) PC with Q4 data acquisition card and terminal board

ii) Software environment: Windows, MATLAB, Simulink, RTW, and QuaRC

iii) SRV-02 DC-motor apparatus (See Figure 2) with potentiometer, optical encoder, and tachometer

iv) Universal power module: UPM-1503

v) Set of leads

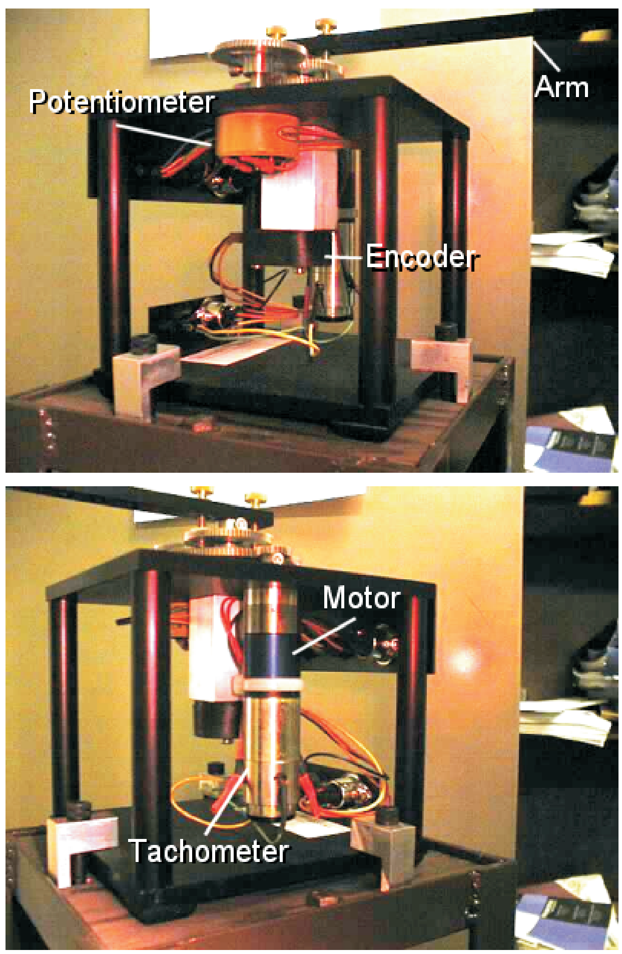

Figure 2: Picture of the Quanser SRV-02 Experimental Setup

4. Experimental Procedure

4.1. Potentiometer Experiment

i) Using the set of leads, universal power module, SRV-02 DC-motor apparatus, and the terminal board of the Q4 data acquisition card, complete the wiring diagram shown in Figure 3.

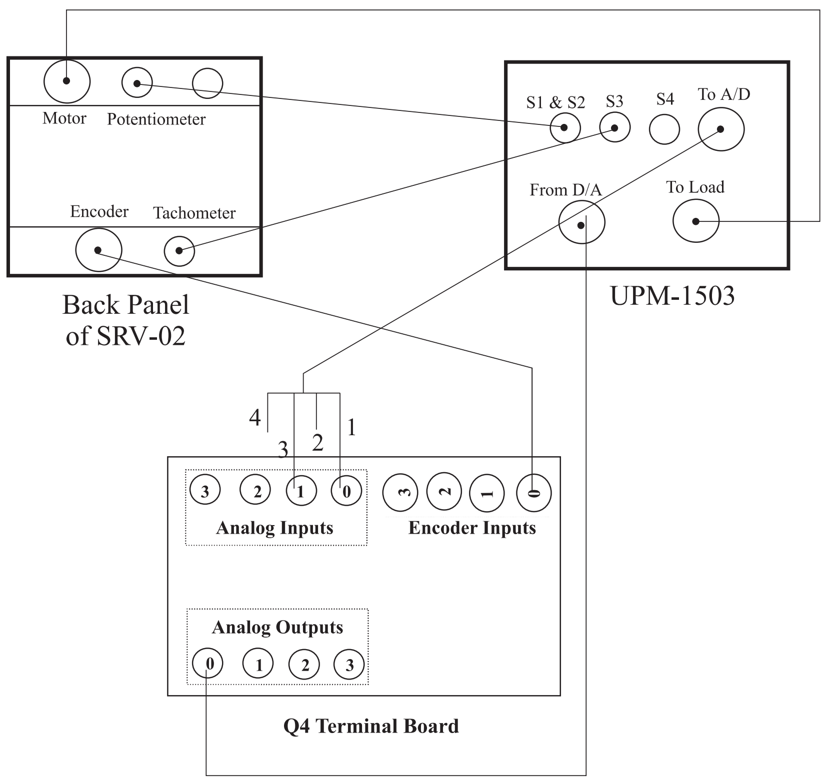

Figure 3: Connections Between Sensors and Data Acquisition Board

ii) Start MATLAB. In the MATLAB window, choose "C:\MeasurementSystemsLab\Experiment5" from the Current Directory window. This directory path choice will change the directory from the default MATLAB directory to the directory where all files needed to perform Experiment 5 are stored.

iii) You can now perform various steps of potentiometer, encoder, and tachometer experiment. However, before proceeding, you **must** request your laboratory teaching assistant to check your electrical connections.

iv) From the **File** menu of the MATLAB window, select the option **Open** to load the Simulink block-diagram "Experiment5_Pot.mdl." shown in Figure 5 to your desktop. This will load the files for determining the gain of the potentiometer K_{pot} $(\frac{\text{radian}}{\text{Volt}})$. The potentiometer gain K_{pot} relates the potentiometer output voltage V_{pot} to the load angular displacement θ_ℓ by $\theta_\ell = K_{\text{pot}} V_{\text{pot}}$.

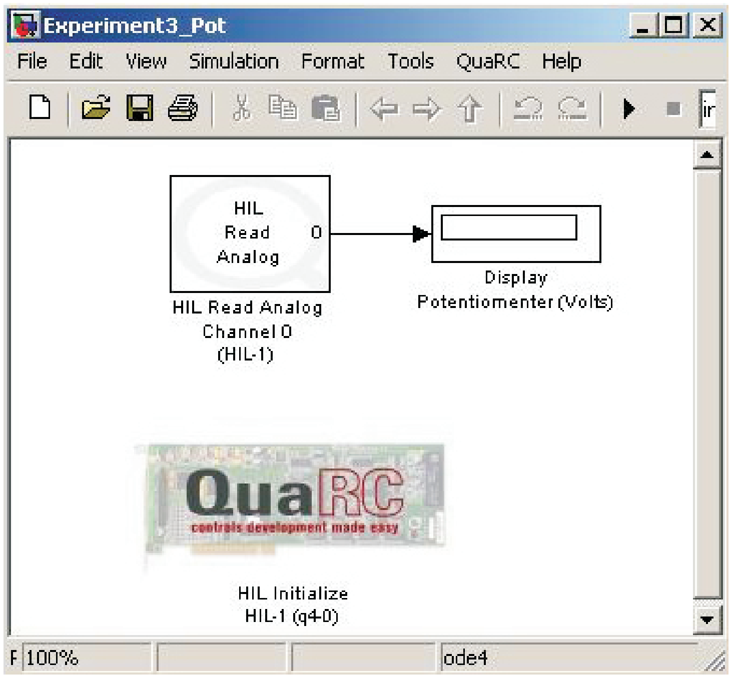

Figure 4: Simulink Block-Diagram for Determining Potentiometer Gain

a) In the MATLAB window, click the black **Start** arrow button to acquire the potentiometer voltage response.

b) Rotate the load connected to the output shaft (center gear) until the potentiometer voltage in the Display block shows 0 Volt. Please ensure that you get continuous variation in the neighborhood of this 0 Volt reading. If you note a discontinuity in the reading, turn the load by 180° and this will provide you close to 0 Volt reading. Read the angular position θ_0° of the load, corresponding to the 0 Volt potentiometer reading, off the protractor marked on the SRV-02 apparatus.

c) Rotate the load to $\theta_0 + 90°$ and note the corresponding potentiometer voltage reading in the Display block.

d) Rotate the load to $\theta_0 - 90°$ and note the corresponding potentiometer voltage reading in the Display block.

e) In the MATLAB window, click the black **Stop** square button when you finish collecting the potentiometer voltage response data.

4.2. Encoder Experiment

i) From the **File** menu of the MATLAB window, select the option **Open** to load the Simulink block-diagram "Experiment5_enc.mdl." shown in Figure 5 to your desktop. This will load the files for determining the gain of the encoder K_{enc} $(\frac{\text{radian}}{\text{count}})$. The encoder gain K_{enc} relates the encoder counter output N to the load angular displacement θ_ℓ by $\theta_\ell = K_{\text{enc}} N$.

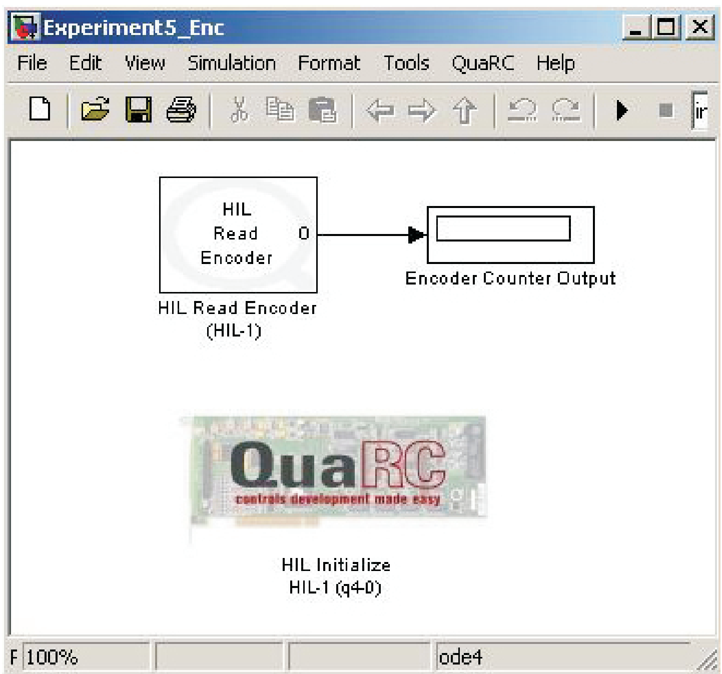

Figure 5: Simulink Block-Diagram for Determining Encoder Gain

a) In the MATLAB window, click the black **Start** arrow button to acquire the encoder count response.

b) Rotate the load to $\theta_0 + 90°$ and note the corresponding encoder counter output in the Display block.

c) Rotate the load to $\theta_0 - 90°$ and note the corresponding encoder counter output in the Display block.

d) In the MATLAB window, click the black **Stop** square button when you finish collecting the encoder count response data.

4.3. Tachometer Experiment

i) Close the currently open the Simulink diagram. From the **File** menu of the MATLAB window, select the option **Open** to load the Simulink block-diagram "Experiment5_Tach.mdl." shown in Figure 6 to your desktop. This will load the files for determining the gain of the tachometer K_{tach} $(\frac{\text{radian}}{\text{second-Volt}})$ and a plot window. The tachometer gain K_{tach} relates the tachometer output voltage V_{tach} to the load angular velocity ω_ℓ by $\omega_\ell = K_{\text{tach}} V_{\text{tach}}$.

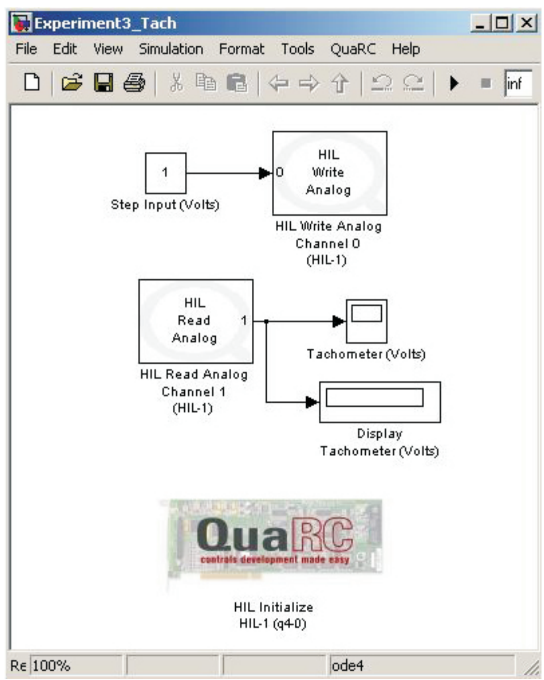

Figure 6: Simulink Block-Diagram for Determining Tachometer Gain

a) In the MATLAB window, click the black **Start** arrow button. This applies a constant 1 Volt input to the DC-motor.

b) Measure the steady-state load angular speed and the corresponding steady-state tachometer output voltage reading in the plot window. **Hint:** Find the time required for 20 complete revolutions of the load and the corresponding steady-state tachometer output voltage reading at the end of 20 revolutions.

c) In the MATLAB window, click the black **Stop** square button when you finish collecting the tachometer voltage response data.

5. Analysis/Assignment

i) In Section 4.1, the goal of this experiment is to calibrate the potentiometer. Devise a way of experimentally determining the voltage/angle relationship of the potentiometer given by (2.2). From the data you collected, plot the relationship and comment on the linearity of your results. What is the proportionality constant $\bar{k}_p$ of the potentiometer?

ii) In Section 4.2, identify if the encoder is an absolute or incremental one. Explain why you came to that conclusion. If the sensor is an incremental encoder, how would you establish a home or reference position for the arm; that is, how could you use the incremental encoder to measure an absolute angle? What is the proportionality constant $\bar{k}_{enc}$ of the encoder?

iii) In Section 4.3, the goal of this experiment is to calibrate the tachometer. Devise a way of experimentally determining the voltage/velocity relationship of the tachometer given by (2.3). From the data you collected, plot the relationship and comment on the linearity of your results. What is the proportionality constant k_t of the tachometer?

Extra Credit: Assume the tachometer was not available[1] and that you still needed to know the angular velocity of the arm. How would you obtain velocity information from the encoder or potentiometer? To obtain *full* credit, implement your velocity-generating method and compare your new velocity signal with the tachometer results. Comment on any differences. **The laboratory TA is responsible for providing any necessary additional components, assisting you in any interfacing with the Q4 board, and approving your final electrical connections. Special attention should be given not to exceed the maximum input voltage of the Q4 board.**

[1] Comercially available mechatronic systems, such as robot manipulators and machine tools, are normally not equipped with tachometers.

References

1. R.N. Bateson, *Introduction to Control System Technology*, Prentice-Hall, Upper Saddle River, NJ, 1999.

2. W. Bolton, *Mechatronics: Electronic Control Systems in Mechanical and Electrical Engineering*, Addison-Wesley, New York, NY, 1999.

3. P. Elgar, *Sensors for Measurement and Control*, Addison-Wesley Longman, Essex, England, 1998.

4. M.B. Histand and D.G. Alciatore, *Introduction to Mechatronics and Measurement Systems*, WCB/McGraw-Hill, Boston, MA, 1999.

Experiment 6: Acceleration Measurements

Concepts emphasized: Dynamic modeling, accelerometers.

1. Objectives

i) Understand the operation of piezoelectric-type accelerometers.

ii) Use accelerometers to identify the time responses of flexible beams.

2. Background

In the following, we will describe the main operating characteristics of accelerometers, with an emphasis on piezoelectric-type ones. We will also briefly present the dynamic characteristics of flexible beams. More detailed information about the subjects covered in this laboratory can be found in references [1, 2, 3, 4].

2.1. Piezoelectric Accelerometers

Accelerometers are mainly used for two specific types of acceleration measurement − impact (shock) and vibration. Impact is effectively a large acceleration over a short period of time, while vibration is a small, repeatable acceleration. Accelerometers are also used to measure the position, velocity, or acceleration of bodies, such as aircraft and ships. Accelerometers are normally mechanically bonded to the object whose acceleration is to be measured. The accelerometer detects acceleration along one axis and is insensitive in orthogonal directions.

The operating principle of an accelerometer is based on the inertial effects associated with a mass connected to a moving object through a spring and damper. When the moving object accelerates, there is a relative displacement between the object and the mass. The relative displacement is either directly measured through a position sensor, such as a linear potentiometer (seismic accelerometer), or indirectly sensed by the output voltage of a piezoelectric crystal (piezoelectric accelerometer). See Figure 1 for a schematic illustration of the two types of accelerometers. Whereas seismic accelerometers are intended mainly for low-frequency accelerations, piezoelectric accelerometers do not give an output for constant or slow-varying accelerations due to the basic characteristics of piezoelectric motion transducers. Instead, they are excellent for dynamic measurements, such as in high-frequency vibrations and impacts. In this laboratory, we will concentrate our studies on piezoelectric accelerometers.'

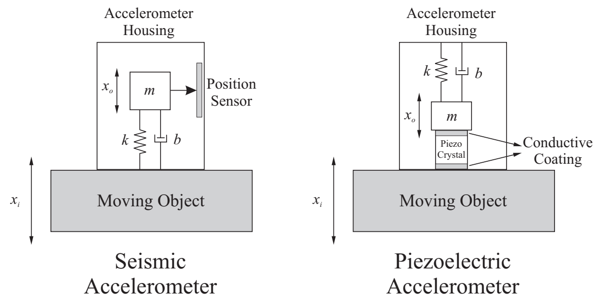

Figure 1: Schematic Representation of Accelerometers

As shown in Figure 1, piezoelectric accelerometers have a piezoelectric crystal in contact with a mass which is supported in a house by a spring. In addition to the natural damping inherent in the crystal and spring, additional damping is sometimes incorporated by filling the housing with oil. When the object accelerates, the mass displaces relative to the object, causing a deformation in the crystal. This deformation in the piezoelectric crystal causes in turn a electric charge between its conductive coatings as a result of the *piezoelectric effect.*[1] This electric charge is proportional to the mechanical deformation, *i.e.*,

$$q = k_q x_o \tag{2.1}$$

where q is the electric charge in Coulombs (C), x_o is the relative displacement between the mass and housing, and k_q is the proportionality constant in $\frac{C}{m}$.

The equivalent electric circuit for a piezoelectric crystal is a current source in parallel with a capacitor and in parallel with a resistor as shown in Figure 2. From this circuit, we can calculate the Laplace transform of the output voltage as

$$V_o(s) = \frac{R_e}{R_e C_e s + 1} I(s) = \frac{R_e s}{R_e C_e s + 1} Q(s) \tag{2.2}$$

where we have used the fact that current is the time derivative of charge. Substituting (2.1) into (2.2), we can obtain the transfer function between the voltage and the displacement x_o as

$$\frac{V_o(s)}{X_o(s)} = \frac{ks}{\tau s + 1} \tag{2.3}$$

[1] The piezoelectric effect describes the phenomenon where electric charges of opposite polarity appear on the faces of certain types of crystal when subjected to a mechanical deformation.

where $k = R_e k_q$ and $\tau = R_e C_e$ is the piezoelectric's time constant. We now need to find the transfer function between x_o and the object's acceleration $\ddot{x}_i$. This is done via the application of Newton's second law, which gives

$$\frac{X_o(s)}{\ddot{X}_i(s)} = \frac{1}{s^2 + 2\zeta\omega_n s + \omega_n^2} \tag{2.4}$$

where the natural frequency $\omega_n = \sqrt{\frac{k}{m}}$ and the damping ratio $\zeta = \frac{b}{2\sqrt{km}}$. Finally, the combination of (2.3) and (2.4) gives the relationship between the sensor's output voltage and acceleration as

$$\frac{V_o(s)}{\ddot{X}_i(s)} = \frac{ks}{(\tau s + 1)\,(s^2 + 2\zeta\omega_n s + \omega_n^2)}. \tag{2.5}$$

A typical frequency response of a piezoelectric accelerometer based on (2.5) is shown in Figure 8.51 of [3].

The low frequency response of (2.5) is limited by the piezoelectric characteristic $\frac{ks}{\tau s+1}$ while the high frequency response is limited by the mechanical resonance. The damping ratio ζ of piezoelectric accelerometers is not usually quoted by the manufacturer, but can be taken as zero for most practical purposes. The accurate frequency range of such an accelerometer is $\frac{3}{\tau} < \omega < 0.2\omega_n$. Accurate low frequency response requires large τ, which is usually achieved by the use of high-impedance voltage amplifiers or charge amplifiers.

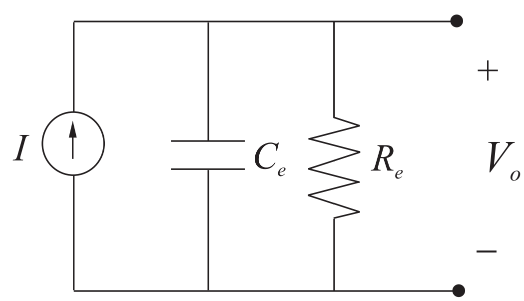

Figure 2: Equivalent Electric Circuit of a Piezoelectric Crystal

2.2. Flexible Beams

Consider the flexible, uniform beam shown in Figure 3 where the left end is clamped. Let $u(x,t)$ denote the beam displacement at the position x for time t and L be the length of the beam. Using the Euler-Bernoulli beam theory, the dynamics of the beam shown in Figure 3 is described by a partial differential equation (PDE) of the form

$$\rho A u_{tt}(x,t) + EIu_{xxxx}(x,t) = 0 \qquad x \in [0, L] \tag{2.6}$$

where ρ is the mass density of the beam, A is the beam cross-sectional area, E is Young's modulus, I is the moment of inertia of the beam cross section, and the subscripts x, t denote the partial derivatives with respect to x, t, respectively. Since (2.6) involves a second-order derivative with respect to time, and a fourth-order derivative with respect to x, two initial conditions and four boundary conditions are required to uniquely determine a solution $u(x,t)$ for the above PDE. Depending if the right end of the beam in Figure 3 is free or pinned, the following boundary conditions will hold

$$\begin{aligned} &\text{Clamped} - \text{freebeam} \quad && u(0,t) = u_x(0,t) = u_{xx}(L,t) = u_{xxx}(L,t) = 0 \\ &\text{Clamped} - \text{pinnedbeam} \quad && u(0,t) = u_x(0,t) = u(L,t) = u_{xx}(L,t) = 0. \end{aligned} \tag{2.7}$$

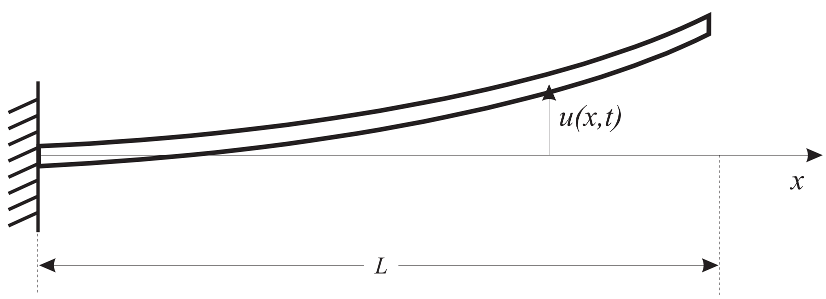

Figure 3: Flexible Beam

An ordinary differential equation approximation for the PDE of (2.6) can be made using the assumed modes method to facilitate the derivation of a solution in the form of a finite series of periodic functions. As a result, the beam's acceleration can be written as

$$u_{tt}(x,t) = \sum_{i=0}^{N} A_i \psi_i(x) \mathrm{e}\left(-\zeta_i \omega_{ni} t\right) \sin\left(\omega_{ni}\sqrt{1-\zeta_i^2}\, t + \phi_i\right), \tag{2.8}$$

where the amplitudes A_i depend on the initial beam deformation, $\psi_i(x)$ represents the beam's i-th mode shape, N is the total number of flexible modes included in the series approximation, ζ_i is the i-th damping ratio which depends on the beam's material and the medium (*e.g.*, air) properties, ω_{ni} denotes the i-th natural damping, and ϕ_i is the phase shift. The natural frequencies are computed using

$$\omega_{ni} = \beta_i \sqrt{\frac{EI}{mL^3}}, \tag{2.9}$$

where m is mass of a beam and the value of β_i is determined from the beam's boundary conditions. For example, the values of β_i, $i = 1, 2, 3$ (first three flexible modes) for a clamped-free beam are

$$\text{Clamped} - \text{free} \quad \text{beam} \quad \beta_1 = 3.5156 \quad \beta_2 = 22.0336 \quad \beta_3 = 61.7010. \tag{2.10}$$

Figures 4—6 show that deformation of a beam with the frist three natural frequencies using ANSYS.

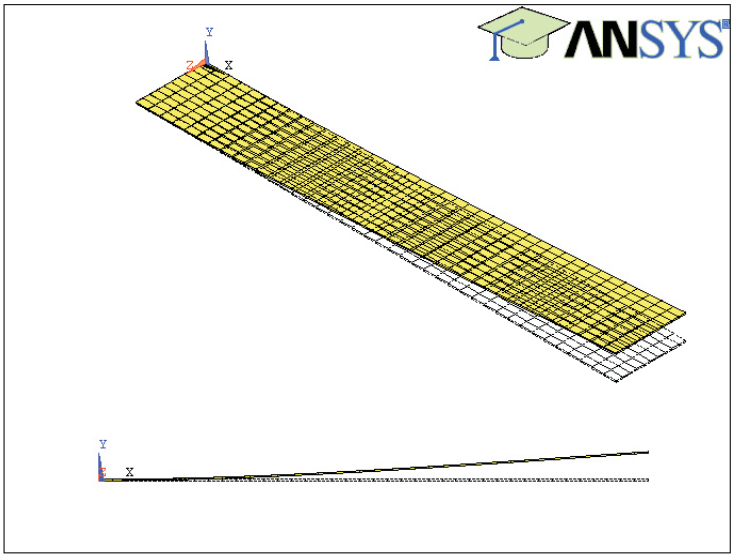

Figure 4: Deformed Beam Shape at the First Natural Frequency

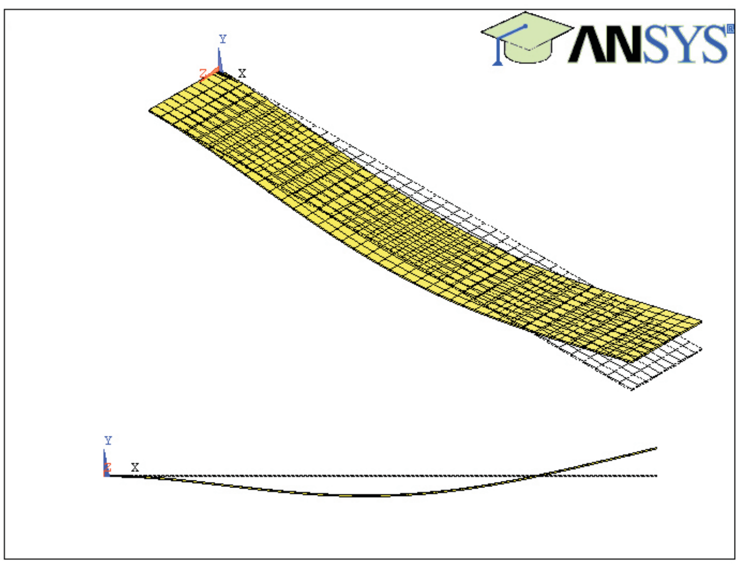

Figure 5: Deformed Beam Shape at the Second Natural Frequency

In general, there are no accurate available theoretical means of calculating ζ_i, and we need to rely on experimental results. Experiments can also be used to estimate the first few natural frequencies. To illustrate this, assume for simplicity that $N = 1$ in (2.8). A typical time response

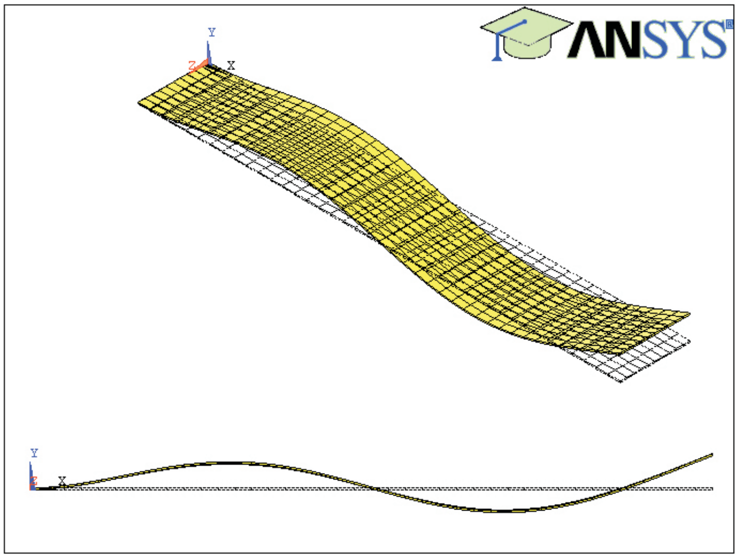

Figure 6: Deformed Beam Shape at the Thrid Natural Frequency

of the beam acceleration for this case is shown in Figure 7. From this response, ω_{n1} and ζ_1 can be computed from

$$\omega_{n1} = \frac{2\pi}{T_1} \qquad A_{m+1} = A_m e^{\left(-2\pi\zeta_1\sqrt{1-\zeta_1^2}\right)}. \tag{2.11}$$

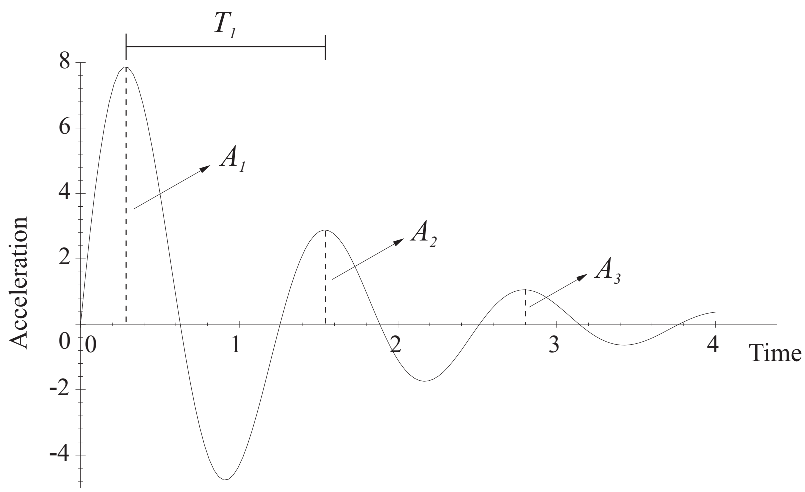

Figure 7: Typical Acceleration Time Response for $N = 1$

When the number of modes increases, the acceleration response becomes a series of superimposed, damped sinusoidal waveforms. Figure 8 exemplifies such a response. Note that in this example one cannot easily identify the different natural frequencies. However, in many situations, the terms for the higher harmonics ($N \geq 2$) attenuate faster. Hence, one can focus at the beginning of the waveform to better visualize the effect of these harmonics, and experimentally evaluate ω_{ni} and ζ_i, $i \geq 2$.

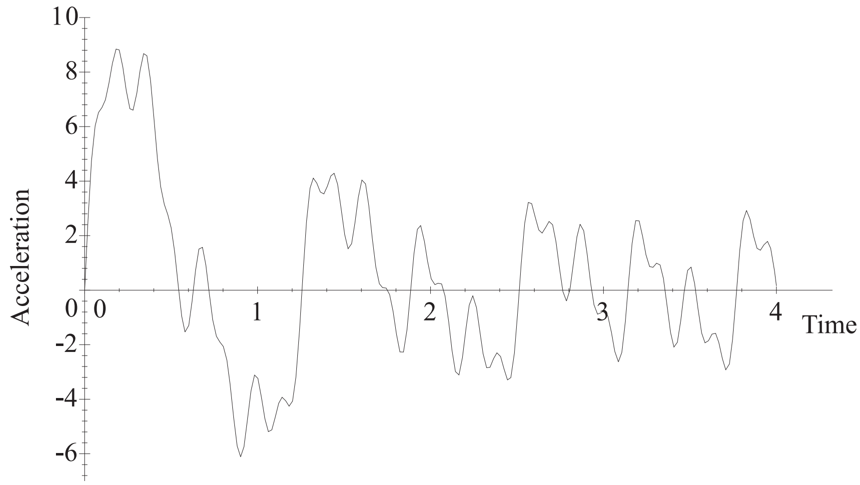

Figure 8: Typical Acceleration Time Response for $N \geq 1$

Alternatively, if the acceleration time response is transformed to the frequency domain, the magnitude frequency response of the acceleration signal will contain a series of spikes located at the natural frequencies. That is, the signal spectrum will clearly indicate its frequencies of vibration. Figure 9 shows the time response and frequency spectrum of the following signal

$$x(t) = 3\sin(2\pi \times 10^2 t) - 6\cos(2\pi \times 10^3 t) + randomnoise. \tag{2.12}$$

Note that it is almost impossible to distinguish the signal's 100 and 1000 Hz frequency contents from the time response; however, this information is quite evident from the frequency spectrum. The procedure that converts a signal time response into its frequency spectrum is called the Fast Fourier Transform (FFT). In recent years, instruments called *spectrum or frequency analyzers* have become quite popular for real-time computation of FFT's.

A third method of identifying the beam's natural frequencies is through the system's sinusoidal response. Recall that the natural frequencies correspond to peaks on the magnitude frequency response. Therefore, if a sinusoidal signal excites the beam with a frequency close to the natural frequencies, the beam's output response (*e.g.*, the tip acceleration) in time will contain large amplitudes relatively to the output amplitudes at other frequencies.

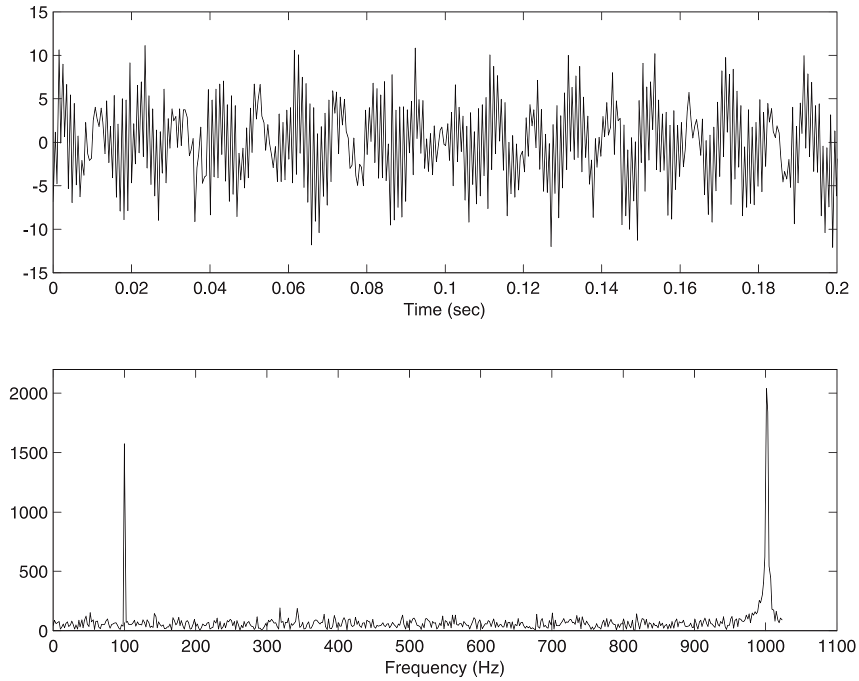

Figure 9: Time Response and Frequency Spectrum of a Signal

3. Equipment List

i) PC with Q4 DACB and terminal board

ii) Software environment: Windows, MATLAB, Simulink, RTW, and QuaRC

iii) A flexible, steel beam with the following characteristics: $L = 0.37$ m, $A = 4.5161 \times 10^{-5}$ m^2, $E = 2.1 \times 10^{11}$ $\frac{N}{m^2}$, $I \approx 2.9743 \times 10^{-12}$ m^4, and $\rho \approx 7860$ $\frac{kg}{m^3}$.

iv) Piezoelectric accelerometer, charge amplifier, and connecting cables.

v) A mini-shaker, a function generator, a power amplifier, power supply, and connecting cables.

4. Experimental Procedure

i) Setup the acceleration experiment testbed properly as shown in Figure 10. **Before proceeding, you must request the laboratory TA to approve your experiment setup**.

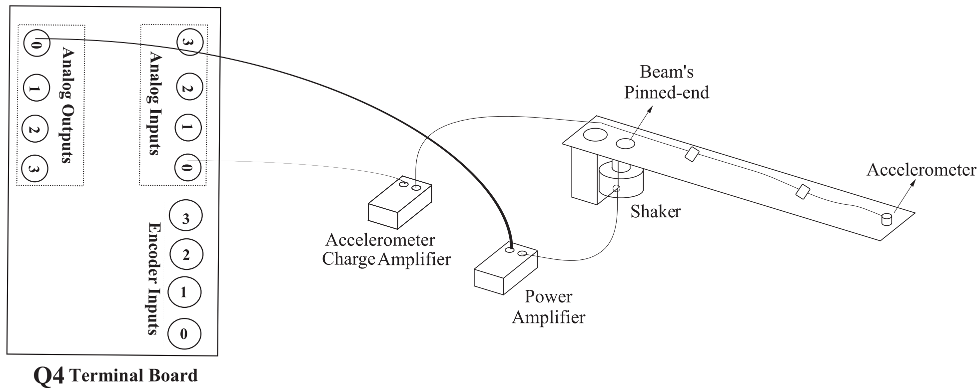

Figure 10: Clamped-Pinned Beam Experimental Setup

ii) From the **Start** button of the Windows toolbar, select the option sequence **Programs–MATLAB–R2007b–MATLAB R2007b** to launch the MATLAB application.

iii) In the MATLAB window, choose "C:\MeasurementSystemsLab\Experiment6" from the Current Directory window. This directory path choice will change the directory from the default MATLAB directory to the working directory for Experiment 6.

iv) From the **File** menu of the MATLAB window, select the option **Open** to load the Simulink block-diagram "Experiment6.mdl." to your desktop.

v) With the shaker turned off, one person induces a vibration on the beam by displacing its tip from the rest position and then releasing it At the same time, another person clicks the black **Start** arrow button to acquire the real-time data for the experiment.

vi) In the Simulink block-diagram, the To File block creates "Exp6Data.mat" which the plot data is saved on. Plot the accelerometer's time response from the MATLAB window by executing the following commands: load Exp6Data and plot(data6(1,:),data6(2,:)).

vii) Using the shaker and function generator, slowly increase frequency of the function generator from 1Hz to 150Hz.

5. Analysis/Assignment

i) In steps *i*)—*vi*) of Section 4, estimate the first natural frequency and the damping ratio of the first mode. Compare your experimental natural frequencies with the theoretical ones calculated by using (2.9) and (2.10). Comment on any discrepancies.

ii) In step *vii*) of Section 4, what is the first three natural frequencies experimentally determined? Compare your results with the expected frequency response of calculated by using (2.9) and (2.10). Comment on any discrepancies.

References

1. W. Bolton, *Mechatronics: Electronic Control Systems in Mechanical and Electrical Engineering*, Addison-Wesley, New York, NY, 1999.

2. P. Elgar, *Sensors for Measurement and Control*, Addison-Wesley Longman, Essex, England, 1998.

3. M.B. Histand and D.G. Alciatore, *Introduction to Mechatronics and Measurement Systems*, WCB/McGraw-Hill, Boston, MA, 1999.

4. S.S. Rao, *Mechanical Vibrations*, Addison-Wesley, Reading, MA, 1995.