

Contents

Preface	x
1 Electrical Measurements: <i>The Basics</i>	1
1.1 Introduction	1
1.2 Concepts & Information	2
1.3 Digital Multi-Meter	7
1.3.1 The Ammeter	7
1.3.2 The Volt Meter	7
1.3.3 The Ohm Meter	8
1.4 The Oscilloscope & function generator	8
1.5 Lab Instructions	9
1.6 Problems	11
References	11
2 Resistor Networks	13
2.1 Introduction	13
2.2 Concepts & Information	13
2.2.1 Labeling & Polarity Conventions	13
2.2.2 Network Analysis Techniques	15
2.2.3 Energy & Power	18
2.3 Lab Instructions	19
2.4 Problems	21
3 Superposition Principle and Thevenin Equivalent Circuits	23
3.1 Introduction	23
3.2 Concepts & Information	23
3.3 Lab Instructions	27
3.4 Problems	31
References	31
4 RC Circuits	33
4.1 Introduction	33
4.2 Background	34
4.2.1 Parallel Plate Capacitor	35
4.2.2 Cylindrical Capacitor	36
4.2.3 Parallel & Series Combinations	37
4.3 Time Constants	38

4.3.1	Charging the Capacitor	38
4.3.2	Dis-charging the Capacitor	39
4.4	Measurement Setup for RC Time Constants	40
4.4.1	Pre-Lab	41
4.4.2	Obtaining Good Measurements	41
4.4.3	Test Your Set Up <u>PRIOR</u> to Recording Data	42
4.5	Integration & Differentiation	42
4.5.1	Integration	42
4.5.2	Differentiation	44
4.6	Lab Instructions: Part A	45
4.7	Data to Include in Write Up	46
4.8	Problems	47
4.9	Phase and Frequency Relations Simple RC Circuits	49
4.10	The Low-Pass Filter	49
4.10.1	Frequency & Phase Response	50
4.10.2	Bode Plots	51
4.11	The High-Pass Filter	52
4.11.1	Frequency & Phase Response	52
4.11.2	Bode Plots	53
4.12	Amplitude and Phase Measurements	54
4.13	Lab Instructions: Part B	57
4.14	Report Format	58
4.14.1	Calculated Curves	58
4.14.2	Measurements	59
4.14.3	Bode Plots	60
4.14.4	Results	60
4.15	Appendix	60
4.16	Problems	61
	References	62
5	The Twin T Filter	64
5.1	Introduction	64
5.2	Background	65
5.2.1	Higher Order Low Pass and High Pass Filters	65
5.2.2	Band Pass and Band Stop Filters	66
5.3	Twin T Filter	67
5.4	Applications	70
5.5	Measuring the Twin T Filter	71
5.6	Lab Instructions	72
5.7	Problems	74
	References	74

6	RL Circuits	75
6.1	Introduction	75
6.2	Background	76
6.2.1	Construction of Inductor Coil	79
6.2.2	Parallel & Series Combinations	80
6.2.3	Mutual Inductance & Transformers	80
6.3	Step Response of an RL Circuit	82
6.3.1	Positive Step Response	82
6.3.2	Negative Step Response	84
6.4	Phase and Frequency Relations of RL Circuits	84
6.5	The RL Low-Pass Filter with an Ideal Inductor	85
6.5.1	Integration	86
6.5.2	Frequency & Phase Response	87
6.6	The RL High-Pass Filter with an Ideal Inductor	89
6.6.1	Differentiation	89
6.6.2	Frequency & Phase Response	90
6.6.3	Bode Plots	91
6.7	Non-Ideal Inductors	91
6.8	Lab Instructions	93
6.9	Problems	95
	References	96
7	Series RLC Circuit	98
7.1	Introduction	98
7.2	Response of Simple RLC	99
7.2.1	KVL	101
7.2.2	Phasors	103
7.3	Measurements	104
7.4	Lab Instructions: Part A	106
7.5	Problems	107
7.6	Detailed Analysis of Series RLC Circuit	109
7.7	1 st and 2 nd Order Differential Equations	109
7.7.1	Steady-State Response	113
7.7.2	General Transient Response	113
7.8	Mechanical Systems	114
7.9	Measuring the Transient Response	116
7.10	Lab Instructions: Part B	117
7.11	Problems	118
	References	118
8	Simple Op Amp Circuits	120
8.1	Introduction	120
8.2	The Non-Ideal Op Amp	121
8.3	The Inverting Amplifier	123
8.4	The Integrator	124
8.5	The Differentiator	127

8.6	Lab Instructions	129
8.7	Problems	131
	References	131
9	Laboratory Problem: S-E Filter	133
9.1	Introduction	133
9.2	Filter Description	134
9.3	Part Placement Assembly	135
9.4	Instructions	136
9.5	Questions	137
	References	138

List of Figures

1.1	Loop of n elements.	3
1.2	Node containing n connections.	3
1.3	Voltage Divider.	4
1.4	Current Divider.	4
1.5	Illustration of resistor bands.	6
1.6	Simplified schematic description of an Ammeter.	7
1.7	Simplified schematic description of an voltmeter.	8
1.8	Resistor configurations for part 1.	9
1.9	A circuit to measure the ammeter resistance.	10
1.10	A circuit to measure the voltmeter resistance.	10
2.1	Sample network.	14
2.2	Simplified network from figure 2.1.	15
2.3	Network from figure 2.2 with loop currents labeled.	16
2.4	Resistor Network.	19
2.5	Resistor Network.	20
3.1	Independent voltage and current source.	24
3.2	Dependent sources.	24
3.3	Circuit to illustrate superposition principle.	25
3.4	Circuits for $V_{AB}^{(1)}$ & $V_{AB}^{(2)}$	25
3.5	Circuits to illustrate Thevenin's Theorem.	26
3.6	Finding R_{th}	26
3.7	The Norton Equivalent Circuit.	27
3.8	Thevenin Equivalent Circuit.	27
3.9	Network for Superposition Measurements.	28
3.10	Network for Superposition Measurements.	28
3.11	Network for Superposition Measurements.	28
3.12	First Network to Estimate R_{th}	29
3.13	Second Network to Estimate R_{th}	30
3.14	Wheatstone Bridge.	31
4.1	Blocking Capacitor.	34
4.2	Illustration of a parallel plate capacitor.	36
4.3	Electric Field in a parallel plate capacitor.	36
4.4	Illustration of a parallel plate capacitor.	37

4.5	Charging cap.	37
4.6	Charging cap.	38
4.7	Relative voltage as a function of time constants during charge up.	39
4.8	Cap Discharge.	39
4.9	Relative voltage as a function of time constants for the discharging of the capacitor.	40
4.10	Measurement set up for time constant.	41
4.11	RC circuit with voltage measured across the capacitor.	43
4.12	Low Pass Filter.	50
4.13	Low Pass Filter amplitude and phase response.	51
4.14	High Pass Filter.	52
4.15	High Pass Filter amplitude and phase response.	53
4.16	Low Pass Filter (LPF) measurement set up.	54
4.17	Channel 2 (bottom) <i>Lags</i> Channel 1 (top).	55
4.18	High Pass Filter (HPF) measurement set up.	55
4.19	Channel 2 (bottom) <i>Leads</i> Channel 1 (top).	56
4.20	Channel 2 (bottom) & Channel 1 are in phase (top).	56
5.1	2^{nd} -order pass filter.	66
5.2	Magnitude response versus LPF filter order.	66
5.3	Band stop filter.	66
5.4	Comparison of band and pass filter and band stop filters.	67
5.5	Twin T filter.	68
5.6	Twin T high pass and low pass sections.	68
5.7	Twin T magnitude & phase plots.	70
5.8	Detailed view of the Twin T magnitude response.	71
5.9	Example application of the Twin T Filter.	71
6.1	Magnetic field due to a straight wire using the right hand rule.	76
6.2	Computing the magnetic field on the axis of a current loop.	77
6.3	Inductor with N turns over a length l with cross sectional area, A	79
6.4	Magnetic field of a coil.	79
6.5	Adjacent circuits.	81
6.6	Iron core transformer.	81
6.7	Circuit for the positive step response.	83
6.8	Relative voltage as a function of time constants for the positive step.	83
6.9	Circuit for the negative step response.	84
6.10	Relative voltage as a function of time constants for the discharging of the capacitor.	85
6.11	Low pass filters realized from RL and RC sections.	86
6.12	Low pass filter frequency and phase response.	88
6.13	High pass filters realized from RL and RC sections.	89
6.14	High pass filter frequency and phase response.	91
6.15	Composite non-ideal inductor.	92
6.16	Low Pass Filter.	93

6.17 High Pass Filter.	94
7.1 Series RLC circuit.	99
7.2 Frequency Response and Phase Response of Series RLC circuit. . .	100
7.3 Impact of Q value on the frequency response.. . . .	101
7.4 Phasor diagrams for $f_{in} = 10$ kHz.	104
7.5 Measurement on series RLC circuit.	105
7.6 Examples of transient responses.	115
7.7 Mass on a spring.	116
8.1 Typical open loop frequency response of an op amp.	121
8.2 Rise time measurement of the response to a step input.	122
8.3 Inverting amplifier.	123
8.4 Pure integrator.	124
8.5 DC model of the pure integrator.	126
8.6 Active, first-order low pass filter.	127
8.7 Pure differentiator.	128
8.8 Active, first-order high-pass filter.	129
9.1 S-E filter[2].	134
9.2 S-E filter[2].	135
9.3 S-E filter board part placement & wiring (Top & Bottom). . . .	135

Preface

The design of electrical circuits, devices and systems is an integral part of the field of Electrical Engineering. A student graduating with a degree in Electrical Engineering should therefore be a good designer. There are 3 requirements in order for a student to become a good designer: 1) understanding of the basic theory, 2) analysis techniques and 3) the measurement of electrical circuits. The purpose of this book is to provide the necessary material to perform basic measurements. This is accomplished by offering a course which will cover laboratory exercises in linear circuit theory. There are 9 chapters which cover techniques and procedures in the measurement and characterization of active and passive circuits.

Each chapter provides the student with sufficient background to efficiently and intelligently apply linear circuit theory to the laboratory exercise topic. The format of each chapter is designed to address:

- i) Understanding the Objective: The theory behind the laboratory exercise topic is covered at the most basic level. In chapters which cover reactive components, capacitors and inductors, a brief review of the relevant equations from Electricity and Magnetism are used to derive the fundamental relations.
- ii) Issues with Measurements: In the formal course coverage of linear circuit theory the descriptions used in basic derivations cannot be directly carried out for the measurement due to the nature of the measurement tools, differences between ideal and non-ideal components and additional linear effects which are ignored in most fundamental derivations. Thus, problems are pointed out and they are avoided through disciplined laboratory procedures.
- iii) Closing the Semantic Gap Between Mathematics and Electrical Measurements: The formal descriptions of electrical circuits can be manipulated into a form which better suits the laboratory measurement. The student needs to *connect* the mathematical description to what is observed. Students tend to appear to *get* this quickly and adapt; however, they often fail to recognize how the basic theory affects a particular measurement. This leads to elementary mistakes and misconceptions which are far too common in both academics and practice. A lack of skill in complex arithmetic is by

far the biggest stumbling block. The classic problem is in the interpretation of complex quantities viewed in measurements vs. what the student has seen in a text book description. The laboratory exercises are designed to provide the student with a *hands on* understanding of how to *translate* between what is observed and how to characterize the observation properly. With a well thought out procedure and redundant practice over a period of time, the student should be able to understand how the result is impacted by direct observation of the measurement device.

- iv) Communication and Interpretation of Results: It is important for the student to understand that there are common analogies between linear circuits, linear systems and even mechanical systems. It is useful for students to identify generalities by understanding the *language* used to characterize a particular effect. The should be able to observe similarities between topics here and their junior courses in electronics and signals and systems.

It is important for students to realize that the course described in this book is separate and distinct from other courses taken during the semester it is offered. Typically the course covered in this book is taken concurrently with a formal circuit theory course. It is not necessary for the topics covered in the laboratory exercises to be sequenced after, e.g. *wait*, until they are covered in the formal circuit theory course. Otherwise, the objectives of this course, which are absolutely necessary, would be compromised. Sufficient background material is covered in order to provide a basic understanding of each laboratory exercise. The topics in this book are designed for the student who has completed a freshman/sophomore level Physics course in Electricity and Magnetism with a passing grade. Students are also expected to have taken the courses in mathematics (e.g. elementary calculus, simple differential equations) required of all freshman and sophomore Electrical Engineering students.

A. Davis, December 2005

Chapter 1

Electrical Measurements: *The Basics*

Lab Objectives:

- §1. To learn how to make simple measurements.
- §2. To understand the operation of the lab equipment.
- §3. To employ circuit theory in measurements.

Pre Lab:

1. Read this lab.
2. Read up on Oscilloscopes and meters.

1.1 Introduction

Engineers do their jobs by applying principals of circuit theory to devices which are fabricated and placed in service either as a manufactured product or as a prototype (1 or 2 versions). Few electrical circuits are built without requiring measurements. Typical measurements can be a DC or AC voltage or current. The resistance of a system is also of interest. Sometimes we need to examine the actual waveform over a period of time.

The instruments at your lab bench will enable you to perform all of the tasks mentioned. Let's review the equipment:

- 1) A digital multi-meter used to measure resistance, AC/DC voltage and current.
- 2) An analog Oscilloscope which allows us to examine periodic wave forms over selected time intervals.
- 3) A DC power supply.
- 4) The Tektronics FG 503 function generator. The function generator provides us with the following AC waveforms:

- a) Square Wave.
- b) Saw-tooth Waveform.
- c) Triangle Waveform.
- d) Sine Wave.
- e) Possibly others.

Notice that any AC waveform can also have a DC offset voltage added to it.

- 5) A frequency counter.
- 6) Miscellaneous tools and cables.

While some of the techniques to measure various electrical quantities are obvious, some measurements require a simple understanding of the instrument used in the measurement and the application of basic electrical concepts in order to obtain a reliable result.

1.2 Concepts & Information

In your circuit theory course you are learning or have learned (if it was not taken concurrently) some basic laws and relations. Some relationships are used so often, that it is well worth the time to review them. Every simple relationship is derived from one or more basic laws. Some of these concepts are stated here to remind you that they still apply in this course and any other course you will be taking. In fact, these laws apply even when you leave the state or the country.

Information has been provided on resistor codes in order for you to understand how to read the resistor values from the color bands.

- a) Ohm's Law

Ohm's law is a linear relationship between voltage, current and resistance. It is expressed as

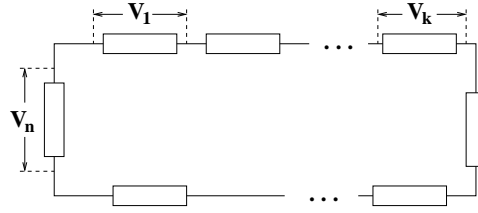
$$V = IR \quad (1.1)$$

where V is the voltage, I is the current and R is the resistance.

- b) Kirchhoff's Voltage Law (KVL) & Kirchhoff's Current Law (KCL)

- i) KVL states: The sum of the voltages around a loop is zero.
In general, if we have n elements connected in a loop as shown in figure 1.1,
then, the sum of ALL voltage drops is expressed as

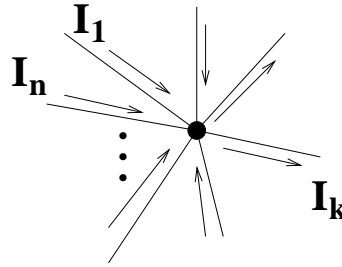
$$\sum_{k=1}^n V_k = 0 \quad (1.2)$$

Figure 1.1: Loop of n elements.

where k is the index, n is the total number of elements and V_k is the voltage drop across the k_{th} element.

ii) KCL states: The sum of currents at a node must be zero.

In general, if we have a node with n connections, as shown in figure 1.2, then, the sum of ALL currents coming into or out of the node is ex-

Figure 1.2: Node containing n connections.

pressed as

$$\sum_{k=1}^n I_k = 0 \quad (1.3)$$

where n is the total number of nodes, k is the index and I_k is the current in the k^{th} node.

c) Series and Parallel Resistors

Of course, the equivalent resistance of n resistors connected in series is the sum or

$$R_{eq} = R_1 + R_2 + \dots + R_n \quad (1.4)$$

Alternatively, the equivalent resistance of n resistors connected in parallel is expressed as

$$\frac{1}{R_{eq}} = \frac{1}{R_1} + \frac{1}{R_2} + \dots + \frac{1}{R_n} \quad (1.5)$$

Both expressions are derived from applying KVL/KCL.

d) Voltage Divider

By placing two or more resistors in series the the total voltage can be divided. The example in figure 1.3 illustrates this using two series resistors. The voltage across R_2 can be found by applying Kirchoff's Voltage Law.

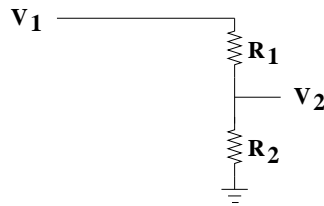


Figure 1.3: Voltage Divider.

The relationship between V_2 and V_1 is given by

$$V_2 = \frac{R_2}{R_1 + R_2} V_1 \quad (1.6)$$

where V_2 is the voltage across R_2 and V_1 is the voltage across R_1 and R_2 .

e) Current Divider

Current division occurs for two or more parallel resistors. Kirchoff's current law can be used to solve for individual currents. An example of this is shown for two parallel resistors in figure 1.4.

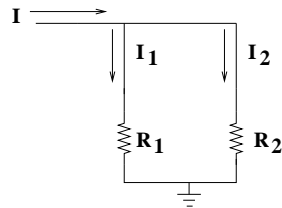


Figure 1.4: Current Divider.

The relationship between I_2 and I is given by

$$I_2 = \frac{G_2}{G_1 + G_2} I \quad (1.7)$$

where G_i is the conductance (e.g. $\frac{1}{R_i}$) for resistor i . An alternative expression is given by

$$I_2 = \frac{R_1}{R_1 + R_2} I \quad (1.8)$$

which replaces the conductance with resistance.

f) AC Measures

i) Root-Mean-Square Voltage (V_{rms})

The average value of a sine wave over a complete period is zero. This can be done by inspection of the graph over a single period. It is positive for $\frac{1}{2}$ -period and it is negative for $\frac{1}{2}$ -period which effectively makes the average zero.

The Root-Mean-Square Voltage for an arbitrary periodic waveform is given by[1]

$$V_{rms} = \sqrt{\frac{1}{T} \int_0^T v^2(t) dt} \quad (1.9)$$

where T is the period and $v(t)$ is the AC voltage as a function of time. The rms voltage of a sine wave can be found by setting $v(t) = V_{peak} \sin \omega t$, which yields

$$V_{rms} = \sqrt{\frac{\omega}{2\pi} \int_0^{\frac{2\pi}{\omega}} V_{peak}^2 \sin^2(\omega t) dt} \quad (1.10)$$

This results in the following expression for a sine wave

$$V_{rms} = \frac{V_{peak}}{\sqrt{2}} = 0.707 V_{peak} \quad (1.11)$$

or, V_{rms} is approximately 71% of the peak value for a sine wave.

ii) $\frac{1}{2}$ -Period Average Value[1]

The average voltage for a sine wave is sometimes computed by taking the mean over a $\frac{1}{2}$ -period interval, or

$$V_{avg} = \frac{\omega}{\pi} \int_0^{\frac{\pi}{\omega}} V_{peak} \sin(\omega t) dt \quad (1.12)$$

which results in

$$V_{rms} = \frac{2V_{peak}}{\pi} = 0.637V_{peak} \quad (1.13)$$

or, approximately 64% of the peak value.

g) Resistor Color Codes

Resistors are marked with 4 bands as shown in figure 1.5. The value of the resistor can be read from the first 3 bands. The first two bands indicate the numerical value while the 3rd band indicates the multiplier power.

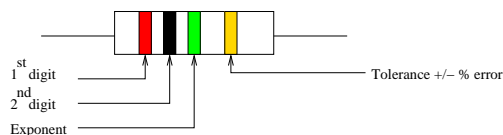


Figure 1.5: Illustration of resistor bands.

Table 1.1 provides a complete list of codes used to enumerate all resistor values.

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

The 4th band is the tolerance band. The tolerance is a figure of merit which indicates how close the resistance should be to the value encoded. Gold indicates 10 % tolerance and silver represents 1 % tolerance. Thus, the actual resistance of a resistor will deviate from the designed value by a small amount.

1.3 Digital Multi-Meter

The measurement of voltage, current and resistance can be performed by a coil with an attached needle. The needle movement is calibrated to the current flowing through the coil. Using Ohm's Law, voltage can also be measured by scaling the meter reading to a resistive network. Voltage, current and resistance are measured using a device with a digital readout, the digital multi-meter (DMM). Although the behavior appears to be somewhat different many features can be understood using the same simple models.

1.3.1 The Ammeter

The ammeter, shown in figure 1.6, consists of a current measuring device and a resistor, R_m , which represents the internal resistance of the measuring device (an ideal ammeter would have an R_m equal to 0).

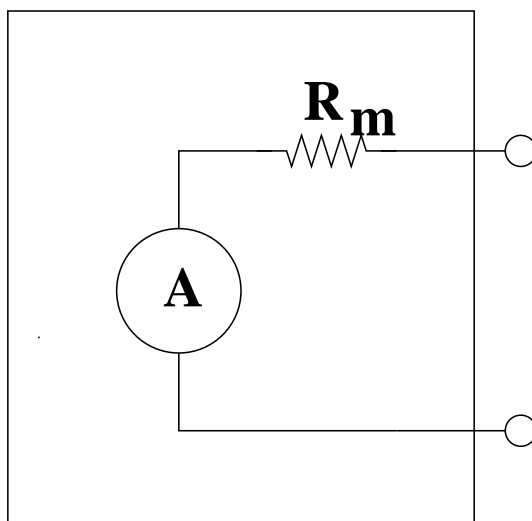


Figure 1.6: Simplified schematic description of an Ammeter.

This means that there is an error, albeit small, due to the internal resistance of the ammeter.

1.3.2 The Volt Meter

The voltmeter, shown in figure 1.7, can be modeled as a meter and a resistor, R_m , in parallel which represents the internal resistance. The ideal voltmeter would have an R_m equal to ∞ .

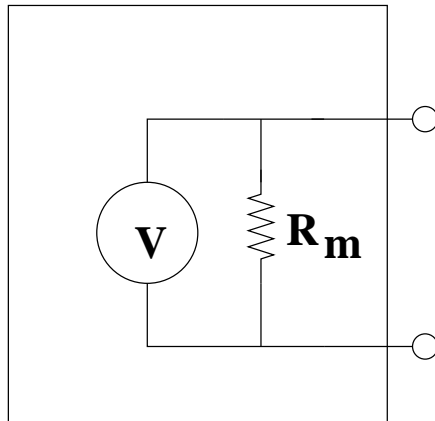


Figure 1.7: Simplified schematic description of a voltmeter.

You should also realize that voltage and current meters have a number of scales used for measurements. These scales are set by resistor networks.

1.3.3 The Ohm Meter

Resistance is measured by supplying a small current to the circuit under test. The voltage and current are measured and the resistance is found by using Ohm's Law.

The ohm meter can also be used to detect “shorts” or “open” circuits since the “short” will show zero resistance and the “open” will show infinite resistance.

Since the DMM in the resistance mode will supply current, some care must be taken when making resistance measurements. In general, resistance is measured with an ohmmeter for only two cases: (1) measuring resistor values and (2) detecting “shorts” or “open” circuits when it is safe to perform this task with an ohmmeter. Care must also be taken, even when there is no power applied to the device under test, because the current supplied by the ohm meter could damage some semiconductor devices.

1.4 The Oscilloscope & function generator

Measurements performed on AC waveforms often require the waveform to be viewed as a function of time. This is the purpose of the Oscilloscope. The Oscilloscope is a CRT with horizontal and vertical deflection plates. The horizontal plates are connected to the internal time base. The vertical plates are connected to the source or circuit under test.

Oscilloscopes have 2-4 channel inputs. These inputs are scaled (Volts/div)

and the following selections are provided: (1) Ground, (2) AC and (3) DC. The Ground setting is used to reference ground on the screen. The AC setting eliminates DC offset. AC waveforms can have DC components and sometimes viewing the complete signal is difficult. For example, a very small AC signal with a very large DC offset. The DC selection provides DC offset. Thus, if you are trying to measure offset, the AC setting will not be a good choice.

The function generator is an instrument which provides test waveforms. A circuit being analyzed often requires test input signals. Since the design should be aware of how his/her circuit functions, the output signal is viewed and compared to the expected result.

You will be expected to know how to read the period of a signal using the time base setting and counting the approximate number of vertical blocks and tic marks on the screen. You will also be expected to know how to measure the voltage of a waveform using the scaled voltage setting and counting the vertical blocks and tic marks.

1.5 Lab Instructions

1) Resistance Measurements

Select R_1 in the range of $220\ \Omega$, R_2 in the range $10\text{K}\Omega$ to $22\text{K}\Omega$, and R_3 in the range $1\text{M}\Omega$ to $2.2\text{M}\Omega$. Assemble them in the configurations shown in figure 2.4 and measure the equivalent resistance R . Make sure that you are selecting actual resistors for this exercise.

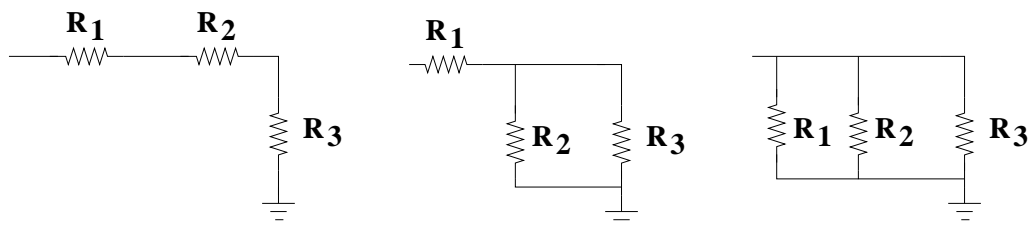


Figure 1.8: Resistor configurations for part 1.

Once you have selected the values for R_1 , R_2 and R_3 , measure each resistor with the DMM and compute the effective resistance.

2) Ammeter Internal Resistance

Set up the circuit in figure 1.9 with $R_1 = 100\text{k}\Omega$ and $R_2 = \infty$

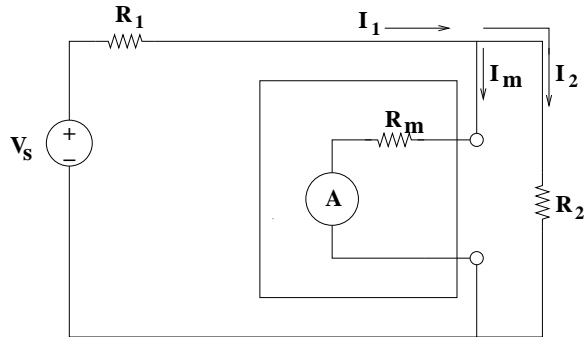


Figure 1.9: A circuit to measure the ammeter resistance.

Adjust the DC voltage, V_s , until the ammeter reads $100 \mu A$ on the $200 \mu A$ scale. Reduce R_2 until the current becomes $50 \mu A$ (e.g. $R_m = R_2$).

3) Voltmeter Internal Resistance

A circuit to measure the internal resistance of a voltmeter is illustrated in figure 1.10.

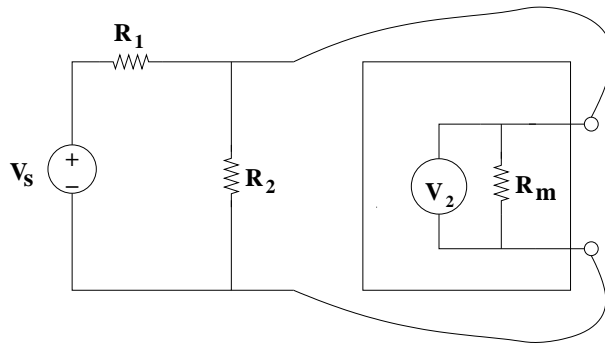


Figure 1.10: A circuit to measure the voltmeter resistance.

The value of R_m is given by

$$R_m = \frac{R_1 R_2}{R_2 V_s - (R_1 + R_2) V_2} V_2 \quad (1.14)$$

where R_1 , R_2 , V_s and V_2 are given in figure 1.10.

Derive this expression.

Set up the circuit in figure 1.9 with $R_1 = 3.3 \text{ M}\Omega$ and $R_2 = 1.2 \text{ M}\Omega$. Find R_m .

Repeat this measurement using $R_2 = 3.3 \text{ M}\Omega$.

4) AC Measurements

Set up the function generator and the oscilloscope with a sine wave; measure the sine wave across a $1 \text{ M}\Omega$ resistor. Set the frequency to 1 kHz and the peak amplitude to 4 volts . Verify that the offset is set to zero volts.

Now, attach the output of the function generator to a $50 \text{ }\Omega$ resistor. What happens ?

Find the rms value of the sine wave.

1.6 Problems

- 1) Show the *rms* value of a sine wave (peak amplitude= V_x) over one period ($T = 2\pi$) is given by equation (1.11). Hint: use equation (1.10) and substitute a suitable trigonometric identity for $\sin^2(\omega t)$.
- 2) Find the rms value of a square wave (peak amplitude= V_x) using equation (1.9).
- 3) Assume that you have 3 1k resistors; two parallel resistors are placed in series with the third resistor as shown in figure 1.8 (middle). The voltage at the left node of R_1 is given by V_1 . If $V_1 = 10\text{V}$, find V_2 .
- 4) Derrive equation (1.14). Traansform figure 1.10 into a schematic and and then solve for R_m .

References

- [1] A. J. Diefenderfer, *Principles of Electronic Instrumentation*. W.B. Sanders, 1972.

Chapter 2

Resistor Networks

Lab Objectives:

- §1. Continue with DC circuit analysis.
- §2. Apply Ohm's Law, KVL & KCL to measurements.
- §3. Find power dissipation in a resistive circuit.

Pre Lab:

1. Read this lab.
2. Derrive the solutions to the expressions for current and voltage.

2.1 Introduction

In this lab you will be applying Kirchhoff's voltage and current Laws, (KVL) and (KCL), to resistive networks. You will also continue with exercises designed to help you understand the influence of the measuring equipment. Finally, you will make some simple measurements to determine the power dissipation in a resistive network.

2.2 Concepts & Information

KVL and KCL can be applied to complex networks in order to find the node voltages and current through each branch. A representative circuit appears in figure 2.1.

2.2.1 Labeling & Polarity Conventions

The nodes in figure 2.1 are labeled A through D while the branch currents can be labeled I_k where k represents resistor R_k . Thus, the current I_1 would represent the current passing through the resistor R_1 .

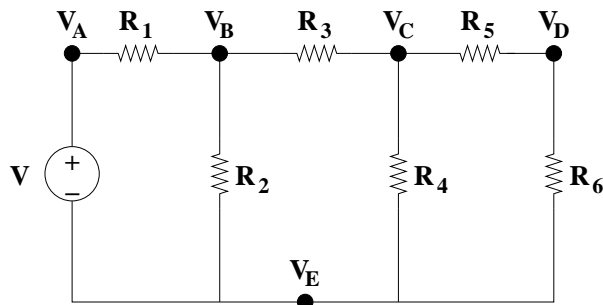


Figure 2.1: Sample network.

Voltages and currents must have signs as well as labels. By convention, voltages are referenced to a potential. The voltage at node B , $V(B)$ or V_B is the voltage measured at node B with respect to node D . The sign is “+” at the node with the higher voltage and “-” at the terminal with the lower voltage.

For example, if we were to measure the node voltage at node B with respect to node D in figure 2.1, then the red (or positive) lead of the DVM is placed at node B and the black (or negative) lead is placed at node E . Mathematically, this is equivalent to finding V_{BD} , or

$$V_{BD} = V_B(\text{red}) - V_E(\text{black}) \quad (2.1)$$

If we are interested in the voltage drop across R_3 , then we want to find the voltage difference between nodes B and C . The voltages at node B with respect to node D and at node C with respect to node D can be measured and their difference can be taken to be

$$V_{BC} = V_{BE} - V_{CE} = V_B - V_C \quad (2.2)$$

Since node E was common to both voltages, the node E is dropped from the final result V_{BC} . If voltage nodes are measured with respect to a common reference, e.g. node E , then the voltages at nodes B and C can be referred to as V_B and V_C . In fact, if we want the voltage between nodes B and C , we could alternatively put the red (positive) lead on node B and the black (negative) lead on node C and directly measure the voltage difference. If we reverse the direction of the leads, then we have

$$V_{CB} = -V_{BC} = V_C(\text{red}) - V_B(\text{black}) \quad (2.3)$$

KVL should be satisfied by summing the voltages around *any* loop in the network. For example, measuring the voltages between nodes A & B , B & C , C & D and D & A would be equivalent to

$$V_{AB} + V_{BC} + V_{CD} + V_{DE} + V_{EA} = 0 \quad (2.4)$$

Notice that $V_{AD} = V_A = V_s$, where V_s is the voltage of the voltage source in figure 2.1. Thus, $V_{DA} = -V_s$. A non-rigorous proof of KVL can be performed

by substituting the definitions of each voltage difference in equation (2.4). This becomes

$$(V_A - V_B) + (V_B - V_C) + (V_C - V_D) + (V_D - V_E) + (V_E - V_A) = 0 \quad (2.5)$$

It can be seen by inspection that equation (2.5) must be equal to 0; equation (2.5) is also equivalent to equation (2.4). By $V_{AE} = V_A = V_s$, equation (2.4) can also be re-written as

$$V_s = V_{AE} = -V_{EA} = V_{AB} + V_{BC} + V_{CD} + V_{DE} \quad (2.6)$$

The current sign conventions follow directly from the conventions adopted with nodal voltages. By convention, a positive current flows from a positive node to a node with a less positive value. Thus, the current in R_3 can be found by

$$I_3 = \frac{V_B - V_C}{R_3} = \frac{V_{BC}}{R_3} \quad (2.7)$$

2.2.2 Network Analysis Techniques

There are 3 options to analyze a network: (1) Network Reduction, (2) Nodal Analysis and (3) Loop/Mesh Analysis. One could also apply nodal or loop analysis to a network which has been simplified.

a) Network Reduction

One method of solving for all of the branch voltages and currents for the network in figure 2.1 is to simplify it through redundant application of series and parallel resistors. By repeated conversions for equivalent series and parallel resistors one could sufficiently reduce the network in order to obtain a solution. For example, R_4 , R_5 , and R_6 can be combined as shown in figure 2.2. This approach increases quickly in complexity.

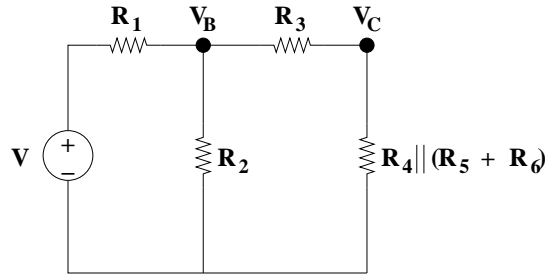


Figure 2.2: Simplified network from figure 2.1.

Notice that the voltages V_B and V_C remain unchanged (notice that V_A is the supply voltage, V). We could carry this again; instead, we will use this to

demonstrate nodal and loop analysis

b) Nodal Analysis

We will apply nodal analysis to the network in figure 2.2. By summing the currents at the nodes in figure 2.2 we can solve for the unknown voltages, V_B & V_C . Thus, nodal analysis relies on KCL.

This yields the following set of equations

$$\begin{aligned} 0 &= \frac{V-V_B}{R_1} - \frac{V_B}{R_2} + \frac{V_C-V_B}{R_3} \\ 0 &= \frac{V_B-V_C}{R_3} - \frac{V_C}{R_{46}} \end{aligned} \quad (2.8)$$

where $R_{46} = R_4 || (R_5 + R_6)$.

Equation (2.8) must be solved for V_B & V_C . This is done using techniques for solving simultaneous equations. The remaining nodal voltage, V_D from figure 2.1 by using a voltage divider between R_5 & R_6 . The solution allows one to find every voltage drop and every current.

c) Loop/Mesh Analysis

By summing voltage drops around each loop in the network one can solve for the currents in the network. Thus, loop/mesh analysis relies on KVL. Figure 2.2 is redrawn in figure 2.3 with the currents I_a and I_b labeled.

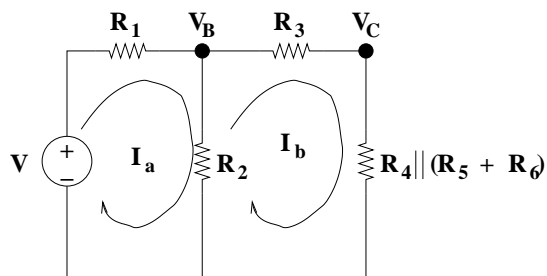


Figure 2.3: Network from figure 2.2 with loop currents labeled.

We intend to solve for the currents I_a and I_b . First, we must obtain the loop equations. This results in

$$\begin{aligned} V &= I_a R_1 + (I_a - I_b) R_2 \\ 0 &= I_b (R_3 + R_{46}) + (I_b - I_a) R_2 \end{aligned} \quad (2.9)$$

where I_a & I_b are the loop currents, V is the voltage source and $R_{46} = R_4 || (R_5 + R_6)$.

The resulting equations can be simplified as follows:

$$\begin{aligned} V &= (R_1 + R_2)I_a - R_2I_b \\ 0 &= -R_2I_a + (R_2 + R_3 + R_{46})I_b \end{aligned} \quad (2.10)$$

In matrix algebra, this takes the form:

$$\begin{bmatrix} V \\ 0 \end{bmatrix} = \begin{bmatrix} (R_1 + R_2) & -R_2 \\ -R_2 & (R_2 + R_3 + R_{46}) \end{bmatrix} \begin{bmatrix} I_a \\ I_b \end{bmatrix} \quad (2.11)$$

The solution of simultaneous equations comes up often in circuit analysis. The system of 2 equations and 2 unknowns can easily be solved. We will use a generic example where our system of equations are,

$$\begin{aligned} b_1 &= a_{11}x_1 + a_{12}x_2 \\ b_2 &= a_{21}x_1 + a_{22}x_2 \end{aligned} \quad (2.12)$$

where x_1 & x_2 are the unknowns, a_{ij} are the coefficients and b_1 & b_2 represent known values. This, in matrix form becomes,

$$\begin{bmatrix} b_1 \\ b_2 \end{bmatrix} = \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \end{bmatrix} \quad (2.13)$$

In order to solve this system of equations we must employ Cramer's Rule, inverting the matrix A, thereby canceling the matrix in multiplied by the unknowns and leaving us with,

$$\begin{bmatrix} x_1 \\ x_2 \end{bmatrix} = \begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix}^{-1} \begin{bmatrix} b_1 \\ b_2 \end{bmatrix} \quad (2.14)$$

In other words,

$$A^{-1}A = I = \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix} \quad (2.15)$$

where A is the 2x2 matrix and I is the identity matrix. It is named the identity matrix because it is the identity operator in matrix algebra. In a larger matrix this translates to all zeros with ones on the diagonal.

Inverting a 2x2 matrix is quite simple and well worth memorizing. This results in

$$\begin{bmatrix} a_{11} & a_{12} \\ a_{21} & a_{22} \end{bmatrix}^{-1} = \frac{1}{|A|} \begin{bmatrix} a_{22} & -a_{12} \\ -a_{21} & a_{11} \end{bmatrix} \quad (2.16)$$

where $|A| = a_{11}a_{22} - a_{21}a_{12}$, or the determinant of the matrix A.

If we apply this to equation (2.11), we obtain

$$\begin{bmatrix} I_a \\ I_b \end{bmatrix} = \frac{1}{D} \begin{bmatrix} (R_2 + R_3 + R_{46}) & -R_2 \\ -R_2 & (R_1 + R_2) \end{bmatrix} \begin{bmatrix} V \\ 0 \end{bmatrix} \quad (2.17)$$

where $D = (R_1 + R_2)(R_2 + R_3 + R_{46}) - R_2^2$.

The two remaining currents through R_4 and R_5 & R_6 can be found using current division.

2.2.3 Energy & Power

Energy & Power

The relationship between energy and voltage is

$$\begin{aligned} V &= \frac{dW}{dq} \\ W &= \int V dq \end{aligned} \quad (2.18)$$

where W , represents energy, in Joules, q represents charge in Coulombs, and V represents the voltage in volts ($1 \text{ Volt} = 1 \frac{\text{Joule}}{\text{Coulomb}}$).

Current is defined as the rate of charge flow; this can be stated mathematically as

$$I = \frac{dq}{dt} \quad (2.19)$$

where q is the charge in Coulombs, t is the time in seconds, and I is the current in Amperes. Thus, $1 \text{ Ampere} = 1 \frac{\text{Coulomb}}{\text{Second}}$.

Power is defined as the rate of energy, or,

$$\begin{aligned} P &= \frac{dW}{dt} = \left(\frac{dW}{dq}\right)\left(\frac{dq}{dt}\right) \\ P &= VI = I^2 R = \frac{V^2}{R} \end{aligned} \quad (2.20)$$

where P is expressed in Watts, where $1 \text{ Watt} = \frac{\text{Joules}}{\text{second}}$.

The average power is given by

$$\begin{aligned} P &= \frac{1}{T} \int_0^T \frac{v^2(t)}{R} dt \\ \text{or} \\ P &= \frac{1}{T} \int_0^T i^2(t) R dt \end{aligned} \quad (2.21)$$

which can be done either for voltage or current in a resistor.

The average DC power is constant and thus results in equation (2.20). The average power for a sine wave is computed by V_{rms} and/or I_{rms} in place of the DC voltage and currents.

If this is a DC circuit, then both $i(t)$ and $v(t)$ are constant over time, which thus simplifies to the familiar relationship

$$\begin{aligned} P &= \frac{V^2}{R} \\ \text{or} \\ P &= I^2 R \end{aligned} \quad (2.22)$$

Where V and I are the DC voltage and current, respectively.

2.3 Lab Instructions

- 1) Construct the circuit shown in figure 2.4. Measure branch voltages and currents.

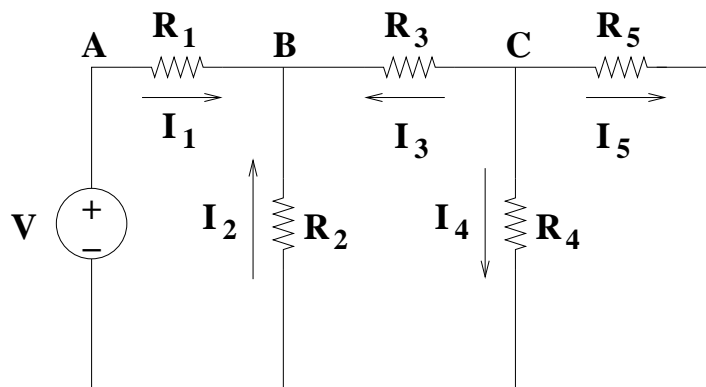


Figure 2.4: Resistor Network.

- i) Measure the following quantities:
- a) V_{AB}, V_{BC}, V_{CD} & V_{DA} .
 - b) I_1, I_2, I_3, I_4 & I_5 .
- ii) Use two sets of resistor values for R_{1-5} :
- a) $R_1 = 1.2k\Omega, R_2 = 2.7k\Omega, R_3 = 1.5k\Omega, R_4 = 1.2k\Omega$ & $R_5 = 6.8k\Omega$.
 - b) $R_1 = 3.3k\Omega, R_2 = 6.8k\Omega, R_3 = 2.2k\Omega, R_4 = 1.2k\Omega$ & $R_5 = 1.2k\Omega$.
- iii) For each set of measurements find:
- a)

$$V_{AB} + V_{BC} + V_{CD} + V_{DA} \quad (2.23)$$

b)

$$V_{AB} + V_{BD} \quad (2.24)$$

c)

$$V_{AB} + V_{BC} + V_{CD} \quad (2.25)$$

d)

$$I_1 + I_2 + I_3 \quad (2.26)$$

e)

$$I_4 + I_5 \quad (2.27)$$

- 2) Construct the circuit shown in figure 2.5. Measure the current and voltage using both methods.

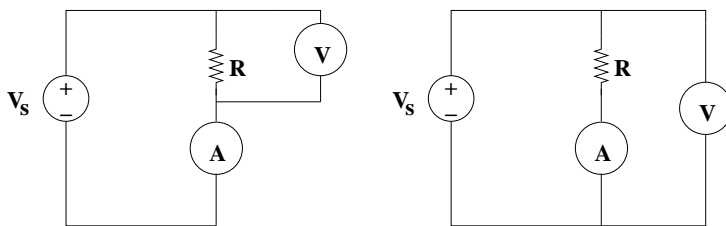


Figure 2.5: Resistor Network.

- i) If $R = 10 \text{ k}\Omega$, the ammeter reads $200 \mu\text{A}$, and the DMM reading is V , which configuration will provide a more accurate value of R if you divide V by $200 \mu\text{A}$? (hint: use the models of the voltmeter and the ammeter).
- ii) Suppose $R = 1.2 \text{ M}\Omega$ and the ammeter reads $4 \mu\text{A}$?
- 3) Measure the I-V characteristic of a resistor.
- i) a) Measure the resistance of a 1 Watt, 100Ω resistor.
- b) Calculate the maximum voltage and current rating for the resistor.
- c) Determine the best way to set up the circuit.

- ii) a) Use the ammeter in the $200 \mu A$ range. Set the current at equidistant points of 0, 40, 80, 120, 160 & $200 \mu A$. Measure the voltage in each case. Record the data in a table. Plot the points on a graph while you are taking the data.
- b) Repeat these measurements on two other current scales. Remember not to exceed the maximum ratings for voltage and current. By shifting the origin of the vertical axis all of the points can be plotted on the same sheet of graph paper.
- c) For each current scale draw the best straight line through the points. Compare the value of R from the slope of the graph to the actual measurement.
- iii) a) Measure the resistance of an $8.2 k\Omega$ resistor. Find R from measurements of the V-I characteristic; use the $2 mA$ scale.
- b) Measure the resistance of a $680 k\Omega$ resistor. Find R from measurements of the V-I characteristic; use the $200 \mu A$ scale.
- 4) Find the power with a 10 volt DC input. Find the power for a 14.4 volt, 1 kHz sine wave. Use a 1 Watt, $10 k\Omega$ resistor.

2.4 Problems

- 1) Assume that you have 3 resistors; two parallel resistors are placed in series with the third resistor as shown in figure 1.8 (middle). The voltage at the left node of R_1 is given by V_1 . Find an expression for the voltage, V_2 , at the right node of R_1 (e.g. the node which has R_1 , R_2 & R_3). Hint: this will be expressed in terms of V_1 as $V_2 = CV_1$, where C is a constant. You will need to find C .
- 2) Find C in terms of R_1 , R_2 & R_3 . Assume that $R_1 = R_2$ but $R_3 \gg R_2$; Can find an approximate expression for C and use it to find V_2 in terms of V_1 ? (Hint: find

$$\lim_{R_3 \rightarrow \infty} C$$

(2.28)

in terms of R_1 & R_2 .

- 3) Reuse the original expression (not the approximation) for C in terms of R_1 , R_2 & R_3 . Assume that $R_2 = R_3$ but now $R_1 \gg R_2$; Find an approximation for C and V_2 in terms of V_1 .

- 4) Repeat the problem again, this time assume that both R_2 & R_3 are $\gg R_1$. Find an approximation for C and V_2 in terms of V_1 .

Chapter 3

Superposition Principle and Thevenin Equivalent Circuits

Lab Objectives:

- §1. Apply the Superposition Theorem to DC circuits.
- §2. Analyze circuits using Thevenin Equivalents.
- §3. Apply Thevenin's Theorem to DC and AC circuits.
- §4. Demonstrate Maximum Power Transfer.

Pre Lab:

1. Read this lab.
2. Derrive the solutions prior to lab.

3.1 Introduction

The Superposition, Thevenin and Norton theorems are represent concepts which are of great fundamental importance in applied in circuit analysis and measurements. The Superposition principle allows one to examine separate contributions from independent sources; this is most helpful in the analysis of transistor and amplifier circuits. Thevenin's and Norton's Theorems allow one to simplify the analysis of circuits. They provide a general approach to determine the resistance, voltage and current "seen" at an arbitrary point within a linear circuit. These theorems can each be applied to simple measurements in order to understand their utility.

3.2 Concepts & Information

- a) Independent and Dependent Sources[1]

An independent voltage source provides a specified voltage between its terminals. The current through the device will not change the output voltage. An independent current source will provide a specified current between its terminals; the voltage across the terminals will not change the current. This is illustrated in figure 3.1.

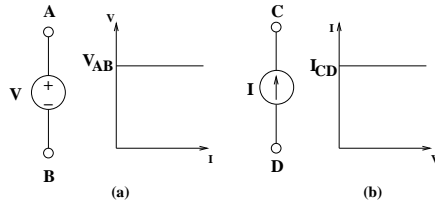


Figure 3.1: Independent voltage and current source.

A dependent source will generate a voltage or current which is a function of a voltage or current elsewhere in the circuit. There are 4 types of controlled sources: (a) the voltage-controlled voltage source, (b) the voltage-controlled current source, (c) the current-controlled voltage source and (d) the current-controlled current source. This is illustrated in figure 3.2.

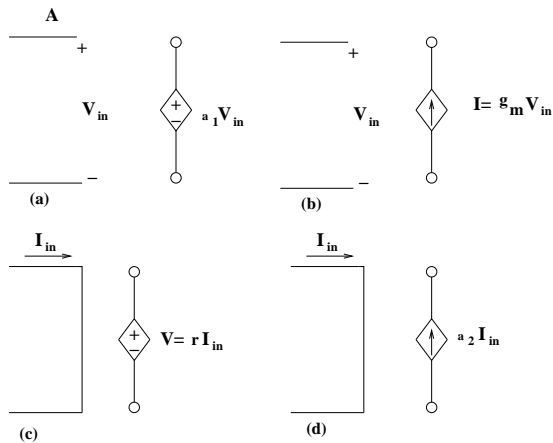


Figure 3.2: Dependent sources.

b) Superposition Theorem[1]

The Superposition Principle states: In any linear circuit containing multiple, independent sources, the current and voltage at any point in the circuit can be found from the algebraic sum of the individual contributions from each source acting alone.

To illustrate the superposition principle we will use the circuit in figure 3.3. Notice, it has both an independent voltage source and an independent current source. If we want to find the voltage, V_{AB} across R_L by applying the

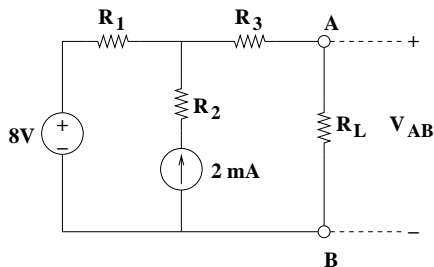


Figure 3.3: Circuit to illustrate superposition principle.

superposition principle, we need to find two voltages, $V_{AB}^{(1)}$ & $V_{AB}^{(2)}$. The first voltage, $V_{AB}^{(1)}$, is found by removing the independent voltage source. This is accomplished by replacing the independent voltage source with a short circuit. The second voltage, $V_{AB}^{(2)}$, is found by replacing the independent current source with an open circuit. This is illustrated in figure 3.4.

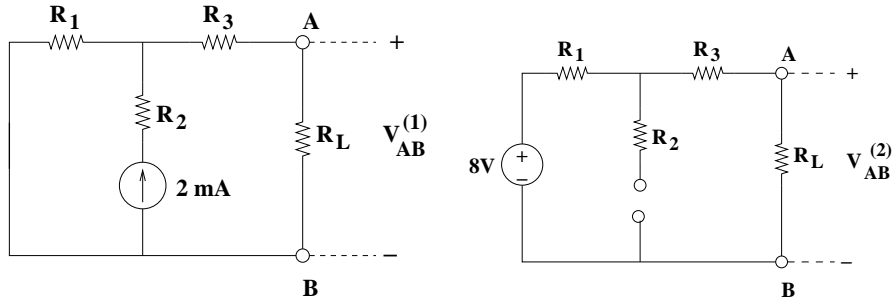


Figure 3.4: Circuits for $V_{AB}^{(1)}$ & $V_{AB}^{(2)}$.

The original voltage across R_L , V_{AB} , is found by

$$V_{AB} = V_{AB}^{(1)} + V_{AB}^{(2)} \quad (3.1)$$

or the sum of the voltages found from the sub circuits formed when the independent sources were removed. This can be carried out for circuits with many independent voltage and/or independent current sources.

c) Thevenin's Theorem[1]

If we have a network subdivided into two parts: A and B as shown in figure 3.5, where network A contains only linear elements, Thevenin's Theorem states that network A can be replaced by an independent voltage source and a series resistor. The voltage, V_{th} is the voltage, V_{AB} or V_{oc} , if network B is replaced by an open circuit.

The circuit replacing network A leaves the resulting voltage across the terminals V_{AB} unchanged. Thus, any linear circuit can be replaced by an independent voltage source and a series resistor.

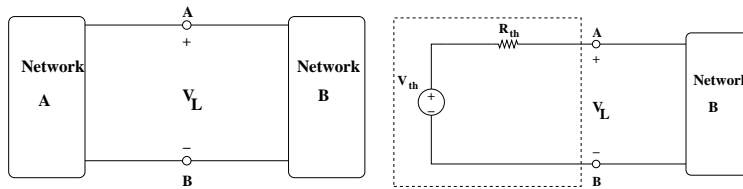


Figure 3.5: Circuits to illustrate Thevenin's Theorem.

Finding the V_{th} and R_{th} can be accomplished by one of several methods. If the circuit is unknown as shown in figure 3.6 the open circuit voltage, V_{oc} , that is the voltage when terminals A and B are open, and the short circuit current I_{sc} , when terminals A and B are shorted, can be measured in order to find R_{th} .

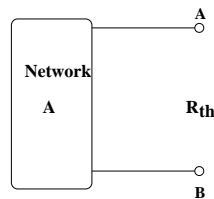


Figure 3.6: Finding R_{th} .

The expression,

$$R_{th} = \frac{V_{oc}}{I_{sc}} \quad (3.2)$$

can be used to find R_{th} from measuring the open circuit voltage and the short circuit current. Thus, $V_{oc} = V_{th}$. If the circuit in network A is known, R_{th} can be found directly by: (a) replacing ALL independent voltage sources with short circuits, (b) replacing ALL independent current sources with open circuits and (c) finding the equivalent resistance.

d) Norton's Theorem[1]

The Norton Equivalent circuit is found by replacing the the Thevenin Equivalent circuit with a current source and a parallel resistor, R_N , as shown in figure 3.7.

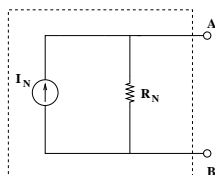


Figure 3.7: The Norton Equivalent Circuit.

Thus, The current source, I_N , is equal to $\frac{V_{th}}{R_{th}}$ and R_N is R_{th} .

e) Maximum Power Transfer[2, 1]

The Maximum Power Transfer Theorem can be found by using Thevenin's Theorem. This is illustrated in figure 3.8. The problem is to find the value of R_L will result provide the most efficient power transfer into R_L . The load resistor can represent the input resistance of another network. The maxi-

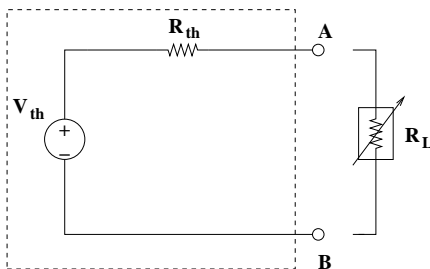


Figure 3.8: Thevenin Equivalent Circuit.

imum power is transferred from a linear, DC, circuit if the loads are matched, (e.g. $R_L = R_{th}$. The power transferred is equal to $\frac{(\frac{V_{th}}{2})^2}{R_{th}}$ or $\frac{V_{th}^2}{4R}$.

3.3 Lab Instructions

1) Construct the circuit shown in figure 3.9.

Measure V_L as a function of R_L . Select the following values for R_L : 10Ω , 20Ω , 40Ω , 80Ω , 100Ω , 200Ω , 400Ω , 800Ω , $1k\Omega$, $2k\Omega$, $4k\Omega$, $8k\Omega$, $10k\Omega$, $20k\Omega$,

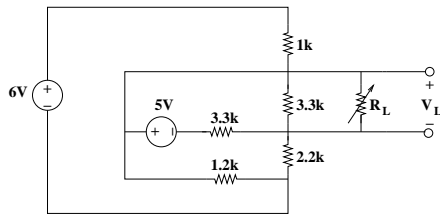


Figure 3.9: Network for Superposition Measurements.

$40k\Omega$, $80k\Omega$, & $100k\Omega$.

- i) Perform the measurement on the circuit shown in figure 3.10 with V_{L1} as function of R_{L1} using the values listed above for R_L .

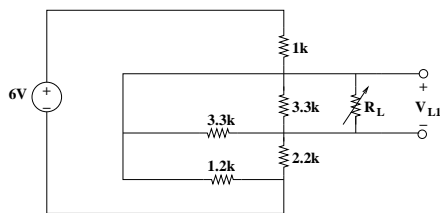


Figure 3.10: Network for Superposition Measurements.

- ii) Perform the measurement on the circuit shown in figure 3.11 with V_{L2} as function of R_{L2} using the values listed above for R_L .

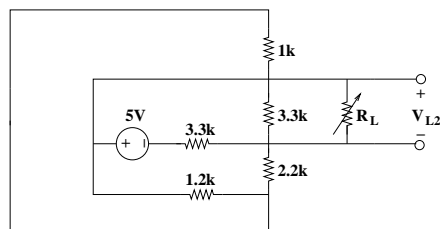


Figure 3.11: Network for Superposition Measurements.

- iii) Perform the measurement on the circuit shown in figure 3.9 with V_L as function of R_L using the values listed above for R_L .

- iv) Predict V_L as a function of the 6 volt source. Change the voltage of the 6 volt source to: 12 Volts, 14 Volts, 16 Volts 18 Volts & 20 Volts.
- v) Summarize the results from parts (a), (b) & (c) in a table. Plot the data from parts (a), (b) & (c) on a log-log scale.
- vi) Summarize the results from part (d) in a table with columns for: R_L , V_{L1} , V_{L2} & V_L ; where is V_{L1} the voltage across R_L the with the 5 Volt source removed and V_{L2} is the voltage across R_L the with the 6 Volt source removed.

2) Thevenin Equivalent Circuit: Problem 1.

- i) Construct the circuit shown in figure 3.12.

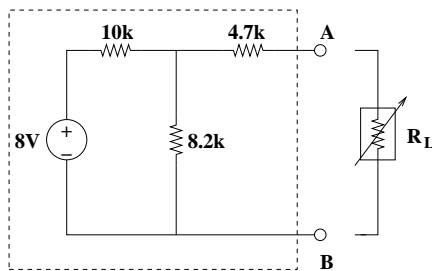


Figure 3.12: First Network to Estimate R_{th} .

- ii) Measure each resistor value.
- iii) Compute V_{oc} , I_{sc} & R_{th} using the circuit in figure 3.12 with the MEASURED resistor values. Compare $\frac{V_{oc}}{I_{sc}}$ with R_{th} .
- iv) Measure V_{oc} & I_{sc} . Compute R_{th} .
- v) Measure V_{ab} as a function of R_L . Vary R_L as follows: 500Ω , $1k\Omega$, $2k\Omega$, $4k\Omega$, $8k\Omega$, $10k\Omega$, $20k\Omega$, $40k\Omega$, $80k\Omega$, $100k\Omega$, $200k\Omega$ & $500k\Omega$.
- vi) Make a table with 4 columns: R_L , V_L (measured), V_L (calculated), & P_L .
- vii) Plot V_L and P_L as a function of R_L .

30 CHAPTER 3. SUPERPOSITION PRINCIPLE AND THEVENIN EQUIVALENT CIRCUITS

3) Thevenin Equivalent Circuit: Problem 2.

- i) Construct the circuit shown in figure 3.13.

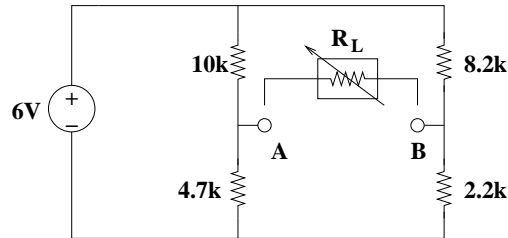


Figure 3.13: Second Network to Estimate R_{th} .

- ii) Measure each resistor value.
- iii) Measure V_{oc} & I_{sc} . Compute R_{th} .
- iv) Measure V_{ab} as a function of R_L . Vary R_L as follows: 100Ω , 200Ω , 400Ω , 800Ω , $1k\Omega$, $2k\Omega$, $4k\Omega$, $8k\Omega$, $10k\Omega$, $20k\Omega$, $40k\Omega$, $80k\Omega$ & $100k\Omega$.
- v) Make a table with 4 columns: R_L , V_L (measured), V_L (calculated), & P_L .
- vi) Plot V_L and P_L as a function of R_L .
- 4) Thevenin Equivalent Circuit: Problem 3.

Estimate R_{th} for the function generator. Use a 10 kHz , 10 Volt_{peak} sine wave with 0 Volt offset.

- 1) Vary R_L as follows: 50Ω , 100Ω , 200Ω , 500Ω , $1k\Omega$, $2k\Omega$, $5k\Omega$, $10k\Omega$, $20k\Omega$, $50k\Omega$, $100k\Omega$, $200k\Omega$, $500k\Omega$ & $1\text{ M}\Omega$.
- 2) Make a table with 4 columns: R_L , V_L (measured), & P_L .
- 3) Plot V_L and P_L as a function of R_L .

3.4 Problems

1) The circuit in figure 3.14 is known as the Wheatstone Bridge. It is quite similar to one of the circuits measured in the lab exercises. This Wheatstone bridge can be used to measure the value of a resistor by:

- i) Placing the unknown resistor, R_x , into the bridge.
- ii) Adjusting R_3 until the current flowing through the meter is zero.
- iii) Find R_x by:

$$R_x = \frac{R_2 R_3}{R_1} \quad (3.3)$$

Show that this expression can be found by algebraically by assuming that $V_A = V_B$.

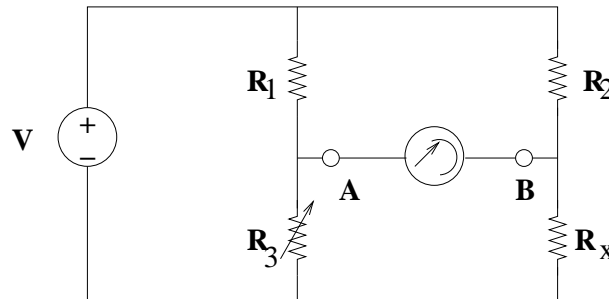


Figure 3.14: Wheatstone Bridge.

References

- [1] M. Van Valkenburg, *Network Analysis*. Prentice Hall, 1974.
- [2] A. J. Diefenderfer, *Principles of Electronic Instrumentation*. W.B. Sanders, 1972.

Chapter 4

RC Circuits

Lab Objectives:

- §1. Understand properties of capacitors in DC & AC Circuits.
- §2. Measure RC time constants.
- §3. Learn about Bode Plots.
- §4. Understand amplitude and phase relationships.
- §5. Analyze and measure the frequency and phase response of Low Pass and High Pass Filters.

Pre Lab:

1. Read this lab.
2. Find the time constants prior to the lab (part a).
3. Find filter cutoffs prior to the lab (part b).

4.1 Introduction

There are many applications for capacitors in DC circuits and AC circuits. Some examples applications are: power supply smoothing, bypassing AC components on DC voltage supplies, low pass filters, high pass filters, band pass filters, band stop filters and notch filters. Capacitance is also found in the analysis of circuit boards, transmission lines, antennas, digital circuits and integrated circuits.

This lab will be broken into two parts: part (a), DC measurements, namely the time constant measurement and part (b), Low Pass and High Pass filters. Students will use the oscilloscope and function generator to measure the RC circuits built for the lab exercises.

4.2 Background

Capacitors store energy from the electric field using geometric placement of charged conductors. Capacitance is found whenever an electric field is formed between two conductors of opposite charge are separated by a short distance. The fundamental relationship defining capacitance is given as[1]

$$Q = CV \quad (4.1)$$

where Q , is the charge in Coulombs, V is the voltage in Volts and C is the Capacitance in Farads. Current is defined by the following expression[1]

$$I = \frac{dQ}{dt} \quad (4.2)$$

where I is the Current in Amps, Q is again the charge in Coulomb and t is time.

The current in a capacitor can be found by combining equations (4.1) and (4.2) yielding[1],

$$I = \frac{d[CV]}{dt} \quad (4.3)$$

For a constant capacitance (with respect to time), equation (6.9) becomes

$$I = C \frac{dV}{dt} \quad (4.4)$$

Equation (4.33) relates the current and voltage in a capacitor (where the capacitance is constant with respect to time). In this case we find that no current flows if the voltage, V , is constant with respect to time. Thus, a capacitor with a constant capacitance blocks a DC voltage. For example, if we combine a time varying signal, $V(t)$, with a DC component, V_{dc} , then the resulting voltage is given by

$$V_{in} = V_{dc} + V(t) \quad (4.5)$$

If this voltage is applied to the capacitor in the circuit shown in figure 4.1, current flow is due only to the time varying voltage component.

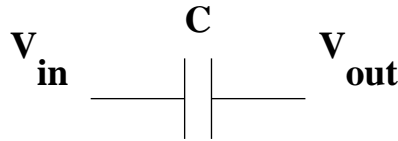


Figure 4.1: Blocking Capacitor.

Using superposition we can separate the independent DC and AC voltage sources. We then find that

$$I = C \frac{dV_{in}}{dt} = C \frac{dV_{dc}^0}{dt} + C \frac{dV(t)}{dt} = C \frac{dV(t)}{dt} \quad (4.6)$$

and the DC component is removed.

Capacitance is found using by combining the relations above with the geometry of the conductors. One could say that capacitance is a function of geometry. Gauss' Law and the Electric Field are used in most cases. Recall that the Electric field is related to force by

$$E = QF \quad (4.7)$$

where F is the force (in Newtons), E is the Electric Field (with units of Newtons/Coulomb or Volts/meter) and Q is the charge.

You should also recall Gauss' Law[1] which states the the electric field contained within a particular area is equal to the charge enclosed divided by ϵ_o ,

$$\phi_{net} = \oint_S \mathbf{E} \cdot \hat{\mathbf{n}} dA = \frac{Q_{inside}}{\epsilon_o} \quad (4.8)$$

where ϕ_{net} is the net electrostatic flux and ϵ_o is the permittivity of free space, equal to $8.85 \times 10^{-14} F/cm$.

There are two items worth noting which result from Gauss' Law: (1) if there is no charge *within* a metal enclosure, then the inside of the metal enclosure is shielded from the electric field, and (2) the flux lines are normal to the surface. Results are quite easy to find for basic geometries.

The energy stored in a capacitor can be expressed as[1]:

$$U = \frac{1}{2} CV^2 \quad (4.9)$$

where U is the potential energy (in Joules), C is the capacitance and V is the voltage.

4.2.1 Parallel Plate Capacitor

The parallel plate capacitor, shown in figure 4.2, serves as one of the simplest examples of a capacitor. It consists of 2 conductive plates, each with an area, A , separated by a distance, d . Charging the plates with a battery places positive charge on one plate and negative charge on the other plate.

The schematic in figure 4.3 illustrates the electric field in the capacitor in figure 4.2. Although there is some fringing near the ends this can be ignored.

The relationship between voltage and the electric field for the parallel plate capacitor in figure 4.2 is given by[1]

$$V = Ed \quad (4.10)$$

where V is the voltage and d is the separation between the conductors.

From Gauss' Law, we find that[1]

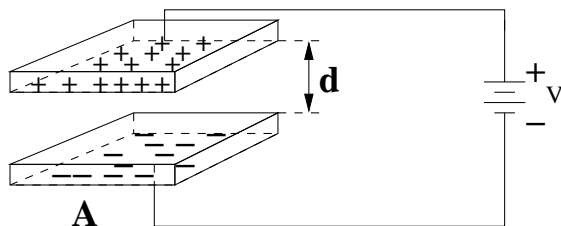


Figure 4.2: Illustration of a parallel plate capacitor.

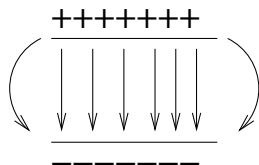


Figure 4.3: Electric Field in a parallel plate capacitor.

$$E = \frac{Q}{\epsilon_o A} \quad (4.11)$$

where ϵ_o is the permittivity of free space, equal to $8.85 \times 10^{-14} F/cm^2$.

The capacitance of a parallel plate capacitor can readily be found by combining equations (4.1), (4.10) and (4.11), leaving us with the following expression[1]:

$$C = \frac{\epsilon_{Di} \epsilon_o A}{d} \quad (4.12)$$

where ϵ_{Di} is the dielectric constant for an insulator other than air.

4.2.2 Cylindrical Capacitor

The cylindrical capacitor is constructed from two concentric cylinders as shown in figure 4.4 (where the inner cylinder is a solid wire). The cylinders are charged to different potentials. This leaves one cylinder with positive charge and one with negative charge.

The electric field from a single cylinder is a function of the distance normal to the surface of the cylinder. The electric field for the capacitor in figure 4.4 is restricted to the region between the cylinders. Thus, the total capacitance (ignoring the fringing near the ends) is[1]

$$C = \frac{2\pi\epsilon_{Di}\epsilon_o L}{\ln(\frac{b}{a})} \quad (4.13)$$

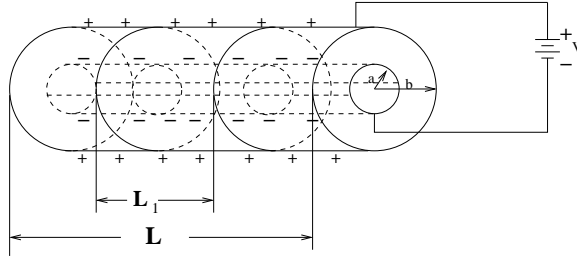


Figure 4.4: Illustration of a parallel plate capacitor.

where L is the length, b is the outer radius and a is the interior radius.

4.2.3 Parallel & Series Combinations

The equivalent capacitance from a parallel combination of n capacitors is the sum of the capacitors and is expressed as,

$$C_{eff} = C_1 + C_2 + \dots + C_n \tag{4.14}$$

The series combination of capacitors yields an effective capacitance given by,

$$\frac{1}{C_{eff}} = \frac{1}{C_1} + \frac{1}{C_2} + \dots + \frac{1}{C_n} \tag{4.15}$$

The capacitive voltage divider is shown in figure 4.5. Again, we wish to find the relationship between V_2 & V_1 (assuming that V_1 is the input voltage & V_2 is the output voltage).

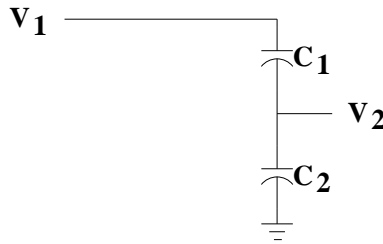


Figure 4.5: Charging cap.

The relationship between V_2 & V_1 is given by

$$\frac{V_2}{V_1} = \frac{C_1}{C_1 + C_2} \tag{4.16}$$

where C_1 , C_2 , V_1 & V_2 are all identified in figure 4.5.

Thus, the effective capacitance is just the opposite of what is seen with resistors. Can you show that these relationships are in fact true ?? hint: start with 2 capacitors and use equation (4.1).

4.3 Time Constants

4.3.1 Charging the Capacitor

If an uncharged capacitor is charged from a battery as shown in figure 4.6 there is an exponential relationship which governs the the voltage across the capacitor as a function of time.

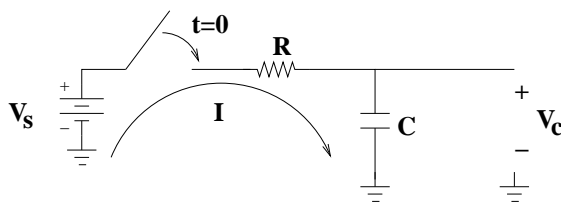


Figure 4.6: Charging cap.

Using Kirchhoff's current law at the output node, the current in the resistor and the capacitor must add to zero as shown in equation (4.17).

$$I_{res} - I_{cap} = 0 \quad (4.17)$$

The actual differential equation can be found by merely substituting the expressions for current in the resistor and the capacitor into equation (4.17). This results in,

$$\frac{V_s - V_c}{R} - C \frac{dV_c}{dt} = 0 \quad (4.18)$$

where V_s is the voltage source and V_c is the voltage across the capacitor. Solving the differential equation yields[1]:

$$V_c = V_s[1 - e^{-\frac{t}{\tau}}] \quad (4.19)$$

where $\tau = RC$. The RC product is given as the time constant. Note that when $t = \tau$, V_c is 63.2% of V_s .

This result is shown graphically in figure 4.7. The voltage response is given relative to V_s and the horizontal axis is scaled to represent the number of time constants[1, 2].

The relative voltage level for a single time constant is also shown.

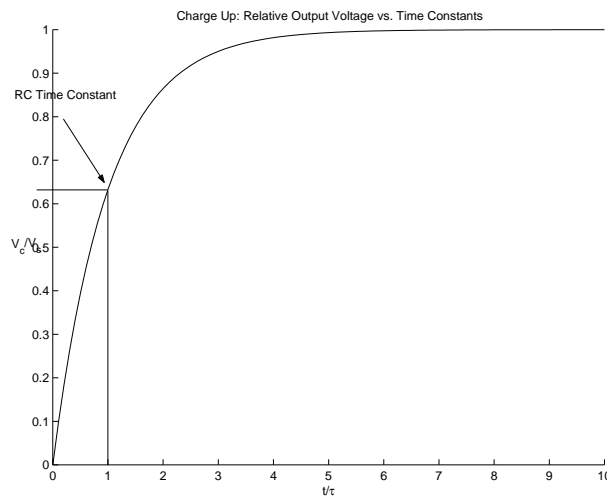


Figure 4.7: Relative voltage as a function of time constants during charge up.

4.3.2 Dis-charging the Capacitor

Assuming that the capacitor is charged to V_s , discharging a capacitor as shown in figure 4.8 can be found from a similar differential equation.

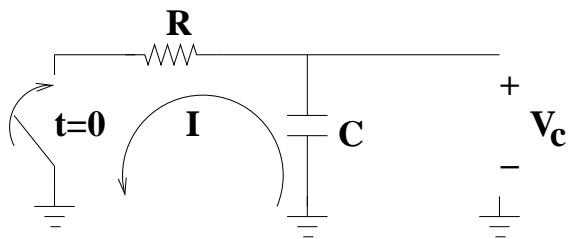


Figure 4.8: Cap Discharge.

This time, the current for the capacitor and the resistor add at the node as follows[1, 2]:

$$\frac{V_s}{R} + C \frac{dV_c}{dt} = 0 \quad (4.20)$$

This yields the final expression for discharging a capacitor[1, 2]:

$$V_c = V_s e^{-\frac{t}{\tau}} \quad (4.21)$$

Again, the time constant is given by $\tau = RC$. When the capacitor discharges, the value of V_c after one time constant is 36.8% of V_s .

This is shown graphically in figure 4.9. The voltage response is given relative to V_s and the horizontal axis is scaled to represent the number of time constants.

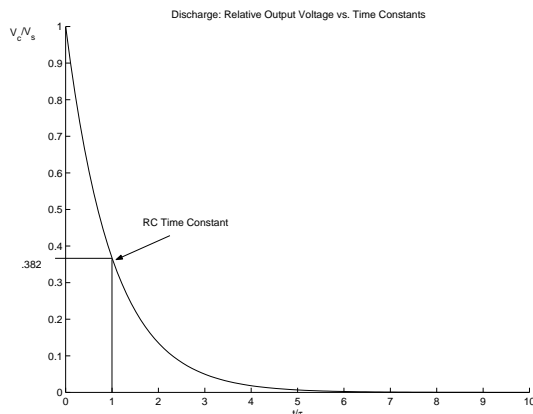


Figure 4.9: Relative voltage as a function of time constants for the discharging of the capacitor.

The relative voltage level for a single time constant is also shown.

4.4 Measurement Setup for RC Time Constants

Equations were derived in the previous section to help us understand the charging and discharging of the capacitor. Unfortunately, we cannot directly implement the circuits shown in figures 4.6 and 4.8.

In our laboratory, we have analog oscilloscopes. Thus, observing a single event in time will not work well because analog oscilloscopes have NO MEMORY !!! This means that the switches which appear in the schematics must be replaced with a suitable periodic waveform in order to simulate the opening and closing of a switch. The square wave is the obvious choice. If someone were to repeat the opening and closing of a switch between a DC voltage and ground and this was observed on an oscilloscope, this would result in a square wave. Let's assume that the peak-to-peak amplitude of the square wave is 10 volts.

Notice that the square wave will charge up to 10 volts and down to zero volts. Thus, the capacitor can be charged and discharged using a square wave. If the switch does not remain closed for a period of time sufficient to fully charge the capacitor, then equation (4.18) will not hold. The initial conditions assume that the capacitor is fully discharged prior to charging it. Conversely, the same is true for discharging the capacitor. If one does not leave the switch closed for a sufficient amount of time then the capacitor will not fully discharge. This is equivalent to opening and closing the switch too fast for a given time constant which means that the selection of the square wave frequency is also important.

In order to guarantee a reliable measurement of the time constant, we must be able to observe the quality of the input square wave. Thus it is very important to know that the square wave is accurately shaped.

4.4.1 Pre-Lab

Compute the rising falling time constants for EVERY RC combination IN ADVANCE. Just have them on a sheet readily accessible. Then you can focus on your measurement set up the circuit.

4.4.2 Obtaining Good Measurements

- i) The Oscilloscope:
 - a) Hookups: Connect channel 1 to the square wave input. You will need to set the trigger on channel 1. Connect channel 2 for the to the capacitor output. This allows you to observe both the V_s and V_c simultaneously. The measurement set up is shown illustrated in figure 4.10.

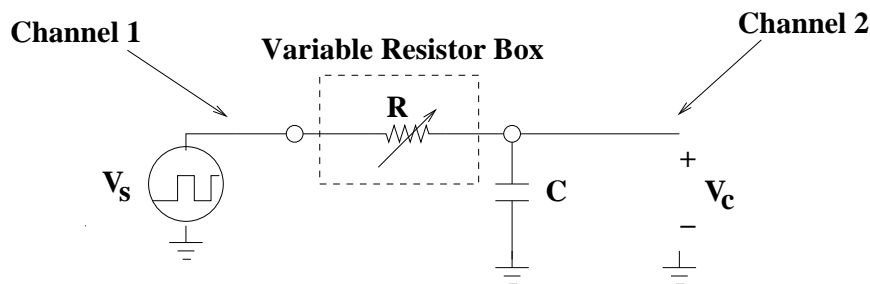


Figure 4.10: Measurement set up for time constant.

- b) Input Coupling: You should also realize the difference between the AC and DC input coupling. If you select AC, a blocking capacitor is placed in the direct path prior to the circuitry inside the oscilloscope. Thus, DC components of the signal are removed. In this case, you need to make a voltage measurement. DC coupling is a good choice for this lab.
- c) Probes: Use the scope probes for the oscilloscope. Otherwise, the connection to the oscilloscope will become part of the circuit under test. This will skew the resulting measurement.
- d) Input Coupling: Use the time base and count the boxes and tic-marks to estimate the time constant prior to using the cursors. Remember, the time base displays the time/div (e.g. the BOX, not the tic marks).

ii) The Signal Generator:

Use a frequency (e.g. a period) that allows at least 5 time constants. Since you know the time constants in advance ... you will need to switch the signal generator to the appropriate square wave.

4.4.3 Test Your Set Up PRIOR to Recording Data

For the measurement, overlay channel 1 on top of channel 2. Channel 1 is the triggering event, thus you will measure the time from the point where channel 1 changes to the point where $V_c = 6.32$ V (or $V_c = 3.68$ V; for the falling time). Use the cursors on the scope.

Once it is set up "play" with the set up a bit. For a given time constant, make the signal generator frequency too high ... you should see something that resembles "shark fins" or a "saw tooth" pattern. This means that there is not enough time provided for the cap to fully charge discharge. Thus, the values of 63.2% and 36.8% from equations (4.19) and (4.21) no longer predict correct time constants because the assumptions were based on the capacitor being allowed to fully charge and discharge. Lower the square wave frequency ... adjust the time base in order to get the portion of the wave form you wish to measure into the center of the screen ... expand it. Get comfortable understanding what the resulting display means.

Check the frequency and period of channel 1 with the scope ... does it accurately represent the selected frequency ??? if not, figure out why and fix it.

Also, be sure that you have a nice, clean square wave. If the input looks like the waveform is "bending" or very different from the square wave then stop fix it before moving on.

4.5 Integration & Differentiation

The differential relation between voltage and current in the capacitor leads to circuits which can perform waveform integration and waveform differentiation. Thus, the waveform from a signal generator can be integrated or differentiated under certain conditions. This is quite useful for a practicing engineer and it is related to RC filters described in the sections which follow.

4.5.1 Integration

The voltage across the capacitor can be thought of as the integral of the current. This can be seen by re-writing equation (4.33) as (assuming C is constant with respect to time)

$$V_c = \frac{1}{C} \int I dt \quad (4.22)$$

This means that the capacitor acts as a pure integrator. Pure integrators are not physically realizable because the gain will grow unbounded as the frequency approaches DC. The voltage across the capacitor, V_c , in figure 4.11 will act as an integrator on V_{in} when the period of the input signal is much smaller than the RC time constant. Notice that in order to observe this V_{in} is measured via channel 1 of the oscilloscope and V_c is measured by channel 2 of the oscilloscope. This allows us to view V_c and V_{in} simultaneously in order to verify that the voltage across the capacitor is indeed the integral of the signal from the signal generator.

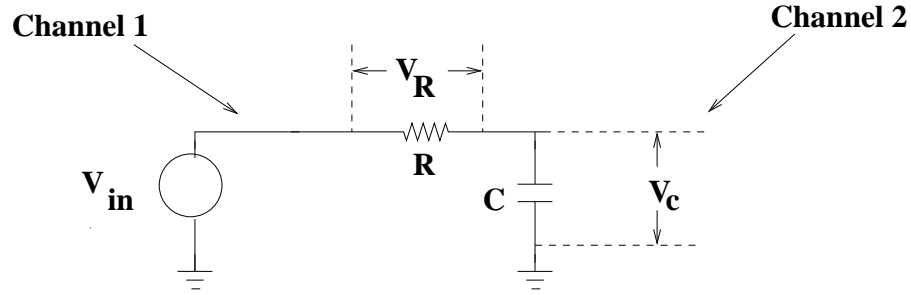


Figure 4.11: RC circuit with voltage measured across the capacitor.

The time series response of the circuit in figure 4.11 can be expressed using KVL as

$$v_{in}(t) = v_R + v_c = iR + \frac{1}{C} \int idt \quad (4.23)$$

where v_R is the voltage drop across the resistor and v_c is the voltage drop across the capacitor ($v_o = v_c$).

If we assume that the period of the input voltage, $v_{in}(t)$, is much greater than the RC product, the amplitude of the output voltage, v_c will be rise to its maximum value. This can easily be seen for a square wave input and this will also work for other input waveforms without a loss of generality. This happens because we can assume the initial conditions are *fully discharged* prior to charging up the capacitor and *fully charged* prior to discharging the capacitor.

If the period of the input signal is made to be much smaller than the RC time constant, e.g. $\frac{1}{T_i} \gg \frac{1}{2\pi RC}$, the voltage drop across the resistor in equation (6.29) will be much larger than that of the capacitor. The voltage drop across the resistor will be approximately equal to the voltage drop across the capacitor, or,

$$v_i(t) \approx v_R \quad (4.24)$$

and

$$i \approx \frac{v_{in}(t)}{R} \quad (4.25)$$

Thus, when $2\pi RC \gg T_i$ the output voltage can be found as a function of

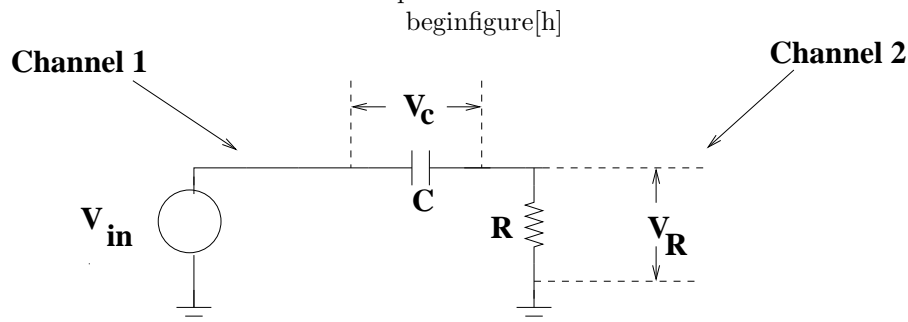
$v_i(t)$ leading to the following expression[3],

$$v_o = v_c = \frac{1}{C} \int i dt \approx \frac{1}{C} \int \frac{v_{in}(t)}{R} dt = \frac{1}{RC} \int v_{in}(t) dt \quad (4.26)$$

which tells us that the output voltage is the integral of the input voltage when the input frequency is much greater than the RC time constant. Notice that under these conditions (e.g. $\frac{1}{T_i} \gg \frac{1}{2\pi RC}$) the amplitude of V_c is much smaller than that of V_{in} . Thus, the voltage taken across the capacitor is attenuated for high frequency inputs.

4.5.2 Differentiation

The capacitor current is the derivative of the voltage across its terminals from equation (4.22). In this respect the capacitor can be made to be a pure differentiator. Pure differentiators are impractical due to infinite gain at high frequencies; however, we can realize a low frequency differentiator with an RC circuit. If we re-arrange the circuit in figure 4.11, swapping the capacitor and the resistor, the voltage across the resistor will actually be the time derivative of the input signal. This circuit is shown in figure 4.5.2 with the appropriate channels selected for an oscilloscope measurement.



RC circuit with voltage measured across the resistor.

The operation of this circuit is somewhat intuitive. A DC signal will be blocked by the capacitor. A square, which has a very high derivative when it switches (theoretically infinite), will pass when its value changes and then be blocked when it remains constant. Let us analyze more carefully:

The time series response of the circuit in figure 4.5.2 can be expressed using KVL as

$$v_{in}(t) = v_c + v_R = \frac{1}{C} \int i dt + iR \quad (4.27)$$

where v_c is the voltage drop across the capacitor and v_R is the voltage drop across the resistor ($v_R = v_c$).

If we assume that the period of the input voltage, $v_i(t)$, is *much greater* than the RC product, the amplitude of the output voltage, v_R will rise to its maximum value. In other words, for input frequencies significantly higher than $\frac{1}{2\pi RC}$ the amplitude of the output voltage is not attenuated. If the period

of the input signal is made to be *much less* than the RC time constant, e.g. $\frac{1}{T_i} \ll \frac{1}{2\pi RC}$, the voltage drop across the capacitor in equation (4.27) will be much larger than that of the resistor. The voltage drop across the capacitor will be approximately equal to the voltage drop across the resistor, or,

$$v_{in}(t) \approx v_c \quad (4.28)$$

and

$$i \approx C \frac{d}{dt}[v_{in}(t)] \quad (4.29)$$

When $2\pi RC \ll T_i$ the output voltage can be found as a function of $v_i(t)$ leading to the following expression[3],

$$v_o = v_R = iR \approx RC \frac{d}{dt}[v_i(t)] = RC \frac{dv_{in}(t)}{dt} \quad (4.30)$$

Thus, the output voltage is the derivative of the input voltage when the input frequency is much greater than the RC time constant.

4.6 Lab Instructions: Part A

Please record the resistor values for each of the lab exercises below and use the recorded resistor values in your computations.

- 1) Time Constant: For each measurement, display the voltages on the oscilloscope. Initially set the time scale so you can observe input and output waveforms. Use the trigger and appropriate time and voltage scales on the scope to measure as accurately as possible. Find the time constant, and compare with the calculated value.

Please be certain that you are NOT using electrolytic capacitors (any capacitor with polarity markings, e.g. +/-) due to safety issues.

For each time constant measurement; compute the time constant first. Then make a table with the following columns: t , V Measured and V Calculated. Use the following times: 0 - 200 μsec , in increments of 20 μsec , e.g. 0, 20, 40, ... etc.. Then plot the data points: $\ln(V_o)$ vs. t , for the discharging capacitor, $\ln(V_o - V_{in})$ vs. t for the charging capacitor.

- a) Measure the time constant for $C = 0.1\mu F$ and the following resistor values:

- i) $R = 5 k\Omega$.

- ii) $R = 1 k\Omega$,

- iii) $R = 500 \Omega$.
 - b) Measure the time constant for two $0.1 \mu F$ capacitors connected in series with $R = 5 k\Omega$.
 - c) Measure the time constant for two $0.1 \mu F$ capacitors in parallel with $R = 1k\Omega$.
- 2) Integration: Set up the circuit and oscilloscope probes as shown in figure 4.11. Use $R=1k\Omega$ and $C=.01\mu F$. Compute the RC time constant.
- Sketch the input and output waveforms observed on the oscilloscope for the following values $\frac{RC}{T} = .001, .05, .2, .75, 1$ and 2 . Make the sketches accurate with respect to relative time and amplitude. Compute expected results and overlay your measurements with sketches of the expected integrals. Do this for the following input waveforms:

i) 10 Volt Square wave.

ii) 10 Volt Sawtooth waveform.

- 3) Differentiation: Set up the circuit and oscilloscope probes as shown in figure 4.5.2. Use $R = 8.2 k\Omega$ and $C = .001 \mu F$. Compute the RC time constant.

Demonstrate the integral of a 10 volt sawtooth wave input. Use the set up in figure 4.5.2. Compute the RC time constant.

Sketch the input and output waveforms observed on the oscilloscope for the following values $\frac{RC}{T} = 10, 1, 0.33,$ and 0.1 . Make the sketches accurate with respect to relative time and amplitude. Compute expected results and overlay your measurements with sketches of the expected integrals. Do this for the following input waveforms:

i) 10 Volt Square wave.

ii) 10 Volt Sawtooth waveform.

4.7 Data to Include in Write Up

- 1) Pre-lab Time Constant Calculations.
- 2) For each time constant measurement:
 - i) Measure the resistor with an ohm meter; record the value.

- ii) Record the measurements at each time interval following the state change of the input square wave.
- iii) Compute the value at each time interval using the measured resistor value and the assumed capacitance value.
- iv) Summarize the measurements and calculated values using the table below:

<i>time</i>	V_{out}	$\ln\left(\frac{V_{out}}{V_s}\right)$	$\ln\left(\frac{V_{out}}{V_s}\right)$
<i>time</i>	V_{out}	$\ln\left(\frac{V_{out}-V_s}{V_s}\right)$	$\ln\left(\frac{V_{out}-V_s}{V_s}\right)$
[<i>secor</i> μsec]	[Volts]	[<i>Measured</i>]	[<i>Calculated</i>]

Table 4.1: Data Recorded for Time Constant Plot.

- v) Plot $\ln|measured|$ and $\ln|computed|$ values vs. *time*.
 - vi) Estimate the time constant.
- 3) Integrator & differentiator circuits; for each waveform:
- i) Provide a detailed plot on graph paper of the input and output waveform measurements for a single period recording the amplitude at appropriate time intervals (e.g. $0, \pi/4, \pi/2, 3\pi/4, \pi$, etc.
 - ii) Calculate the integral or derivative and overlay the plot of the calculated curve on top of the measured points.

When you overlay measurement values with calculated points please carefully annotate the data so that the reader can easily differentiate between a measured value and a calculated curve.

4.8 Problems

- 1) Assume that you have two capacitors in series forming a capacitive voltage divider as illustrated in figure 4.5. Find the relationship between V_2 & V_1 using equation (4.16) for the following cases:
 - a) $C_1 \gg C_2$
 - b) $C_2 \gg C_1$
- 2) Refer to figure 4.6. The rise time is defined as the time it takes for V_c to charge up from 10% of V_s to 90% of V_s (e.g. time measured from $V_c = .1V_s$ until $V_c = .9V_s$). Show that the rise time for the circuit in figure 4.6 is given by:

$$T_{rise} \approx 2.2RC \quad (4.31)$$

- a) Start by applying KCL (hint: equations (4.17) & (4.18); then adjust the limits in the integral accordingly).
 - b) Start by applying KVL
- 3) Refer to figure 4.8. The fall time is defined as the time it takes for V_c to change when discharging from 90% of V_s to 10% of V_s (e.g. time measured from $V_c = .9V_s$ until $V_c = .1V_s$). Show that the rise time for the circuit in figure 4.8 is also given by:

$$T_{fall} \approx 2.2RC \quad (4.32)$$

- a) Start by applying KCL (hint: equations (4.20); then adjust the limits in the integral accordingly).
- b) Start by applying KVL

4.9 Phase and Frequency Relations Simple RC Circuits

The voltage and current will be in phase in AC circuits constructed only from resistors. If we have a sine wave input, the voltage and current will have reach their peak and zero-crossing values at the same time. Capacitors cause a 90° phase difference between voltage and current. This means that for a sine wave voltage across a capacitor the resulting current will be shifted by $+90^\circ$. This can be illustrated using equation (6.9). If we assume that the voltage across the capacitor is $V_p \sin(\omega t)$ then, from equation (6.9) the current can be expressed as[1, 3]

$$I = \frac{d[CV]}{dt} = C \frac{d}{dt}[V_p \sin(\omega t)] = \omega C V_p \cos(\omega t) = \omega C V_p \sin(\omega t + 90^\circ) \quad (4.33)$$

where ω is the radian frequency (e.g. $2\pi f$), V_p is the peak voltage and t represents time in seconds.

For a capacitor the phase of the current leads the voltage or the voltage lags the current. Dividing the voltage by the current to find the resistance, as Ohm's Law suggests, leads to the definition of reactance, X_c , or[3]

$$X_c(\omega) = \frac{-1}{\omega C} \quad (4.34)$$

where X_c is measured in Ohm's and ω is again the radian frequency (e.g. $2\pi f$).

The impedance is defined as the generalized resistance which includes both resistance and reactance. The impedance of a series connected resistor and capacitor is given by

$$Z(\omega) = R + jX_c(\omega) \quad (4.35)$$

where Z is the impedance in Ohm's and $j = \sqrt{-1}$; the variable, (ω) , is often omitted leaving Z or X_c .

Thus, impedance is a complex quantity and the reactance is the imaginary part of the impedance. Both quantities change with frequency. The symbol j (sometimes i is used) and the “-” sign in equation (4.34) represents the -90° phase shift.

Equation (4.34) suggests an inverse frequency dependence. At DC, X_c is infinite, at low frequencies X_c is high (depending upon the values of C and ω) while at high frequencies, X_c becomes small. If the frequency is very high, X_c approaches 0 and the capacitor becomes an AC short.

4.10 The Low-Pass Filter

The RC circuit in figure 6.11 is a low pass filter. The low pass filter is a series RC section where the output voltage is the voltage across the capacitor. The capacitor provides a path to ground for AC signals while DC signals are blocked. Lower frequencies tend to be attenuated by the capacitor while higher

frequencies are shorted to ground. The lower the frequency, the less its voltage, V_c , is attenuated.

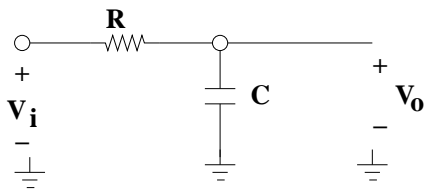


Figure 4.12: Low Pass Filter.

The output voltage (across the capacitor), $V_{out} = V_c$, has a maximum value equal to $V_{in} = V_i$. Attenuation is equivalent to a gain which is less than 1. Thus, the terms are interchangeable; here we will use attenuation. We can express the attenuation of V_{out} with respect to V_{in} as

$$A_v = \frac{V_{out}}{V_{in}} = \frac{V_o}{V_i} \quad (4.36)$$

where $A_v(\omega)$ is the attenuation of the circuit as a function of frequency.

4.10.1 Frequency & Phase Response

When attenuation or gain is a function of frequency it is referred to as the frequency response. We can find the frequency response by computing the attenuation directly from the circuit impedance if we treat reactance and resistance as we would treat ordinary resistors forming a voltage divider. This yields[3]

$$A_v(\omega) = \frac{jX_c}{R + jX_c} \quad (4.37)$$

where X_c is the reactance of the capacitor and R is the resistance of the resistor. If we substitute equation (4.34) into equation (6.33) the relation between V_{out} and V_{in} can be found as a function of R , C and ω . Equation (6.33) is reduced to[3]

$$A_v(\omega) = \frac{1}{1 + j\omega RC} = \frac{1}{1 + j\left(\frac{f}{f_o}\right)} = \frac{1 - j\left(\frac{f}{f_o}\right)}{1 + \left(\frac{f}{f_o}\right)^2} \quad (4.38)$$

where $f_o = \frac{1}{2\pi RC}$ is known as the “cutoff” frequency.

The power is expressed as the squared magnitude of $A_v(\omega)$ or

$$P_{LPF}(f) = |A_v(\omega)|^2 = A_v(\omega)A_v^*(\omega) = \frac{1}{1 + \left(\frac{f}{f_o}\right)^2} \quad (4.39)$$

where $A_v^*(\omega)$ is the complex conjugate of $A_v(\omega)$ and $P_{LPF}(f)$ is the power as a function of frequency.

The phase response is expressed as the inverse tangent of the imaginary part divided by the real part of $A_v(\omega)$. This becomes[3],

$$\phi(\omega) = -\tan^{-1}(\omega RC) = \tan^{-1}\left(\frac{1}{\omega RC}\right) - 90 = -\tan^{-1}\left(\frac{f}{f_o}\right) \quad (4.40)$$

Notice that when $f = \frac{f_o}{2\pi}$, that $P_{LPF}(f = f_o/2\pi) = P_{LPF}(f = 0)$. The attenuation in power is cut by a factor of exactly 2. The bandwidth of the filter is measured by f_o .

4.10.2 Bode Plots

The frequency response can be plotted using either the amplitude or the power (e.g. magnitude squared amplitude). It is common to use the decibel scale, where the output voltage is referenced to the input voltage as a function of frequency. The attenuation or gain, A_v , can be converted to decibels, or

$$A_v[dB] = 20\log_{10}(|A_v(f)|) = 10\log_{10}(|A_v(f)|^2) \quad (4.41)$$

where $A_v(f)$ is the attenuation or gain.

The Bode Plot is a plot of the magnitude in decibels as a function of the log frequency[3], (e.g. $A_v[dB](f)$ vs. $\log_{10}(f)$), which is shown in the top of figure 6.12. The phase response is typically plotted along with the Bode Plot using the same independent variable (log frequency)[3]. This appears in the lower half of figure 6.12.

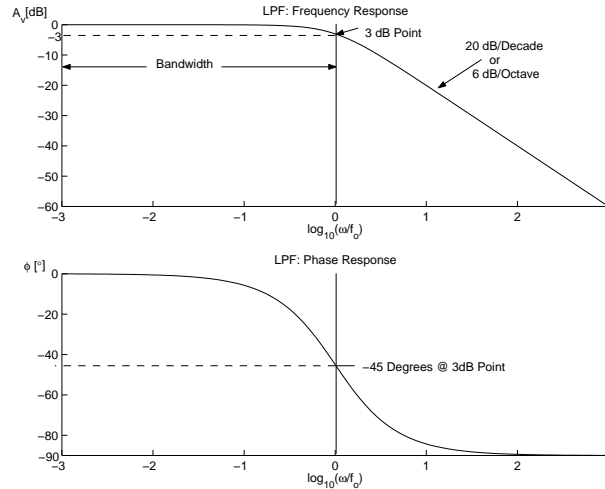


Figure 4.13: Low Pass Filter amplitude and phase response.

These plots provide a complete characterization of the RC low pass filter. The point where $f = \frac{f_o}{2\pi}$ has special significance. At this point $A_v = 3dB$ and

the phase is -45° . This is known as the *3dB point*. The *effective bandwidth* (or *bandwidth*) of the low pass filter is measured from DC to f_o . The *roll off* is defined as the slope of the frequency response of the Bode Plot in the linear part of the region where $f > f_o$. The phase response is negative; at DC, the phase is 0° and it approaches -90° as $f \rightarrow \infty$.

4.11 The High-Pass Filter

The RC circuit in figure 6.13 is a high pass filter. The high pass filter is a series RC section where the output voltage is the voltage across the resistor. The capacitor is connected in the direct path between the input and output. The resistor is connected between the output and ground. This means that DC signals will be blocked from the output while AC signals will pass to the output. In general, the higher the frequency, the lower the attenuation across V_R .

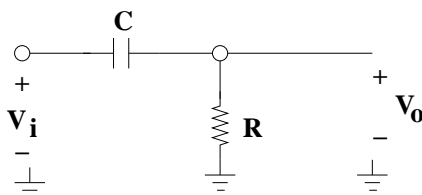


Figure 4.14: High Pass Filter.

The attenuation is identical to the expression in equation (6.28). There are differences due to the swapping of the capacitor and the resistor.

4.11.1 Frequency & Phase Response

When attenuation or gain is a function of frequency it is referred to as the frequency response. We can find the frequency response by computing the attenuation directly from the circuit impedance if we treat reactance and resistance as we would treat ordinary resistors forming a voltage divider. This yields[3]

$$A_v(\omega) = \frac{R}{R + jX_c} \quad (4.42)$$

where X_c is the reactance of the capacitor and R is the resistance of the resistor. If we substitute equation (4.34) into equation (6.41) the relation between V_o and V_i can be found as a function of R , C and ω . Equation (6.41) is reduced to

$$A_v(\omega) = \frac{j\omega RC}{1 + j\omega RC} = \frac{j\left(\frac{f}{f_o}\right)}{1 + j\left(\frac{f}{f_o}\right)} = \frac{1 + j\left(\frac{f_o}{f}\right)}{1 + \left(\frac{f_o}{f}\right)^2} \quad (4.43)$$

where $f_o = \frac{1}{2\pi RC}$ is known as the “cutoff” frequency.

The power is expressed as the squared magnitude of $A_v(\omega)$ or

$$P_{HPF}(f) = |A_v(\omega)|^2 = A_v(\omega)A_v^*(\omega) = \frac{1}{1 + \left(\frac{f_o}{f}\right)^2} \quad (4.44)$$

where $A_v^*(\omega)$ is the complex conjugate of $A_v(\omega)$ and $P_{HPF}(f)$ is the power as a function of frequency.

The phase response is expressed as the inverse tangent of the imaginary part divided by the real part of $A_v(\omega)$. This becomes[3],

$$\phi(\omega) = \tan^{-1}\left(\frac{1}{\omega RC}\right) = 90 - \tan^{-1}(\omega RC) = \tan^{-1}\left(\frac{f_o}{f}\right) \quad (4.45)$$

Once again notice that when $f = \frac{f_o}{2\pi}$, that $P_{HPF}(f = f_o/2\pi) = P_{HPF}(f = 0)$. The attenuation in power is cut by a factor of exactly 2.

4.11.2 Bode Plots

The Bode plot with a phase response is the standard method for characterizing any filter response. Equation(6.28) is used to convert A_v to dB 's.

The Bode Plot for the high pass filter is shown in the upper portion of figure 6.14. The phase response is shown in the lower portion of figure 6.14.

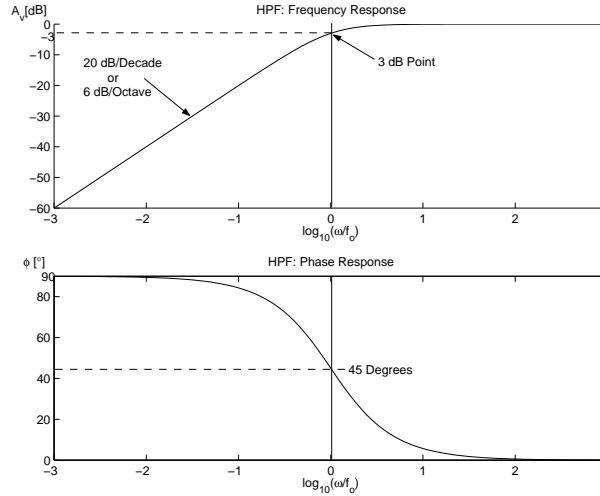


Figure 4.15: High Pass Filter amplitude and phase response.

The high pass filter frequency and phase response is analogous to that of the low pass filter. The point where $f = \frac{f_o}{2\pi}$ is referred to as the *3dB point*. The *roll off* is defined as the slope of the frequency response of the Bode Plot in the linear part of the region where $f > f_o$. The phase response is negative; at DC, the phase approaches 90° and it approaches 0° as $f \rightarrow \infty$.

4.12 Amplitude and Phase Measurements

Obviously, in order for you to successfully complete this lab you will need to measure the amplitude of the filter input and output signals. This is something you have already done in the early lab. The difficulty is making the measurement of relative phase. Signals on the oscilloscope are most conveniently represented in polar form: amplitude and relative phase. The relative phase can be found by measuring the time delay between the output signal and the input signal, $\pm\Delta t$, taking the ratio of $\pm(\frac{\Delta t}{T})$, where T is the measured period of the input sine wave. This fraction must then be converted to either degrees (multiplying by 360) or radians (multiplying by 2π) in order to compare it to the calculated phase. The \pm sign of the phase delay is determined by finding out if the output signal *lags*, ($-\Delta t$), as in the low pass filter, or *leads* ($+\Delta t$), as in the high pass filter.

The following example is provided in order to illustrate what the low pass response and the high pass response will look like on the oscilloscope. Let's use a $12\text{ k}\Omega$ resistor and a $.08\ \mu\text{F}$ capacitor; this means that $R = 12000\Omega$ and $C = 8 \times 10^{-8}\text{F}$. This leads to $f_o = 166\text{Hz}$; however, the actual value of f_o can vary by as much as $\pm 10\%$ since the relative error of the resistor and capacitor will be combined due to the RC product.

A direct measurement of f_o can be performed from the amplitude and phase measurement when $A_v = 0.71$, but you will find that each time the measurement is repeated the result will be slightly different because there is also some difficulty in determining peaks and zero crossings of the relative sine waves. A reliable means of finding the actual value of f_o is to characterize the filter by making repeated measurements of the output amplitude and time delay over a number of input frequencies while holding the amplitude of the input sine wave constant. In our example the peak-to-peak input voltage amplitude is set to be approximately 10Volts . One frequency measurement is illustrated for the low pass filter in the first example and the high pass filter in the second example:

- i) Low Pass Filter: The low pass filter set it up as shown in figure 4.16; notice that both oscilloscope channels are used. Channel 1 is applied to the input waveform and channel 2 is applied to the output signal. This enables you to simultaneously monitor the input signal and the output signal. Triggering on channel 1 is practical (when expanding the screen in order to improve accuracy) since the peak amplitude of this sine wave will be held constant.

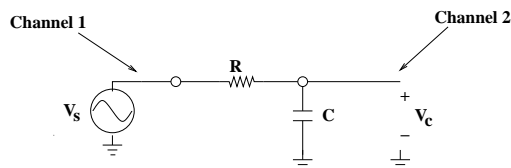


Figure 4.16: Low Pass Filter (LPF) measurement set up.

I have taken a single measurement where I have set the input frequency to 574 Hz. The output signal (channel 2) will *lag* the input signal (channel 1). This is illustrated in figure 4.17.

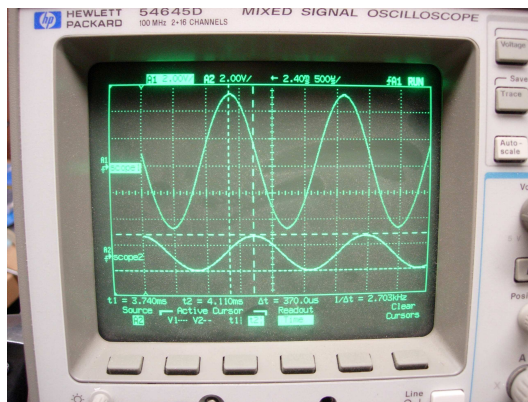


Figure 4.17: Channel 2 (bottom) *Lags* Channel 1 (top).

At each input frequency there are 3 important measurements to be taken with the oscilloscope: 1) input frequency period, T , 2) output frequency amplitude, V_{op-p} , and 3) the time delay of the lag, $-\Delta t$. This assumes that a measurement of the input signal was also made (but not repeated each time assuming the set up is not disturbed during the measurement intervals). For the example in figure 4.17, these are $T = 1.74msec$, $V_{op-p} = 2.75V$ (minus sign due to the lag) and $\Delta t = -370\mu sec$. The measurement of $V_{inp-p} = 10.12V$. Thus for the frequency of $574Hz$, $A_v = 0.2556$, $A_{dB} = -5.9dB$ and $\phi = -76.45$ deg.

- ii) The High Pass Filter: The high pass filter set it up as shown in figure 4.18; notice that channel 1 and channel 2 are arranged so that we can again simultaneously monitor both the input signal and the output signal.

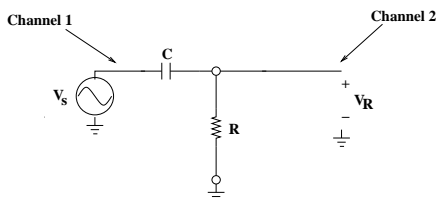


Figure 4.18: High Pass Filter (HPF) measurement set up.

I have taken a single measurement where I have set the input frequency to 101 Hz. The output signal (channel 2) will *lead* the input signal (channel 1). This is illustrated in figure 4.19.

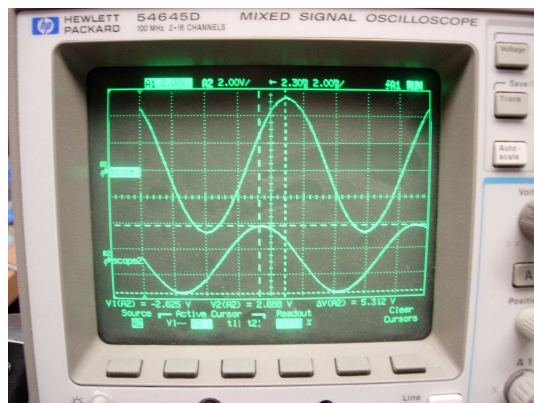


Figure 4.19: Channel 2 (bottom) *Leads* Channel 1 (top).

The procedure for the high pass filter is identical to the one just discussed for the low pass filter. Since I have used the same resistor and capacitor, the calculated value, f_o , will be identical to that of the low pass filter. For the example in figure 4.19, these are $T = 9.9msec$, $V_{op-p} = 5.312V$ (minus sign due to the lag) and $\Delta t = +1.6msec$. The measurement of $V_{inp-p} = 10.12V$. Thus for the frequency of $101Hz$, $A_v = 0.5249$, $A_{dB} = -2.80dB$ and $\phi = 58.18$ deg.

It is also possible to see both the output signal (channel 2) and the input signal (channel 1) when they are in phase (e.g. there is zero phase difference between the two signals). This can happen with the low pass filter when $f_{in} \ll f_o$ or for the high pass filter when $f_{in} \gg f_o$. In both of these cases the attenuation, although present, is negligible. In this case the input and output signals will have identical (or close to identical) amplitudes and the relative phase should also be identical. This is shown in figure 4.20: Notice that the signal displayed

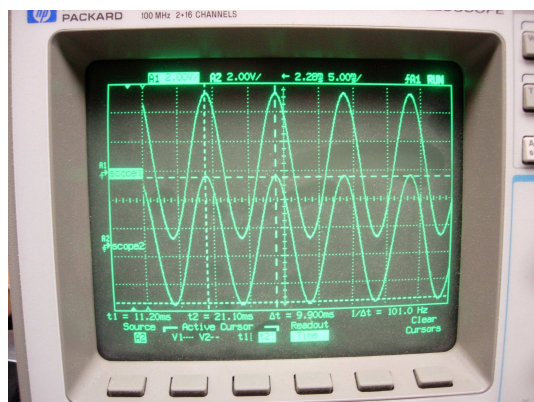


Figure 4.20: Channel 2 (bottom) & Channel 1 are in phase (top).

on channel 2 could overlay the signal on channel 1 and the resulting display would make it difficult to distinguish between the two signals.

4.13 Lab Instructions: Part B

- 1) Low Pass Filter: Construct the low pass filter and set it up as shown in figure 4.16. Use $R = 1\text{ k}\Omega$ and $C = .01\text{ }\mu\text{F}$.
 - i) Make a table with the following columns: f , V_{in} , V_{out} Measured, V_{out} Calculated, $|A_v|^2$ Measured, $|A_v|^2$ Calculated, $A_v[dB]$ Measured, $A_v[dB]$ Calculated, Δt , T Measured, ϕ Measured and ϕ Calculated.
 - ii) Measure the attenuation for the following frequencies: 10 Hz, 100 Hz, 1 kHz, 3 kHz, 10 kHz, 30 kHz and 100 kHz. Also measure at $f = f_o$. Set the peak amplitude of the input sine wave to 10 volts.
 - iii) Complete the entries in the table.
 - iv) Plot $A_v[dB]$ Measured vs. $\log_{10}(f)$ and ϕ Measured vs. $\log_{10}(f)$. Overlay each graph with a plot of $A_v[dB]$ Calculated vs. $\log_{10}(f)$ and ϕ Calculated vs. $\log_{10}(f)$.
 - v) Compute the cutoff frequency. Identify the $3dB$ point extrapolated from the measurements and overlay this with the computed value.

- 2) High Pass Filter: Construct the high pass filter and set it up as shown in figure 4.18. Use $R = 8.2\text{ k}\Omega$ and $C = .001\text{ }\mu\text{F}$.
 - i) Make a table with the following columns: f , V_{in} , V_{out} Measured, V_{out} Calculated, $|A_v|^2$ Measured, $|A_v|^2$ Calculated, $A_v[dB]$ Measured, $A_v[dB]$ Calculated, Δt , T Measured, ϕ Measured and ϕ Calculated.
 - ii) Measure the attenuation for the following frequencies: 100 Hz, 1 kHz, 10 kHz, 100 kHz and 300 kHz. Also measure at $f = f_o$. Set the peak amplitude of the input sine wave to 10 volts.
 - iii) Complete the entries in the table.

- iv) Plot $A_v[dB]$ Measured vs. $\log_{10}(f)$ and ϕ Measured vs. $\log_{10}(f)$. Overlay each graph with a plot of $A_v[dB]$ Calculated vs. $\log_{10}(f)$ and ϕ Calculated vs. $\log_{10}(f)$.
- v) Compute the cutoff frequency. Identify the $3dB$ point extrapolated from the measurements and overlay this with the computed value.

4.14 Report Format

Whenever you are characterizing a filter you will need to first plot the Bode magnitude and phase plots presented in the write up using the R and C values given for the particular lab exercise. Then you will overlay your plots with the actual measurements. There will be some variation between computed results and the actual measurements because the resistors, capacitors and inductors (later on) will be somewhat different than the value stamped on the component. This changes RC products and thus will have some impact on f_o .

Although the report format is given using Lab 4b as an example, this same approach should be applied to labs 5, 6, 7a, 7b, the lab on simple op amps and your final project. In other words, this outlines how you will report results for each filter you will characterize in this course.

4.14.1 Calculated Curves

The log-frequency axis in the write-up is scaled as $\log_{10}(f/f_o)$; however, for your report I will want your log-frequency axis to be scaled as $\log_{10}(f)$. Since you are preparing your plots on the frequency axis you will need to substitute $\omega = 2\pi(f)$ in each of the expressions used to compute the magnitude squared and the phase response.

You can use equation (4.39) to compute the magnitude of the attenuation squared for the low pass filter. You will need to substitute, which yields

$$|A_v(f)|^2 = \frac{1}{1 + (2\pi fRC)^2} = \frac{1}{1 + (\frac{f}{f_o})^2} \quad (4.46)$$

Please remember that $|A_v(f)|^2[dB] = 10\log_{10} [|A_v(f)|^2]$ You will use equation (4.40) to compute the phase response, $\phi(f)$, substituting $\omega = 2\pi(f)$, or

$$\phi(f) = -\tan^{-1} \left(\frac{f}{f_o} \right) = \tan^{-1} \left(\frac{f_o}{f} \right) - 90 \quad (4.47)$$

You will use equation (4.44) to compute the magnitude of the attenuation squared for the high pass filter, as

$$|A_v(f)|^2 = \frac{1}{1 + (\frac{1}{2\pi fRC})^2} = \frac{1}{1 + (\frac{f_o}{f})^2} \quad (4.48)$$

and you will use equation (4.45) to compute the phase response.

$$\phi(f) = \tan^{-1} \left(\frac{f_o}{f} \right) = 90 - \tan^{-1} \left(\frac{f}{f_o} \right) \quad (4.49)$$

This information needs to be summarized in an orderly fashion. This means that you will need to compute ALL of the entries listed in Table 4.14.1.

f_1	V_{in}	$ V_{out} $	$\log_{10}(f)$	$ A_v ^2$	A_v	ϕ
[kHz]	[Volts]	[Volts]	□	□	[dB]	[°]

Table 4.2: Filter Characterization: Calculated Values for Curve Sketch.

You should have ALL values in Table 4.14.1 computed for each filter to be characterized prior to your scheduled lab date. You should also have a sketch of the Bode magnitude and phase plots for each filter to be characterized.

4.14.2 Measurements

The measurements will require some additional equations. The sine-wave amplitudes are each positive real numbers. The magnitude of these ratios (e.g. A_v) and the square of this quantity (e.g. $|A_v|^2 = |\frac{V_{out}}{V_{in}}|^2$) are found by division and squaring, respectively. You will need to translate this into decibels for the magnitude plot using $Av[dB] = 10\log_{10}|\frac{V_{out}}{V_{in}}|^2$. The phase from an oscilloscope measurement is obtained by:

- 1) Measure the period of the input sine wave (e.g. T).
- 2) Measure the time difference, (e.g. Δt) between the input and output sine-waves. note that
- 3) Compute the phase by Δt can be positive or negative if the output sine-wave leads or lags the input signal phase. The phase is given by:

$$\phi = \pm 2\pi \frac{\Delta t}{T} (\text{radians}) \quad (4.50)$$

or

$$\phi = \pm 360 \frac{\Delta t}{T} (\text{degrees}) \quad (4.51)$$

The sign is dependent upon the output leading or lagging the input.

You should become familiar with the relationship between $\frac{\Delta t}{T}$ and actual phase shifts. A phase shift of $\pm 90^\circ$ implies that $\frac{\Delta t}{T} = \pm \frac{1}{4}$ and a phase shift of $\pm 180^\circ$ implies that $\frac{\Delta t}{T} = \pm \frac{1}{2}$.

You will need to supply ALL of the entries listed in Table 4.14.2. Notice, f_1 is the function generator setting that I have asked you to use. When you are measuring a filter response you must NEVER use f_1 for the measurements. Instead, you will always measure the period of the input sine-wave (e.g. T) on the oscilloscope and record the appropriate frequency in your table.

f_1 [kHz]	V_{in} [Volts]	T [μs]	ΔT [μs]	$ V_{out} $ [Volts]	$f = \frac{1}{T}$ [kHz]	$\log_{10}(f)$ []	$ A_v ^2$ []	A_v [dB]	ϕ [°]
----------------	---------------------	-------------	--------------------	------------------------	----------------------------	----------------------	-----------------	---------------	---------------

Table 4.3: Filter Characterization: Measured Values.

4.14.3 Bode Plots

For the Bode magnitude plot, you will sketch the function for each filter using the calculated $Av[dB] = 10\log_{10}(|Av|^2)$ vs. $\log_{10}(f_1)$. Then you will take the measured values of $Av[dB]$ & $\log_{10}(f)$ and plot them on top of the calculated curve. This means that the value, $\log_{10}(f)$, for each measured data point, must be overlayed on the axis $\log_{10}(f_1)$.

For the Bode phase plot, you will sketch ϕ vs. $\log_{10}(f_1)$ for each filter. Then you will overlay the calculated curve with the measured values of ϕ and plot them on top of the calculated curve. This means that the value, $\log_{10}(f)$, for each measured data point, must be overlayed on the axis $\log_{10}(f_1)$.

4.14.4 Results

The results of the measurement of a filter should include an estimate of f_o , which can be found from the measurements. The measured value of f_o should be compared to the calculated value based on the values of the selected R and C . The roll-off, that is the slope of the magnitude response in the transition region should also be estimated. To measure the roll-off, find the linear portion of the transition band ($f > f_o$ for the LPF & $f < f_o$ for the HPF). Select a frequency, call it f^1 , in this region and find the corresponding measured value of the attenuation. Select a second frequency ($f^2 = 2f^1$), find its corresponding attenuation value. Take the difference between each of the attenuation values and this will provide an estimate of the slope in $dB/Octave$. For a 1st-order LPF or a 1st-order HPF, the roll-off should be close to 6 $dB/Octave$.

4.15 Appendix

$$e^{j\phi} = \cos(\phi) + j\sin(\phi) \quad e^{-j\phi} = \cos(\phi) - j\sin(\phi)$$

$$\text{rectangular: } z = a + jb \quad \text{polar: } z = re^{j\phi}$$

$$\text{Real}\{z\} = a \quad \text{Imag}\{z\} = b$$

$$z^* = \text{conj}\{z\} = a - jb \quad \omega = 2\pi f$$

$$\cos(-x) = \cos(x) \quad \sin(-x) = -\sin(x)$$

$$X_c = -\frac{1}{\omega C} \quad X_L = \omega L$$

Rectangular to Polar Conversion:

$$r = \sqrt{a^2 + b^2}; \phi = \tan^{-1}\left(\frac{b}{a}\right)$$

Polar to Rectangular Conversion:

$$a = r\cos(\phi); b = r\sin(\phi)$$

4.16 Problems

1) Convert the following complex numbers to polar form:

- a) $3 + j4$
- b) $-22 + j30$
- c) $-j$
- d) $6.4 + j6.4$

2) Convert the following complex numbers to rectangular form:

- a) $10e^{j22^\circ}$
- b) $6.2e^{-j30^\circ}$
- c) $20e^{j90^\circ}$

3) Compute the following:

- a) Solve using rectangular co-ordinates only

$$\frac{3 + j5}{-20 + j35}$$

- b) Repeat the problem; this time convert numerator and denominator to polar co-ordinates, perform division and then convert back to rectangular co-ordinates.
- c)

$$12e^{j20^\circ} + 5e^{j30^\circ}$$

Express the answer in polar and rectangular co-ordinates

4) Reactance and resistance have identical units, Ω 's; series and parallel reactances sum just like series and parallel resistors. If you are given two capacitors, (C_1 & C_2), connected in parallel show that the equivalent capacitance is given by

$$C_{eq} = C_1 + C_2$$

Do this proof by first computing the reactance of C_1 & C_2 ; then find the equivalent reactance of the parallel connected capacitors.

Use the same approach to show that if the two capacitors are connected in series the equivalent capacitance is given by

$$\frac{1}{C_{eq}} = \frac{1}{C_1} + \frac{1}{C_2}$$

- 5) Given: $C = 10\mu F$, $f = 10kHz$
- Find: the reactance, X_c , and the impedance, Z_c , (jX_c)
 - Find: the impedance, Z , if the capacitor is connected in series with a $300\ \Omega$ resistor
 - Plot the impedance, Z , on the complex plane

References

- [1] P. A. Tipler, *College Physics*. Worth-Publishing, 1987.
- [2] M. Van Valkenburg, *Network Analysis*. Prentice Hall, 1974.
- [3] A. J. Diefenderfer, *Principles of Electronic Instrumentation*. W.B. Sanders, 1972.

Chapter 5

The Twin T Filter

Lab Objectives:

- (a) Understand properties of higher-order, simple filters.
- (b) Learn about null-placement and its value in filter design.
- (c) Understand how to intuitively analyze the Twin T filter.
- (d) Analyze and measure the frequency and phase response of the Twin Tee filter.
- (e) Understand how component matching will affect the null frequency and the depth of the null.

Pre Lab:

- (a) Read this lab.
- (b) Find the time constants prior to the lab.
- (c) Find filter null frequency and predicted frequency response prior to the lab.

5.1 Introduction

In the previous lab on RC circuits we found the frequency response of the low pass filter and the high pass filter. These filters were similar in that there was a cutoff frequency which identified the upper band edge of the low pass filter and the lower band edge of the high pass filter. The cutoff frequency was defined as the frequency where the power is $\frac{1}{2}$ of its maximum value and the voltage is $\frac{1}{\sqrt{2}}$ of its maximum value; this is also known as the *3dB* point. The *roll off* of the filter was defined as the rate at which the attenuation increases as a function of the log-frequency. For the simple first order low

pass and first order high pass filter, the roll off will be 6 dB/octave; in the transition band the attenuation will increase by 6 dB for each doubling of the frequency (e.g. \log_2). This is equivalent to 20 dB/decade; meaning that the attenuation increases by 20 dB for each order of magnitude increase in the frequency (e.g. \log_{10}). These are *standard roll-offs*.

In this section simple combinations of first order low pass sections and high pass sections will be used to construct more selective filtering. Relations between filter order, filter type and roll off will be discussed. Finally, we will examine how to increase the attenuation beyond what is possible using filters which obtain standard 1st-order roll offs. This is done by placing *nulls* or *notches* in the frequency band. We will examine a particular application of this approach where the null is used to attenuate a specific frequency. Finally, we will examine the accuracy of the null placement and attenuation when components do not precisely match. The topic for this lab is the *Twin T* filter. Prior to discussing this filter a short section will explain more complex filters with standard roll offs and simple ideas regarding null placement.

5.2 Background

The work done with simple RC circuits in the previous lab can be expanded to gain a general understanding of more complex passive filters. The 1st-order RC low pass filter (LPF) might not attenuate enough at higher frequencies.

5.2.1 Higher Order Low Pass and High Pass Filters

The *roll off* of the first order filter will decrease by 6dB/octave. This means that for each doubling of the frequency after the cutoff that the attenuation will increase by an additional 6 dB beyond what it was originally. Suppose we have a cutoff at 15 kHz and we need to attenuate the filter output by at least 30 dB at a frequency of 120 kHz ? The 1st-order LPF will attenuate the output by only 18 dB at 120 kHz since this represents no more than 3 octaves above the cutoff frequency. The roll-off, or the rate of increase in attenuation would need to be increased.

The most common solution to increase the roll-off of the LPF is to cascade it with an identical filter as shown in figure 5.1. Both the magnitude and phase responses change when compared with the first order filter.

The roll-off of the magnitude response increases from 6 dB/octave to 12 dB/octave (e.g. 40 dB/decade). The roll-off of the simple filters we have described is summarized in figure 5.2.

The order of a simple filter is equivalent to the number of reactive components in the circuit. Thus, a first-order RC filter has a single capacitor, a

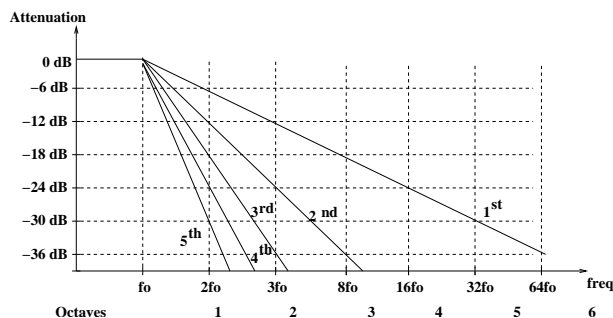
Figure 5.1: 2^{nd} -order pass filter.

Figure 5.2: Magnitude response versus LPF filter order.

2^{nd} -order RC filter has two capacitors, etc. There are other, more complex types of low pass filters. These generalizations are valid only for extensions of the simple RC filters described in the previous section.

5.2.2 Band Pass and Band Stop Filters

Suppose that we need to pass ALL signals between 15 kHz and 40 kHz only ? This means we must simultaneously attenuate signals above 40 kHz and below 15 kHz. The most common solution is to cascade a 1^{st} -order LPF with a 1^{st} -order HPF filter. This is illustrated in figure 5.3.



Figure 5.3: Band stop filter.

There is an alternative function to the filter structure in figure 5.3. Suppose that we need to attenuate ALL signals between 15 kHz and 40 kHz only ? Either selective passing or selective attenuation can be accomplished with this filter structure.

The filter in figure 5.3 can be used to pass a particular band of frequencies or it can be used to attenuate a particular band of frequencies. This depends upon the HPF and LPF cutoff frequencies. If the LPF cut-off frequency is higher than the HPF cut-off frequency, then the filter will pass a band of frequencies, hence the *band-pass filter* (BPF). If the opposite is true, then

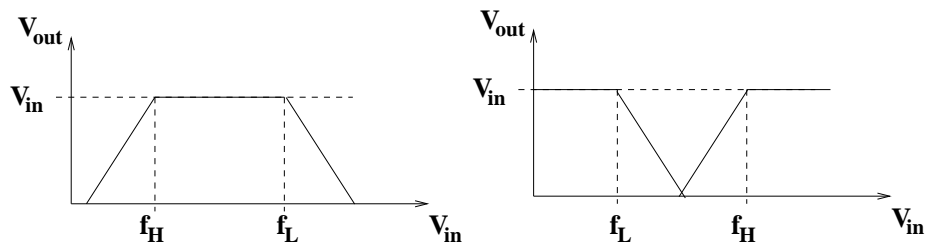


Figure 5.4: Comparison of band and pass filter and band stop filters.

the filter will attenuate a band of frequencies; this is referred to as a *band-stop* filter BSF. The frequency responses for a first-order BPF and BSF are illustrated in figure 5.4.

The first-order BPF or BSF will require at least 2 reactive components. This means it is equivalent in complexity to a 2^{nd} -order LPF or HPF. An examination of the frequency response in earlier equations indicates that the magnitude of the frequency response varies based upon the either the value of the denominator term alone or the value of the denominator with an imaginary component in the numerator. In the LPF, for example, as the frequency variable increases the denominator term increases which causes the magnitude of the frequency response to decrease in value with a limit of zero as the frequency approaches infinity. In the case of the HPF, the numerator which is purely imaginary increases in size while the denominator value also increases. The result is that the magnitude of the frequency response is zero at DC and the numerator and denominator cancel as the frequency approaches infinity. Thus, in the simple LPF and HPF the magnitude is zero at $= \infty$ or $= 0$ respectively.

5.3 Twin T Filter

The Twin T filter is the simplest example of a circuit which places a *null* (or a single frequency where the filter attenuation is very high) at a specific frequency (other than $f = 0$ or $f = \infty$ in the frequency response; the Twin T filter is an example of a *notch* filter. The roll-off of in the region of the frequency response null can be significantly higher than that of the filters described earlier. The placement of several or more nulls in the frequency response is often done in the design of more complex LPF, HPF and BPF circuits. While complex filters are beyond the scope of this book, the reason for opting for a complex filter structure over the simple filter structures discussed earlier is to improve the filter selectivity by increasing the roll off beyond the 6 dB/octave/order of the simple filter structures. The Twin T filter is shown in figure 5.5.

The frequency response of the Twin T filter can be seen intuitively by divid-

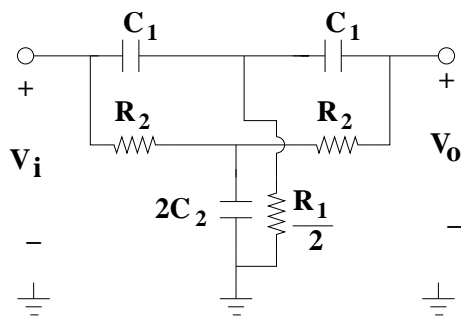


Figure 5.5: Twin T filter.

ing the schematic from figure 5.5 into two parallel filter sections, one HPF and one LPF. This is shown in figure 5.6 and it requires some explanation since we see 4 schematics. The top two figures on the left and right hand side are the low pass and high pass section taken directly from figure 5.5. The two figures below are equivalent circuits for the low pass and high pass sections.

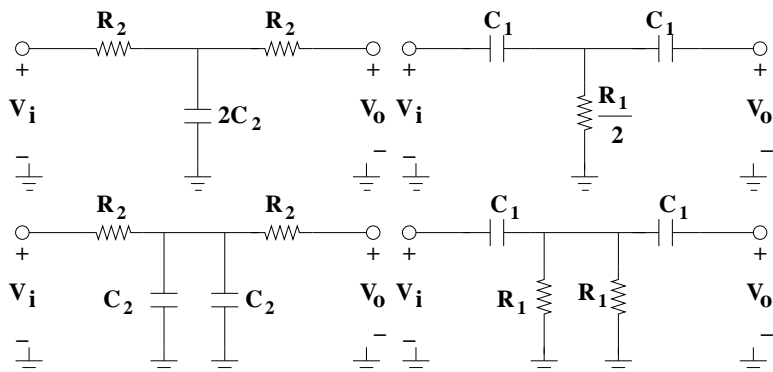


Figure 5.6: Twin T high pass and low pass sections.

The low pass and high pass sections are each similar to two cascaded LPFs and HPFs. Let's examine the low pass section first. We see that the value of the capacitor, $2C_2$, has two capacitors of value C_2 in parallel. When viewed this way we can see that this closely resembles two high-pass filters cascaded except that the location of the second HPF has the resistor preceded by the capacitor. If we examine the high pass section, we notice that the resistor $\frac{R_1}{2}$ can be redrawn with two resistors of value R_1 in parallel.

Typically, the resistor and capacitor values are selected so that $R_2 = R_1 = R$ and $C_2 = C_1 = C$. The output, $V_{out}(\omega)$ with respect to $V_{in}(\omega)$ can be

expressed as

$$A_v(\omega) = \frac{1 - (\omega RC)^2}{(1 + j\omega RC)^2} \quad (5.1)$$

where $A_v(\omega)$ is the attenuation of the circuit as a function of frequency.

The notch frequency (location of the null) is given by

$$f_o = \frac{1}{2\pi RC} \quad (5.2)$$

The power spectrum is given by the product of A_v with its complex conjugate or

$$A_v^2(f) = \left[\frac{1 - (\frac{f}{f_o})^2}{1 + (\frac{f}{f_o})^2} \right]^2 \quad (5.3)$$

where f_o is the notch frequency.

The phase response will be given by

$$\phi_v(f) = \tan^{-1} \left[\frac{2(\frac{f}{f_o})}{1 - (\frac{f}{f_o})^2} \right] \quad (5.4)$$

In the theory of linear systems equation (5.1) is referred to as the transfer function. The numerator of the transfer function has a magnitude of 0 at $f = f_o$. The null at f_o is also referred to as a *zero* or a *transmission-zero* for this reason. This means that the magnitude of equation (5.1) is equal to 0. This more apparent from equation (5.3).

The Bode plot (magnitude and phase) is shown in figure 5.7. Notice that the filter does not attenuate at all very far above or below the notch frequency. It is also worth pointing out that the notch depth appears to be limited in figure 5.7; however, this is a result of the graph and is not related to a limit in the notch depth. A more detailed scan and re-plot of the magnitude around the notch frequency is given in figure 5.8. Notice that the a signal at the null can be attenuated by -60 dB to -100 dB.

The actual depth of the null in fabricated Twin T filters can be made to be -60dB to -70dB. Increasing the notch depth beyond this is possible but greater care must be taken in the component accuracy and the implementation. The notch depth depends upon several factors[?]: (1) Component matching between the resistors and capacitors, (2) impedance matching at the input and output of the filter and (3) parasitic capacitors and resistors formed by the circuit board. One can theoretically obtain -100 dB attenuation or more; however, this presumes that the value of the resistors are very closely matched. The same is true for the values of the capacitors. An attenuation of -60dB is equivalent to a relative error of 1 part in 1,000. or .1% while an attenuation of -100dB is equivalent to an overall relative error of 1 part in 100,000, or .001 %. It's difficult to obtain resistors or

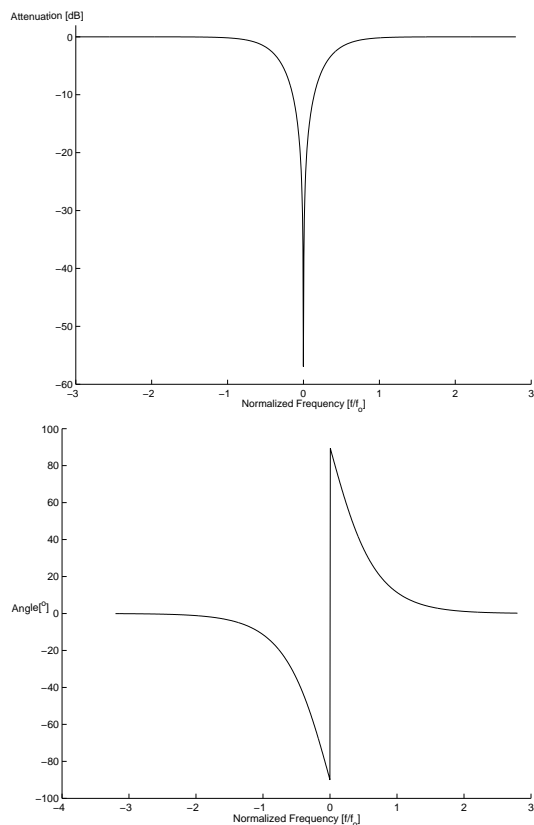


Figure 5.7: Twin T magnitude & phase plots.

capacitors which are matched to 0.1 %. There are also impedance losses and related problems which can add up quickly. The circuit board itself can change the effective values of capacitors and resistors. For example, a conductive line drawn on a circuit board will have a capacitance per unit length for a particular line width. These parasitics must also be taken into account if attenuation at the null is to be greater than -70 dB. Variations in design parameters due to the effects mentioned above will also shift the notch frequency. This creates a problem because the attenuation might not be maximized at the desired frequency.

5.4 Applications

One common application where the Twin T filter is often used is in the measurement of distortion in a high quality sine-wave generator[?, ?]. The measurement setup is shown in figure 5.9. Distortion can be loosely defined

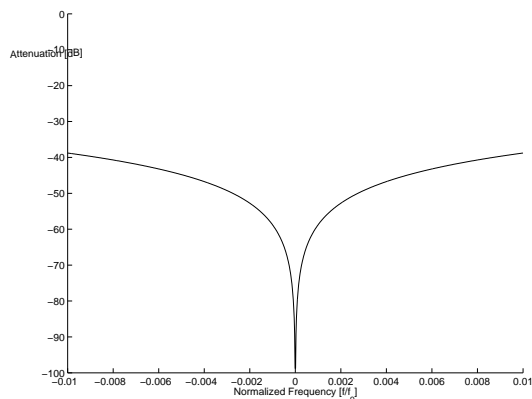


Figure 5.8: Detailed view of the Twin T magnitude response.

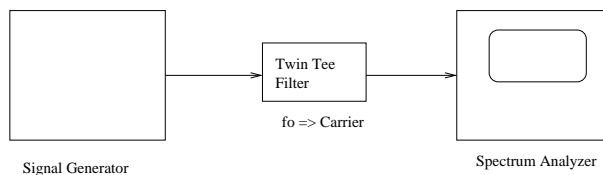


Figure 5.9: Example application of the Twin T Filter.

as 'sinusoidal-like' signals which exist at frequencies other than the selected sine wave frequency. Typically, these signals are similar (in the frequency spectrum) to the sine wave but they can be more than 120 dB (1 part in 1 million) below the selected sine-wave. This measurement is most efficiently performed with a spectrum analyzer. Unfortunately, most spectrum analyzers will not have a distortion floor or dynamic range large enough to allow one to view the signal+distortion on the same scale. Thus, the input signal must be attenuated prior to the spectrum analyzer as shown in

Another application of the Twin T filter is to attenuate *line-noise* from the 60Hz, 120 Volt, wall socket.

5.5 Measuring the Twin T Filter

In this lab you will measure several Twin T filters. It is important to realize from the discussion above (regarding the accuracy of the filter) that in the proto-board with the components available in our labs that it will be difficult to obtain even -40 dB attenuation. The proto-board introduces large parasitic capacitances and often introduces other undesirable problems.

You can expect that the depth will be difficult to measure near the null and

also that the notch frequency will not be precisely at the design frequency. This means that it is of paramount importance to *sweep* the signal generator over a wide enough range for you to be able to see the output sine wave being attenuated. It is also important that you carefully sweep over a reduced frequency range when attempting to locate the notch frequency. Although the measurements around this filter are widely spaced, you will need to pick many points near the notch in order to estimate the notch frequency.

5.6 Lab Instructions

- 1) Construct the Twin T filter setting the resistor, $R = 1 \text{ k}\Omega$ and $C = .093 \mu\text{F}$. Compute the frequency of the null.
 - i) Perform a wide sweep and a narrow sweep in order to insure proper operation and also to accurately verify the location of the notch frequency. Record the measured estimate of f_o .
 - ii) Measurement of the frequency response.
 - a) Make a table with the following columns: f , V_{in} , V_{out} Measured, V_{out} Calculated, $|A_v|^2$ Measured, $|A_v|^2$ Calculated, $A_v[dB]$ Measured, $A_v[dB]$ Calculated, Δt , T Measured, ϕ Measured and ϕ Calculated.
 - b) Measure the attenuation for the following frequencies: 10 Hz, 30 Hz, 100Hz, 300 Hz, 800 Hz, 1 kHz, 3 kHz, 10 kHz, 30 kHz and 100 kHz. Select 5 frequencies above and below $f = f_o$ within $\pm 10\%$ of $f = f_o$. Set the peak amplitude of the input sine wave to 10 volts.
 - c) Complete the entries in the table.
 - d) Plot $A_v[dB]$ Measured vs. $\log_{10}(f)$ and ϕ Measured vs. $\log_{10}(f)$. Overlay each graph with a plot of $A_v[dB]$ Calculated vs. $\log_{10}(f)$ and ϕ Calculated vs. $\log_{10}(f)$.
- 2) Construct the Twin T filter setting the resistor, $R = 2 \text{ k}\Omega$ and $C = .0093 \mu\text{F}$. Compute the frequency of the null.

- i) Perform a wide sweep and a narrow sweep in order to insure proper operation and also to accurately verify the location of the notch frequency. Record the measured estimate of f_o .
- ii) Measurement of the frequency response.
 - a) Make a table with the following columns: f , V_{in} , V_{out} Measured, V_{out} Calculated, $|A_v|^2$ Measured, $|A_v|^2$ Calculated, $A_v[dB]$ Measured, $A_v[dB]$ Calculated, Δt , T Measured, ϕ Measured and ϕ Calculated.
 - b) Measure the attenuation for the following frequencies: 100Hz, 300 Hz, 800 Hz, 1 kHz, 3 kHz, 5kHz, 10 kHz, 20kHz, 30 kHz, and 100 kHz. Select 5 frequencies above and below $f = f_o$ within $\pm 10\%$ of $f = f_o$. Set the peak amplitude of the input sine wave to 10 volts.
 - c) Complete the entries in the table.
 - d) Plot $A_v[dB]$ Measured vs. $\log_{10}(f)$ and ϕ Measured vs. $\log_{10}(f)$. Overlay each graph with a plot of $A_v[dB]$ Calculated vs. $\log_{10}(f)$ and ϕ Calculated vs. $\log_{10}(f)$.
- 3) Construct the Twin T filter setting the resistor, $R = 1\text{ k}\Omega$ and $C = .0093\ \mu F$. Compute the frequency of the null.
 - i) Perform a wide sweep and a narrow sweep in order to insure proper operation and also to accurately verify the location of the notch frequency. Record the measured estimate of f_o .
 - ii) Measurement of the frequency response.
 - a) Make a table with the following columns: f , V_{in} , V_{out} Measured, V_{out} Calculated, $|A_v|^2$ Measured, $|A_v|^2$ Calculated, $A_v[dB]$ Measured, $A_v[dB]$ Calculated, Δt , T Measured, ϕ Measured and ϕ Calculated.
 - b) Measure the attenuation for the following frequencies: 100Hz, 300 Hz, 800 Hz, 1 kHz, 3 kHz, 10 kHz, 20kHz, 30 kHz, 100 kHz and

130kHz. Select 5 frequencies above and below $f = f_o$ within $\pm 10\%$ of $f = f_o$. Set the peak amplitude of the input sine wave to 10 volts.

- c) Complete the entries in the table.
- d) Plot $A_v[dB]$ Measured vs. $\log_{10}(f)$ and ϕ Measured vs. $\log_{10}(f)$. Overlay each graph with a plot of $A_v[dB]$ Calculated vs. $\log_{10}(f)$ and ϕ Calculated vs. $\log_{10}(f)$.

5.7 Problems

- 1) What are the measured values of the notch frequencies ?
- 2) What is the relative error between the *design-value* of f_o and the actual value of f_o for each of your filters ?
- 3) What is the maximum notch-depth (in dB) at the notch frequencies ?

Chapter 6

RL Circuits

Lab Objectives:

- (a) Understand properties of inductors in DC & AC Circuits.
- (b) Measure the step response of an RL circuit.
- (c) Understand amplitude and phase relationships of ideal inductors.
- (d) Analyze and measure the frequency and phase response of Low Pass and High Pass Filters with ideal inductors.
- (e) Understand duality relations between capacitors and inductors.
- (f) Understand non-ideal effects with inductors.

Pre Lab:

- (a) Read this lab.
- (b) Find the time constants prior to the lab.
- (c) Find filter cutoffs prior to the lab.

6.1 Introduction

In the previous 2 labs we have examined transient and steady state behavior of RC circuits. The definition of capacitance was defined using relations between the electric field, voltage and charge. This led to the relation between voltage, current and capacitance. We used this relation to understand the transient response of an RC circuit. Reactance was defined specifically for the capacitor. This was used to analyze first-order low pass and high pass filters constructed from simple RC sections. Finally, this was extended to band-pass, band-stop and notch filters.

Inductance will be derived from the magnetic field and magnetic flux. We will see that although inductance is new to our study of electronics, the analysis is quite similar. The transient response of an RL filter produces a result quite similar to that of the RC circuit. Reactance will be defined for the inductor and this will be compared to the reactance of the capacitor. First order low-pass and high-pass filters can be realized by simple RL sections. Band-pass, band-stop and notch filters can also be constructed using RL circuits.

Inductors are used in many of the same applications as capacitors: power supply smoothing and analog filters. Inductance plays a role in the analysis of circuit boards, transmission lines, and antennas. Transformers represent applications which are unique to inductors.

6.2 Background

Inductors store energy from the magnetic field. The simplest example of an inductor is a wire which is wound around a rod. In order to understand how inductance works we will review selected parts of magnetic field theory. A time-varying charge produces a force which is given by

$$\begin{aligned} F &= q\mathbf{v} \times \mathbf{B} \\ dF &= I d\mathbf{l} \times \mathbf{B} \end{aligned} \quad (6.1)$$

where q is the charge, v is the velocity of the charge, I is the current, $d\mathbf{l}$ is a differential length and B is the magnetic field in Teslas.

Alternatively a current passing through a wire segment of differential length $d\mathbf{l}$ (l is measured in same direction as the charge velocity) provides differential force[1]. This is given by the second expression in equation (6.1).

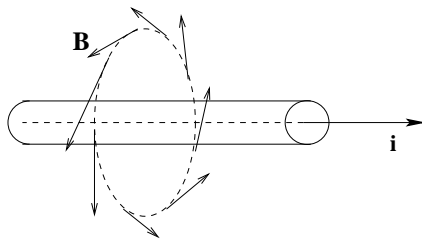


Figure 6.1: Magnetic field due to a straight wire using the right hand rule.

Figure 6.1 illustrates the location of the magnetic field for a current moving along a straight wire. The direction of the magnetic field is obtained using the right-hand rule.

If we form a line integral around the closed path, parallel to \mathbf{B} , encircling the current then the following relationship always holds:

$$\oint_C \mathbf{B} \cdot d\mathbf{l} = \mu_o I \quad (6.2)$$

where μ_o is defined as the permeability of free space and is equal to approximately $.4\pi \mu\text{H/m}$.

This result is known as Ampère's Law. Since the electric field is conservative the work done on a charge moved in a closed path is given by

$$\oint_C \mathbf{E} \cdot d\mathbf{l} = 0 \quad (6.3)$$

where E is the electric field and the integral is **always** zero around a closed path.

This line integral around a closed path is zero for the electric field and not always zero for the magnetic field[1]. This should not be confused with Gauss's Law where the integral over an enclosure is not zero for the electric field, but is in fact zero for the magnetic field. This is given in equation (6.4) and these two equations together with equation (6.2) are three of (the four) Maxwell's Equations. Recall that ϵ_o is the permittivity of free space and is equal to approximately $8.85 \times 10^{-12} \text{ F/m}$.

$$\begin{aligned} \phi_{net} &= \oint_S \mathbf{E} \cdot \hat{\mathbf{n}} dA = \frac{Q_{inside}}{\epsilon_o} \\ \phi_{net} &= \oint_S \mathbf{B} \cdot \hat{\mathbf{n}} dA = 0 \end{aligned} \quad (6.4)$$

If current is moving in a circular wire of radius R , then a magnetic field is induced on the axis at the center of the circle as shown in figure 6.2[1]. The field is zero in the direction perpendicular to the axis.

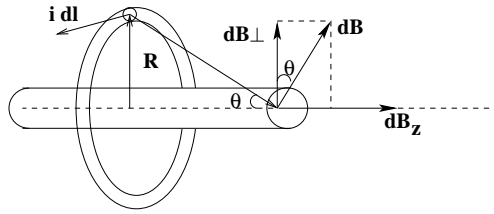


Figure 6.2: Computing the magnetic field on the axis of a current loop.

The flux generated by the wire loop illustrated in figure 6.2 is defined as

$$\phi_{Mag} = \int \mathbf{B} \cdot \hat{\mathbf{n}} dA = BA \cos \theta \quad (6.5)$$

where A is the cross-sectional area of the current loop.

If we were to wind multiple current loops around the same fixed radius, R , the flux would increase linearly with the number of loops or windings[1]. The flux linkage is defined as the sum of the total flux for multiple current loops formed by many windings[1, 2]. This is given by

$$\psi = -N\phi_{Mag} = Li \quad (6.6)$$

where ψ is the flux-linkage, ϕ_{Mag} is the magnetic flux, i is the current and L is the inductance.

Inductance is defined in equation (6.6). Inductance can be found for arbitrary geometries; however, we will restrict our derivations to one or more current loops where multiple current loops are realized by winding a wire around a cylinder.

The relationship between flux, flux-linkage and the electro-motive-force (emf) generated from a magnetic field is given by the following expression[1, 2]

$$V = \frac{d\psi}{dt} = -N \frac{d\phi_{Mag}}{dt} \quad (6.7)$$

where V is the emf.

This is the known as Faraday's Law and it is the remaining law summarized in Maxwell's equations. Faraday's Law states that a time varying magnetic field will induce a voltage[1, 2]. The differential relationship between inductance, voltage and current can be found by combining equations (6.6) and (6.7) yielding

$$V = \frac{d[LI]}{dt} = L_{const} \frac{dI}{dt} \quad (6.8)$$

where V is again the emf and L is the inductance.

If we assume that L is constant with respect to time we can remove L from the differential. This is the implication of L_{const} in equation (6.8). The voltage in an inductor can be compared to the current in a capacitor given in equation (6.9)

$$I = \frac{d[CV]}{dt} = C_{const} \frac{dV}{dt} \quad (6.9)$$

where Q , is the charge in Coulombs, V is the voltage in Volts and C is the Capacitance in Farads.

Equation (6.8) relates the current and voltage in an inductor (where the inductance is constant with respect to time). From this expression we see that inductance is a DC short (e.g. I does not change with respect to time thus there is no back emf). Comparing this to equation (6.9) we see that the capacitor is a DC open circuit.

There is a duality between inductance and capacitance. This is summarized below[2]:

$$\begin{array}{ccc}
 \textit{Inductor} & & \textit{Capacitor} \\
 L & \longleftrightarrow & C \\
 \psi & \longleftrightarrow & Q \\
 I & \longleftrightarrow & V
 \end{array} \tag{6.10}$$

The energy stored in the inductor can be expressed as[1]:

$$U = \frac{1}{2}LI^2 \tag{6.11}$$

where U is the potential energy (in Joules), L is the inductance and I is the current.

6.2.1 Construction of Inductor Coil

Inductors are not difficult to construct. A simple inductor can be realized by winding a wire for N turns around a cylindrical object, such as a test tube. This is illustrated in figure 6.3.

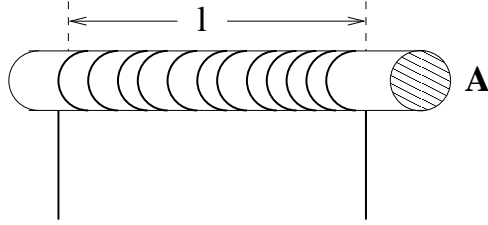


Figure 6.3: Inductor with N turns over a length l with cross sectional area, A .

The resulting magnetic flux is shown in figure 6.4. Notice that the flux lines encircle the windings.

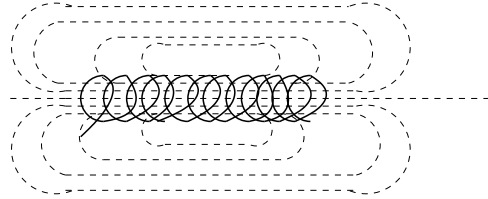


Figure 6.4: Magnetic field of a coil.

The flux-linkage is given by

$$\psi = NBA \tag{6.12}$$

and the magnetic field in the coil is given by

$$B = N \frac{\mu_o i}{l} \quad (6.13)$$

The inductance can readily be found by combining equations (6.6), (6.12) and (6.13) to obtain[1]

$$L = N^2 \frac{\mu A}{l} \quad (6.14)$$

where $\mu = \mu_o \mu_r$; μ is defined as the permeability and μ_r is defined as the relative permeability.

The relative permeability is 1 in vacuum and close to 1 for an air-filled core. For an iron-filled core μ_r is typically 4,000 to 6,000. Thus, replacing the test tube with an iron cylinder will significantly increase the inductance.

Care should be taken with equation (6.14) since it assumes a very long coil; effects near the ends are not included. There are tables and handbooks which provide the information necessary to construct very accurate coils[3].

6.2.2 Parallel & Series Combinations

The equivalent inductance from a series combination of n inductors is the sum of each individual inductance; this is expressed as,

$$L_{eff} = L_1 + L_2 + \dots + L_n \quad (6.15)$$

The parallel combination of inductors yields an effective inductance given by,

$$\frac{1}{L_{eff}} = \frac{1}{L_1} + \frac{1}{L_2} + \dots + \frac{1}{L_n} \quad (6.16)$$

Thus, the effective inductances just the opposite of what is seen with capacitance. Show that these relationships are in fact true.

6.2.3 Mutual Inductance & Transformers

The magnetic field of a circuit can induce a current in a nearby circuit. This is illustrated in figure 6.5. Thus, the relationship between flux-linkage, flux, inductance and current can be generalized further. There is no analogous property with capacitors.

This generalization for the two adjacent circuits shown in figure 6.5 is given by equation (6.17)[1, 2].

$$\begin{aligned} \psi_1 &= -N\phi_{Mag1} &= L_1 i_1 + M i_2 \\ \psi_2 &= -N\phi_{Mag2} &= L_2 i_2 + M i_1 \end{aligned} \quad (6.17)$$

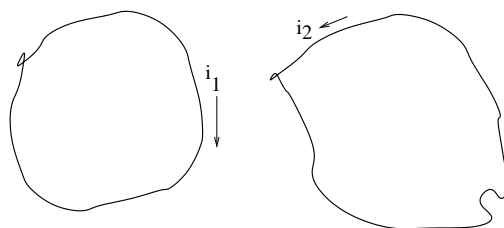


Figure 6.5: Adjacent circuits.

where ψ_1 , ϕ_{Mag1} and ψ_2 , ϕ_{Mag2} are the flux-linkage and flux of the first and second circuits respectively.

We see that the flux-linkage in the first circuit is related to the current in the first circuit by the self-inductance¹ of the first circuit and is also related to the current in the second circuit by a second constant defined as the mutual inductance, M . This is also true for the flux-linkage of the second circuit.

Mutual inductance can be further generalized to include more than two circuits. This makes the analysis somewhat more complicated. One can see that shielding from the magnetic field is typically more difficult than shielding from the electric field.

Mutual inductance represents the principle behind the operation of a transformer. Transformers are used in to change voltages and to isolate power supplies. An iron core transformer can be constructed by wrapping two coils around adjacent sides of a square iron rectangle as shown in figure 6.6. The coil with the voltage, V_1 , is referred to as the *primary* and the coil with the voltage V_2 is referred to as the *secondary*. V_1 and V_2 are assumed to be AC voltages.

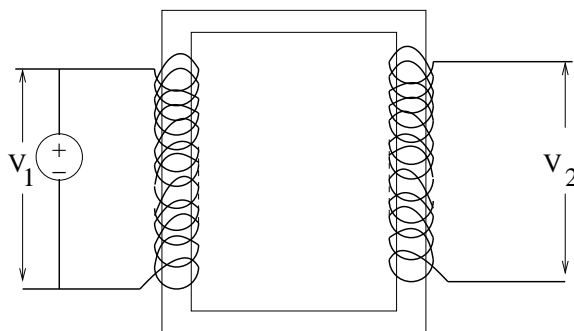


Figure 6.6: Iron core transformer.

¹The term *inductance* is automatically assumed to be identical to *self-inductance*.

The voltage, V_1 is fed into the primary and a second voltage, V_2 is then induced in the secondary. The relationship between V_1 and V_2 is given by [1, 2]

$$\begin{aligned} V_1 &= -N_1 \frac{d\phi}{dt} \\ V_2 &= -N_2 \frac{d\phi}{dt} \end{aligned} \quad (6.18)$$

where N_1 is the number of windings in the primary and N_2 is the number of windings in the secondary.

Combining terms we obtain [1, 2]:

$$\begin{aligned} V_2 &= \frac{N_2}{N_1} V_1 \\ i_2 &= \frac{N_1}{N_2} i_1 \end{aligned} \quad (6.19)$$

Thus, we see that the induced voltage is related to the primary voltage by the ratio of the number of turns in the secondary and primary. If N_2 is greater than N_1 , the secondary voltage, V_2 is increased. This is known as a *step-up* transformer. If the opposite is true (e.g. $N_1 > N_2$), then it is known as a *step-down* transformer.

The currents change with the reciprocal ratio of the windings. This guarantees that the rms output power can be no greater than the rms input power. However, the rms output power can be less due to losses. The *efficiency* is defined as ratio of output power to input power. This is one of the reasons for choosing an iron-core over an air core.

6.3 Step Response of an RL Circuit

6.3.1 Positive Step Response

If we assume that V_R is initially zero. Current is fed from the battery by closing the switch at time $t = 0$. This is shown in figure 6.7. There is an exponential relationship which governs the the voltage across the resistor as a function of time.

Using Kirchhoff's voltage law start by summing the voltages around the loop will yield equation (6.20).

$$V_s = V_L + V_{res} = L \frac{dI}{dt} + IR \quad (6.20)$$

where V_s is the voltage source and V_L is the voltage across the inductor.

The actual differential equation can be found by merely substituting the expressions for voltage in the resistor and the inductor into equation (6.20). Dividing through by R gives a differential equation for the currents which will be identical to that done for the capacitor. This results in

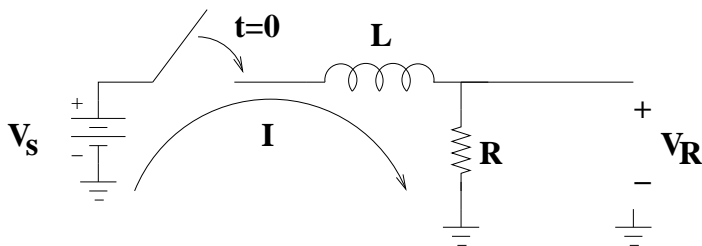


Figure 6.7: Circuit for the positive step response.

$$I_s = \frac{V_s}{R} = I + \frac{L}{R} \frac{dI}{dt} \quad (6.21)$$

Solving the differential equation and converting the currents to voltages yields:

$$V_R = V_s [1 - e^{-\frac{t}{\tau}}] \quad (6.22)$$

where $\tau = \frac{L}{R}$, which is the time constant. Note that when $t = \tau$, V_R is 63.2% of V_s .

This is shown graphically in figure 6.8. The voltage response is given relative to V_s and the horizontal axis is scaled to represent the number of time constants.

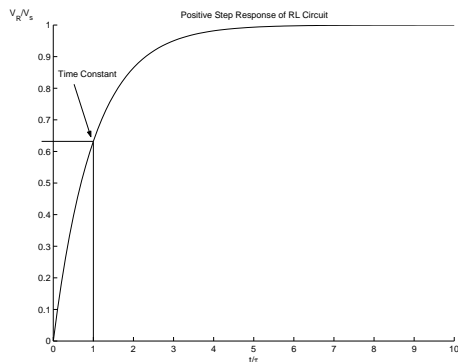


Figure 6.8: Relative voltage as a function of time constants for the positive step.

The relative voltage level for a single time constant is also shown.

6.3.2 Negative Step Response

Assuming that the voltage, V_R , is initially set to some voltage, V_o , the circuit for the negative step response is given in figure 6.9. The voltage across the resistor can be found from a similar differential equation.

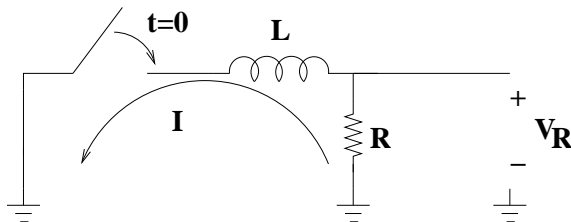


Figure 6.9: Circuit for the negative step response.

Using KVL the voltages around the loop are found to be

$$V_{res} + V_L = IR + L \frac{dI}{dt} = 0 \quad (6.23)$$

Solving equation (1) yields the following expression:

$$V_R = V_s e^{-\frac{t}{\tau}} \quad (6.24)$$

Again, the time constant is given by $\tau = \frac{L}{R}$. The value of V_R after one time constant is 36.8% of V_o .

This is shown graphically in figure 6.10. The voltage response is given relative to $V_o = V_s$ and the horizontal axis is scaled to represent the number of time constants.

The relative voltage level for a single time constant is also shown.

6.4 Phase and Frequency Relations of RL Circuits

There are analogies which can be drawn between phase shifts in inductors and capacitors. Inductors also cause a 90° phase difference between voltage and current. For a sine wave input the voltage across the inductor will lead the current in the inductor by $+90^\circ$; this is analogous to the current leading the voltage in the capacitor by $+90^\circ$. Thus, for a sine-wave input current, the voltage across the inductor is expressed as [4, 2]

$$V = \frac{d[LI]}{dt} = L \frac{d}{dt} [I_p \sin(\omega t)] = \omega L I_p \cos(\omega t) = \omega L I_p \sin(\omega t + 90^\circ) \quad (6.25)$$

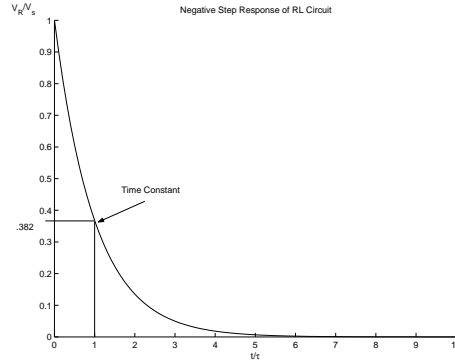


Figure 6.10: Relative voltage as a function of time constants for the discharging of the capacitor.

where ω is the radian frequency (e.g. $2\pi f$), I_p is the peak voltage and t represents time in seconds.

For the inductor the phase of the voltage leads the phase of the current or the current lags the voltage. Dividing the voltage by the current to find the resistance, as Ohm's Law suggests, leads to the definition of reactance, X_L , or

$$X_L(\omega) = \omega L \quad (6.26)$$

where X_L is measured in Ohm's and ω is the radian frequency ($2\pi f$).

The impedance is defined as the generalized resistance which includes both resistance and reactance. The impedance of a series connected resistor and inductor is given by

$$Z(\omega) = R + jX_L(\omega) \quad (6.27)$$

where Z is the impedance in Ohm's and $j = \sqrt{-1}$; the variable, (ω) , is often omitted leaving Z or X_L .

Equation (6.26) suggests that the inductor is a short at DC while the impedance increases linearly with increasing frequency.

6.5 The RL Low-Pass Filter with an Ideal Inductor

The RL circuit in the upper half of figure 6.11 is an RL low pass filter[4, 5]. It is composed of a series RL section where the output voltage is taken across the resistor. The inductor is placed in the direct path. The RC low pass filter is shown in the lower half of figure 6.11 to illustrate the duality between the RC and RL low pass filter[4, 2].

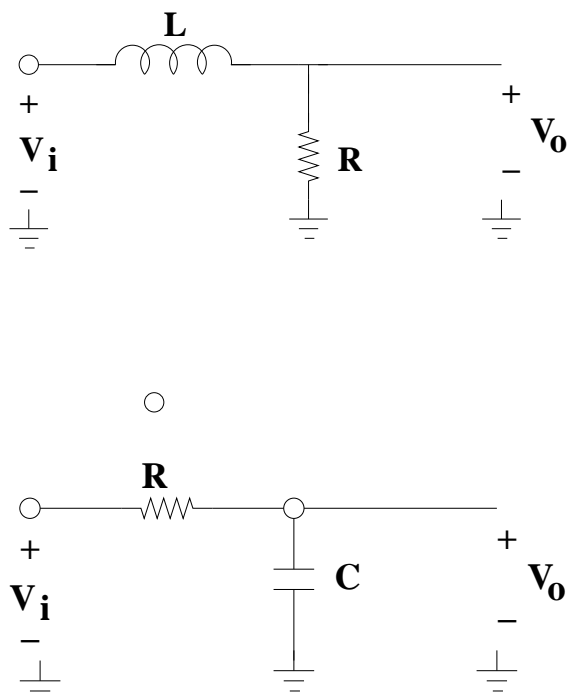


Figure 6.11: Low pass filters realized from RL and RC sections.

The output voltage (across the resistor), $V_{out} = V_L$, has a maximum value equal to $V_{in} = V_i$. We can express the attenuation of V_{out} with respect to V_{in} as

$$A_v = \frac{V_{out}}{V_{in}} = \frac{V_o}{V_i} \quad (6.28)$$

where $A_v(\omega)$ is the attenuation of the circuit as a function of frequency.

6.5.1 Integration

The time series response of the circuit in figure 6.11 can be expressed using KVL as

$$v_i(t) = v_R + v_L = iR + L \frac{dI}{dt} \quad (6.29)$$

where v_R is the voltage drop across the resistor and v_L is the voltage drop across the inductor ($v_o = v_R$).

If we assume that the period of the input voltage, $v_i(t)$, is *much greater* than the $\frac{L}{R}$ time constant, the amplitude of the output voltage, v_R will rise to

its maximum value. In other words, for input frequencies significantly lower than $\frac{R}{2\pi L}$ the amplitude of the output voltage is not attenuated. If the period of the input signal is made to be *much smaller* than the $\frac{L}{R}$ time constant, e.g. $\frac{1}{T_i} \gg \frac{R}{2\pi L}$, the voltage drop across the inductor in equation (6.29) will be much larger than that of the resistor. The voltage drop across the inductor will be approximately equal to the input voltage[4], or,

$$v_i(t) \approx v_L \quad (6.30)$$

and

$$i = \frac{1}{L} \int V_L dt \approx \frac{1}{L} \int v_i(t) dt \quad (6.31)$$

Thus, when $\frac{R}{2\pi L} \gg T_i$ the output voltage can be found as a function of $v_i(t)$ leading to the following expression,

$$v_o = v_R \approx \frac{R}{L} \int v_i(t) dt \quad (6.32)$$

which tells us that the output voltage is the integral of the input voltage when the input frequency is much greater than $\frac{R}{2\pi L}$.

6.5.2 Frequency & Phase Response

The method used to compute the frequency response will be identical to that used for the frequency response of the capacitor; the only difference is that we will substitute the reactance, $\frac{-1}{\omega C}$, with ωL . We start by forming a voltage divider. This yields

$$A_v(\omega) = \frac{R}{R + jX_L} = \frac{1}{1 + j\omega \frac{L}{R}} \quad (6.33)$$

where X_L is the reactance of the inductor and R is the resistance. If we substitute equation (6.26) into equation (6.33) the relation between V_{out} and V_{in} can be found as a function of R , L and ω . Equation (6.33) can be reduced to

$$A_v(\omega) = \frac{1}{1 + j\left(\frac{f}{f_o}\right)} = \frac{1 - j\left(\frac{f}{f_o}\right)}{1 + \left(\frac{f}{f_o}\right)^2} \quad (6.34)$$

where $f_o = \frac{R}{2\pi L}$ is the “cutoff” frequency. Does equation (6.34) look familiar?

The power is expressed as the squared magnitude of $A_v(\omega)$ or

$$P_{LPF}(f) = |A_v(\omega)|^2 = A_v(\omega)A_v^*(\omega) = \frac{1}{1 + \left(\frac{f}{f_o}\right)^2} \quad (6.35)$$

where $A_v^*(\omega)$ is the complex conjugate of $A_v(\omega)$ and $P_{LPPF}(f)$ is the power as a function of frequency.

The phase response is expressed as the inverse tangent of the imaginary part divided by the real part of $A_v(\omega)$ [4, 5]. This becomes,

$$\phi(\omega) = -\tan^{-1}\left(\omega \frac{L}{R}\right) = 90 - \tan^{-1}\left(\frac{R}{\omega L}\right) = -\tan^{-1}\left(\frac{f}{f_o}\right) \quad (6.36)$$

When we express the frequency response as a function of f_o the result is identical to that of the RC lowpass filter; when $f = f_o$, $P_{LPPF}(f = f_o) = P_{LPPF}(f = 0)$ and the attenuation in power is cut by a factor of exactly 2. The bandwidth of the filter is *always* measured by f_o (e.g. the *3dB* point).

Figure 6.12 provides a plot of the frequency and phase response of the RL low pass filter. Notice that the Bode Plot for the RL low pass filter is identical to that of the RC low pass filter (which should be expected).

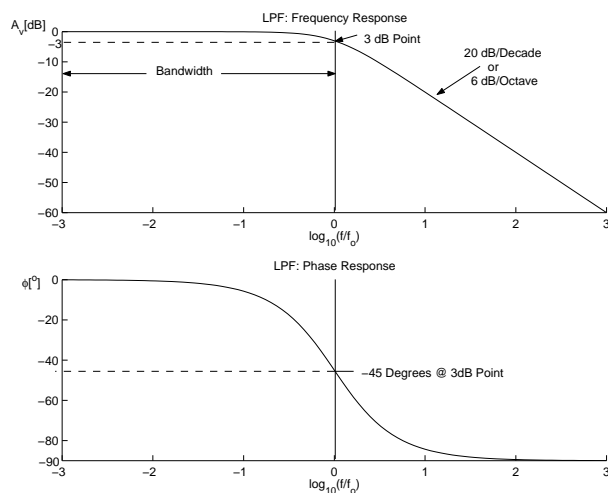


Figure 6.12: Low pass filter frequency and phase response.

These plots provide a complete characterization of the RL low pass filter. To review, *3dB* point occurs when $f = f_o$. At this point $A_v = -3dB$ and the phase is -45° . The *effective bandwidth* (or *bandwidth*) of the low pass filter is measured from DC to f_o . The *roll off* is defined as the slope of the frequency response of the Bode Plot in the linear part of the region where $f > f_o$. The roll off of an ideal low pass filter is always 6 dB per octave or 20 dB per decade. The phase response is negative; at DC, the phase is 0° and it approaches -90° as $f \rightarrow \infty$.

6.6 The RL High-Pass Filter with an Ideal Inductor

The circuit in the top half of figure 6.13 is an RL high pass filter[4, 5]. The high pass filter is a series RL section where the output voltage is taken across the inductor. The resistor is connected in the direct path between the input and output. The inductor is connected between the output and ground. The RC high pass filter is shown in the bottom half of figure 6.13 to illustrate the duality between RL and RC high pass filters.

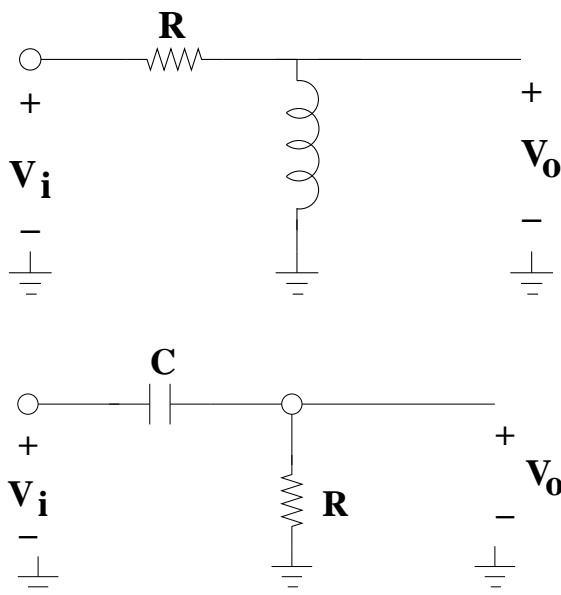


Figure 6.13: High pass filters realized from RL and RC sections.

In both filters DC signals will be blocked from the output while AC signals will pass to the output. In the RL filter, the inductor is a DC short while the capacitor in the RC filter is a DC open circuit.

6.6.1 Differentiation

The time series response of the circuit in figure 6.11 can be expressed using KVL as

$$v_i(t) = v_R + v_L = iR + L \frac{dI}{dt} \quad (6.37)$$

where v_R is the voltage drop across the resistor and v_L is the voltage drop across the inductor ($v_o = v_L$).

If we assume that the period of the input voltage, $v_i(t)$, is *much smaller* than the $\frac{L}{R}$ product, the amplitude of the output voltage, v_L will rise to its maximum value. In other words, for input frequencies significantly higher than $\frac{R}{2\pi L}$ the amplitude of the output voltage is not attenuated. If the period of the input signal is made to be *much greater* than the $\frac{L}{R}$ time constant, e.g. $\frac{1}{T_i} \ll \frac{R}{2\pi L}$, the voltage drop across the resistor in equation (6.37) will be much larger than that of the inductor. The voltage drop across the resistor will be approximately equal to the input voltage[4], or,

$$v_i(t) \approx v_R \quad (6.38)$$

and

$$i = \frac{v_R}{R} \approx \frac{v_i(t)}{R} \quad (6.39)$$

When $\frac{2\pi L}{R} \ll T_i$ the output voltage can be found as a function of $v_i(t)$ leading to the following expression,

$$v_o = v_L = L \frac{di}{dt} \approx L \frac{d}{dt} \left[\frac{v_i(t)}{R} \right] = \frac{L}{R} \frac{dv_i(t)}{dt} \quad (6.40)$$

Thus, the output voltage is the derivative of the input voltage when the input frequency is much less than the $\frac{R}{2\pi L}$.

6.6.2 Frequency & Phase Response

We will again find the frequency response by computing the attenuation directly from the circuit impedance by forming a voltage divider. This yields

$$A_v(\omega) = \frac{jX_L}{R + jX_L} \quad (6.41)$$

If we substitute equation (6.26) into equation (6.41) the relation between V_o and V_i can be found as a function of R , L and ω . Equation (6.41) is reduced to

$$A_v(\omega, f) = \frac{j\omega \frac{L}{R}}{1 + j\omega \frac{L}{R}} = \frac{j \left(\frac{f}{f_o} \right)}{1 + j \left(\frac{f}{f_o} \right)} = \frac{1 + j \left(\frac{f_o}{f} \right)}{1 + \left(\frac{f_o}{f} \right)^2} \quad (6.42)$$

where $f_o = \frac{R}{2\pi L}$ is the “cutoff” frequency.

The power is expressed as the squared magnitude of $A_v(\omega)$ or

$$P_{HPF}(f) = |A_v(\omega)|^2 = A_v(\omega)A_v^*(\omega) = \frac{1}{1 + \left(\frac{f_o}{f} \right)^2} \quad (6.43)$$

where $A_v^*(\omega)$ is the complex conjugate of $A_v(\omega)$ and $P_{HPF}(f)$ is the power as a function of frequency.

The phase response is expressed as the inverse tangent of the imaginary part divided by the real part of $A_v(\omega)$. This becomes,

$$\phi(\omega) = -\tan^{-1}\left(\frac{R}{\omega L}\right) = 90 - \tan^{-1}(\omega L) = \tan^{-1}\left(\frac{f_o}{f}\right) \quad (6.44)$$

Once again notice that when $f = f_o$, that $P_{HPF}(f = f_o) = P_{HPF}(f = 0)$. The attenuation in power is cut by a factor of exactly 2.

6.6.3 Bode Plots

The Bode Plot for the high pass filter is shown in the upper portion of figure 6.14. The phase response is shown in the lower portion of figure 6.14.

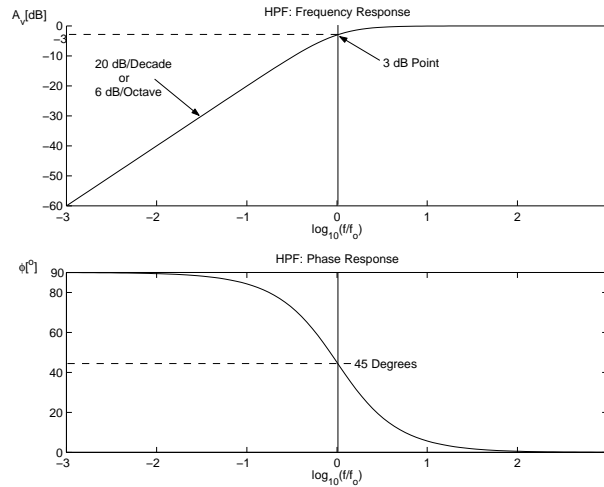


Figure 6.14: High pass filter frequency and phase response.

The high pass filter frequency and phase response is analogous to that of the low pass filter. The point where $f = \frac{f_o}{2\pi}$ is referred to as the *3dB point*. The *roll off* is defined as the slope of the frequency response of the Bode Plot in the linear part of the region where $f > f_o$. The phase response is negative; at DC, the phase approaches 90° and it approaches 0° as $f \rightarrow \infty$.

6.7 Non-Ideal Inductors

The inductors characterized in the previous sections are assumed to be ideal. Unfortunately, the inductors you will use in the lab are far from ideal. Non-

ideal effects occur in ALL electronic assemblies. These unwanted effects include additional resistance, capacitance and inductance; they manifest themselves in components, circuit boards and wiring. These effects are often negligible in practice (especially in our labs). When using the inductor you will find that some non-ideal effects will change the behavior of the RL filters described earlier. The measurements will deviate what is expected based upon the predictions of previous sections. How do we deal with non-ideal effects in a component? The answer is to construct the non-ideal component from a network of *ideal components* to quantify each of the undesired effects. This approach is identical to what is done in modeling circuit boards and integrated circuits. A composite model of the non-ideal inductor is described in figure 6.15.

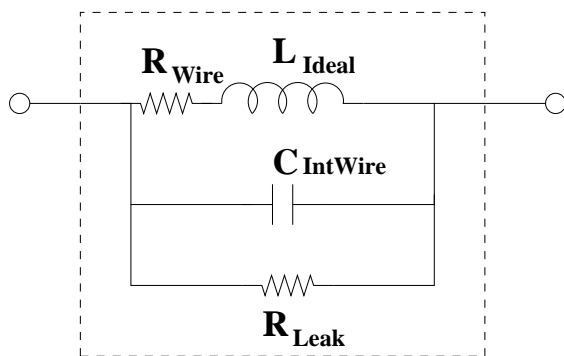


Figure 6.15: Composite non-ideal inductor.

The model inside the dashed box contains the components (all assumed to be ideal) which represent non-ideal effects. The major effects modeled are the wire resistance for the coil, the inter-wire capacitance and the resistance due to dielectric leakage (we can ignore this one)[4, 3]. Each of these effects are frequency dependent. For example, a long length of wire has a finite but small resistance. This is often the only non-ideal effect noticed at low frequencies. In this case the low frequency model of the inductor might include *only* the wire resistance, R_{Wire} , which is a series resistance. The wire resistance is only noticeable when $X_L(\omega)$ is small relative to R_{Wire} ; obviously this occurs only when ω is low (hence low frequencies). Thus, the low frequency model of the inductor replaces the reactance from the ideal model, X_L , with the following expression:

$$Z_L = R_{Wire} + j\omega L \quad (6.45)$$

The frequency response derived from equation (6.33) changes to

$$A_v(\omega) = \frac{R}{R + R_{Wire} + jX_L(\omega)} \quad (6.46)$$

Compute the DC gain (*when* $\omega = 0$) of the low pass filter using equation (6.46). Notice that the gain will not be 1. Since R_{Wire} is quite small the gain slightly less than unity. This is observable with our inductors.

At high frequencies X_L tends to be much larger than R_{Wire} so the wire resistance can be ignored. However, there is an inter-wire capacitance which is formed by the parallel layers of wire conductors used in the windings of the coil. This capacitance has an influence at high frequencies and must be included. Can you think of how you could analyze the low pass filter with a non-ideal inductor ?

6.8 Lab Instructions

- 1) Construct the low pass filter and set it up as shown in figure 6.16. Measure the time constant for $L = 50mH$ and the resistor value of $R = 3.3 k\Omega$.

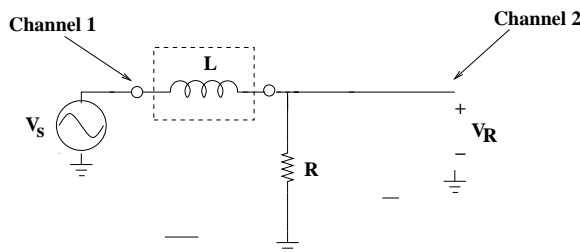


Figure 6.16: Low Pass Filter.

- i) Measure the positive step response at the following times: $.1 \tau$, $.5\tau$, τ , 1.5τ , and 2τ .
- ii) Measure the negative step response at the following times: $.1 \tau$, $.5\tau$, τ , 1.5τ , and 2τ .
- iii) Measure the wire resistance of the 50 mH inductor.

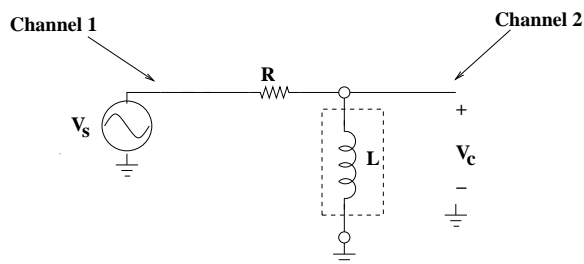


Figure 6.17: High Pass Filter.

- 2) Low Pass Filter: Continue with the low pass filter in figure 6.16 using $R = 3.3 \text{ k}\Omega$ and $L = 50 \text{ mH}$.
 - i) Make a table with the following columns: f , V_{in} , V_{out} Measured, V_{out} Calculated, $|A_v|^2$ Measured, $|A_v|^2$ Calculated, $A_v[\text{dB}]$ Measured, $A_v[\text{dB}]$ Calculated, Δt , T Measured, ϕ Measured and ϕ Calculated.
 - ii) Measure the attenuation for the following frequencies: 10 Hz, 100 Hz, 1 kHz, 3 kHz, 10 kHz, 30 kHz and 100 kHz. Also measure at $f = f_o$. Set the peak amplitude of the input sine wave to 10 volts.
 - iii) Complete the entries in the table.
 - iv) Plot $A_v[\text{dB}]$ Measured vs. $\log_{10}(f)$ and ϕ Measured vs. $\log_{10}(f)$. Overlay each graph with a plot of $A_v[\text{dB}]$ Calculated vs. $\log_{10}(f)$ and ϕ Calculated vs. $\log_{10}(f)$.
 - v) Compute the cutoff frequency. Identify the 3dB point extrapolated from the measurements and overlay this with the computed value.
- 3) Integration: Demonstrate the integral of a 10 volt square wave input. Use the set up in figure 6.16.
 - i) Use the time constant found earlier to precompute the times listed below.

- ii) Sketch the input and output waveforms observed on the oscilloscope for the following values $\frac{\tau}{T} = .001, .05, .2, .75, 1$ and 2 .
- 4) High Pass Filter: Construct the high pass filter and set it up as shown in figure 6.17. Use $R = 33\text{ k}\Omega$ and $L = 50\text{mH}$.
- i) Make a table with the following columns: f , V_{in} , V_{out} Measured, V_{out} Calculated, $|A_v|^2$ Measured, $|A_v|^2$ Calculated, $A_v[\text{dB}]$ Measured, $A_v[\text{dB}]$ Calculated, Δt , T Measured, ϕ Measured and ϕ Calculated.
- ii) Measure the attenuation for the following frequencies: 100 Hz, 1 kHz, 10 kHz, 100 kHz and 300 kHz. Also measure at $f = f_o$. Set the peak amplitude of the input sine wave to 10 volts.
- iii) Complete the entries in the table.
- iv) Plot $A_v[\text{dB}]$ Measured vs. $\log_{10}(f)$ and ϕ Measured vs. $\log_{10}(f)$. Overlay each graph with a plot of $A_v[\text{dB}]$ Calculated vs. $\log_{10}(f)$ and ϕ Calculated vs. $\log_{10}(f)$.
- v) Compute the cutoff frequency. Identify the 3dB point extrapolated from the measurements and overlay this with the computed value.
- 5) Differentiation: Demonstrate the integral of a 10 volt sawtooth wave input. Use the set up in figure 6.16. Compute the RC time constant.
- i) Compute the $\frac{L}{R}$ time constant.
- ii) Sketch the input and output waveforms observed on the oscilloscope for the following values $\frac{\tau}{T} = 10, 1, 0.33,$ and 0.1 .

6.9 Problems

- 1) Explain the differences between the ideal and non-ideal LPF at low frequencies using the low frequency model of the inductor and the measurements taken in the lab.

- 2) Estimate the inter-wire capacitance described in figure 6.15 from the measurements taken on the high pass filter.
 - i) What would you use for a high frequency model of the inductor ?
 - ii) Estimate the inter-wire capacitance using this model and the HPF measurements.

- 3) Given: $L = 50mH$, $f = 15kHz$
 - a) Find: the reactance, X_L and the impedance, Z_L , (jX_L)
 - b) Find: Z if the inductor is connected in series with a $300\ \Omega$ resistor
 - c) Plot the impedance, Z , on the complex plane

References

- [1] P. A. Tipler, *College Physics*. Worth-Publishing, 1987.
- [2] M. Van Valkenburg, *Network Analysis*. Prentice Hall, 1974.
- [3] R. Myers, L. Campbell, T. Dorbuck, G. Hall, L. McCoy, T. McMullen, and P. Williams, eds., *The Radio Amateurs' Handbook*. Amature Radio Relay League, 1974.
- [4] A. J. Diefenderfer, *Principles of Electronic Instrumentation*. W.B. Sanders, 1972.
- [5] R. D. Strum and J. R. Ward, *Electric Circuits and Networks*. Quantum Publishers, 1973.

Chapter 7

Series RLC Circuit

Lab Objectives:

- (a) Basic characterization of simple series RLC circuit.
- (b) KVL, phase relations & phasors.
- (c) Understand the transient response of the series RLC circuit.
- (d) Completely characterize and measure second-order systems.

Pre Lab:

- (a) Read this lab.
- (b) Compute parameters prior to the laboratory session.

7.1 Introduction

The previous labs have provided us with a good introduction to transient and AC analysis of RC and RL circuits. The capacitor and inductor, although different, satisfy the same first order differential equations for the transient and steady state response. The analysis can be generalized.

The series RLC circuit combines both capacitive and inductive reactance. This circuit can be characterized by a second order differential equation. The capacitive and inductive reactance tend to offset each other; however, they can combine coherently. When this occurs the total impedance of the series RLC circuit is purely resistive. This occurs at the resonant frequency.

KVL and total impedance can be expressed graphically using phasor diagrams for particular frequencies. We will compute the values for the phasors based on measurements of the series RLC. We will also plot the phasor representation of the circuit for particular frequencies.

This lab will be subdivided into two parts; part (a) will focus on phasor analysis and solving KVL for several frequencies while part (b) will focus on special cases of the solution to the second order differential equation and how these solutions affect the transient and steady state behavior.

7.2 Response of Simple RLC

The series RLC circuit is shown in figure 7.1. It is composed of a capacitor, an inductor and a resistor connected in series. The voltage drop across each element must sum to the input voltage to satisfy KVL. This will work as long as the phase angles are included in the computation.

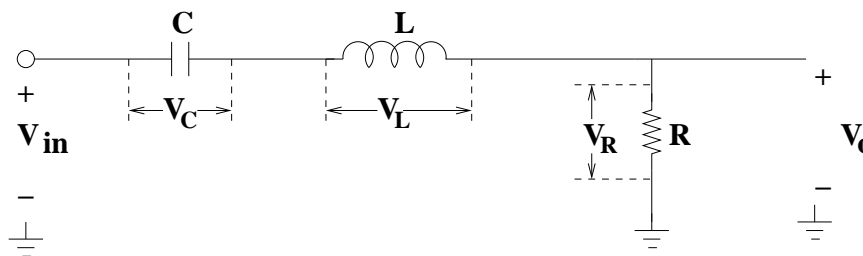


Figure 7.1: Series RLC circuit.

The frequency response of the circuit in figure 7.1 can be found by means of a simple voltage divider, or

$$\frac{V_o}{V_{in}} = \frac{V_R}{V_{in}} = \frac{R}{R + j(X_L + X_C)} = \frac{R}{R + j(\omega L - \frac{1}{\omega C})} \quad (7.1)$$

where X_L is the inductive reactance, X_C is the capacitive reactance and ω is the angular frequency.

This leads to the expressions for the amplitude and phase response

$$\begin{aligned} |A_v|^2 &= \frac{R^2}{R^2 + (X_L + X_C)^2} \\ \phi(\omega) &= \tan^{-1} \left[\frac{\frac{1}{\omega C} - \omega L}{R} \right] \end{aligned} \quad (7.2)$$

where A_v is the attenuation.

Since the signs for X_L and X_C are opposite that there is value of ω where $X_L = X_C$. At this frequency the reactive terms will cancel each other and the impedance becomes purely resistive. This means that V_o in equation (7.1) will be at its maximum value (V_{in}) and there will be a phase angle of zero relative to V_{in} . This is known as the resonant frequency at it can

be found by solving for the value of ω forcing $X_L = X_C$. The resonant frequency can be expressed in radians/sec or in Hz and is given by

$$\omega_R = \frac{1}{\sqrt{LC}}; f_R = \frac{1}{2\pi\sqrt{LC}} \quad (7.3)$$

where ω_R is in angular frequency and f_R is in Hz.

The frequency and phase response for a series RLC circuit has been plotted relative to the resonant frequency in figure 7.2. The filter has a bandpass response. Thus, there are two frequencies on each side of the peak which represent $3dB$ points, f_L and f_H . The difference, $f_H - f_L$ is defined as the filter bandwidth.

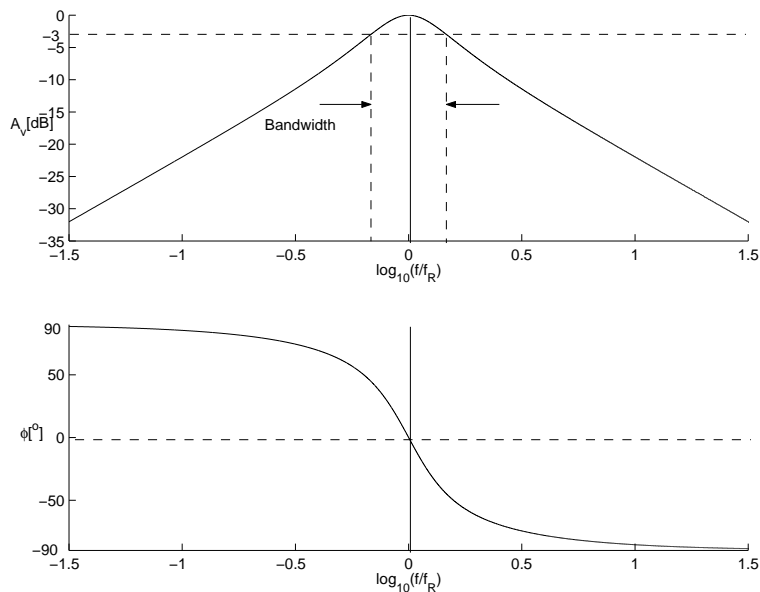


Figure 7.2: Frequency Response and Phase Response of Series RLC circuit.

Notice that the roll-off on either side of the filter is greater than 20 dB/decade . This is somewhat better than one would expect by constructing a bandpass filter from a high-pass filter and a low pass filter using RC or RL sections. There is a quantitative measure of the sharpness or selectivity in the frequency response. This is known as the **Q** of a circuit. The definition of the **Q** of a circuit is

$$Q = \frac{2\pi(\text{Maximum Stored Energy})}{(\text{Energy Dissipated per cycle})} \quad (7.4)$$

The Q of the series RLC circuit is [1, 2]

$$Q = \frac{1}{R} \sqrt{\frac{L}{C}} = \frac{\omega_R L}{R} \quad (7.5)$$

where R , L & C are the resistor, inductor and capacitor values and ω_R is the resonant frequency.

To illustrate the impact of the Q of a circuit figure 7.3 shows a simple example of a high Q and low Q circuit.

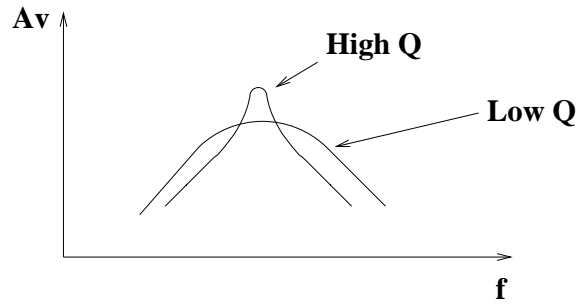


Figure 7.3: Impact of Q value on the frequency response..

The $3dB$ points and the bandwidth can also be found in terms of the Q value as

$$\begin{aligned} \omega_{LO} &= \omega_R \left[-\frac{1}{2Q} + \sqrt{\left(\frac{1}{2Q}\right)^2 + 1} \right] \\ \omega_{HI} &= \omega_R \left[\frac{1}{2Q} + \sqrt{\left(\frac{1}{2Q}\right)^2 + 1} \right] \\ \omega_{BW} &= \omega_{HI} - \omega_{LO} = \frac{\omega_R}{Q} \end{aligned} \quad (7.6)$$

where ω_{HI} , ω_{LO} and ω_{BW} are the high, low $3dB$ points and the bandwidth, respectively.

We will prepare an example using $R = 5.6 \text{ k}\Omega$, $L = 50 \text{ mH}$ and $C = .001 \text{ }\mu\text{F}$. The resonant frequency is easily found using equation (7.5); it is approximately 22.5 kHz.

7.2.1 KVL

Solving for KVL yields

$$V_{in} = V_C + V_L + V_R \quad (7.7)$$

where V_{in} is the input voltage, V_C , is the voltage drop across the capacitor, V_L is the voltage drop across the inductor and V_R is the voltage drop across the resistor.

The impedance can be found by summing the resistance and the reactance for each element. This will be identical to the denominator in equation (7.1) and it is given by

$$Z_D = R + j(X_L + X_C) = R + j\left(\omega L - \frac{1}{\omega C}\right) \quad (7.8)$$

where Z_D denotes the denominator impedance.

The impedance, Z_D , will be at its minimum at resonance (e.g. $\omega = \omega_R$) when the capacitive and inductive reactances negate each other leaving $Z_D = R$.

The values of $X_L(2\pi 10^3)$ and $X_C(2\pi 10^3)$ can easily be found and are given by

$$\begin{aligned} X_L &= \omega L = 2\pi 10^3 \frac{1}{sec} 50 \times 10^{-3} H = j3.14 k\Omega \\ X_C &= -\frac{1}{\omega C} = -\frac{1}{2\pi 10^3 \frac{1}{sec} .001 \times 10^{-6} F} = -j16 k\Omega \end{aligned} \quad (7.9)$$

Notice that the results are purely imaginary. This means that the inductor and capacitor are assumed to be ideal. Z_D can easily be found by substituting the values found in equation (7.9) into equation (7.8). This is given both in rectangular and polar form below

$$Z_D = (5.6 - j12.86) k\Omega = 14.0 k\Omega e^{-j66.5^\circ} \quad (7.10)$$

Notice that the phase angle is negative. This make sense since below the resonant frequency the capacitive reactance will dominate the imaginary terms. Above resonance the inductive reactance will dominate the imaginary terms.

The values of V_R , V_L and V_C can be found using simple voltage division, or,

$$\begin{aligned} V_R &= \frac{R}{Z_D} V_{in} = \frac{5.6}{14} V_{in} e^{j66.5^\circ} = 0.4 V_{in} e^{j66.5^\circ} \\ V_L &= \frac{X_L}{Z_D} V_{in} = \frac{3.14}{14} V_{in} e^{j156.5^\circ} = 0.22 V_{in} e^{j156.5^\circ} \\ V_C &= \frac{X_C}{Z_D} V_{in} = \frac{16}{14} V_{in} e^{-j23.5^\circ} = 1.14 V_{in} e^{-j23.5^\circ} \\ V_{in} e^{-j0^\circ} &= V_R + V_L + V_C \end{aligned} \quad (7.11)$$

If we convert to rectangular form it is seen that KVL is indeed satisfied.

$$\begin{aligned} V_R &= (0.159 + j0.367) V_{in} \\ V_L &= (-0.202 + j0.088) V_{in} \\ V_C &= (1.04 - j4.55) V_{in} \\ \hline \Sigma &= (1 + j0) V_{in} = V_{in} \end{aligned} \quad (7.12)$$

One should be careful when using rms values; KVL will not hold if rms values are used. Let's examine the rms sum. It is found by averaging the voltages in equation (7.15) resulting in

$$\begin{array}{rcl} [V_R]_{rms} & = & 0.283V_{in} \\ [V_L]_{rms} & = & 0.155V_{in} \\ [V_C]_{rms} & = & 0.807V_{in} \\ \hline \Sigma & = & 1.24V_{in} \neq V_{in} \end{array} \quad (7.13)$$

Thus, directly measuring each element with a volt meter will not satisfy KVL unless the phase angles are incorporated. That's the reason that the rms values do not satisfy KVL; phase angles are not included in rms computations.

7.2.2 Phasors

The voltages in a circuit with reactive elements are complex numbers. Complex voltages are represented graphically in the complex plane. The complex plane is similar to the infamous XY plane. Real quantities are plotted on abscissa and purely imaginary quantities are plotted on the ordinate. Real quantities have a phase angle of 0° while imaginary quantities have a phase angle of $\pm 90^\circ$. Phasors are vectors which represent complex voltages and are plotted on the complex plane; a phasor, as in any vector, is represented by a magnitude and a direction. Phasors are represented in polar co-ordinates with positive phase in quadrant I and negative phase in quadrant IV. Thus, positive angles are defined in the counter-clockwise direction. Voltages can be referenced to the current in a the series RLC; this results in voltages which are proportional to the impedance of each circuit element. V_{in} can be expressed as

$$V_{in} = \mathbf{I}R + j(\mathbf{I}\omega L - \mathbf{I}\frac{1}{\omega C}) \quad (7.14)$$

where I is the current in the series RLC circuit.

It is common for voltage drops due to inductance to be plotted on the positive imaginary axis and for voltage drops due to capacitance to be plotted on the negative imaginary axis. Resistance is plotted on the positive, real axis.

It is useful to see this applied to an example and we will continue with the RLC analyzed in the previous section. We originally referenced all signals to the input voltage, V_{in} , leading to a phase of 0° . It is more convenient to make V_R 0° . This can be done by subtracting 66.5° from each voltage phase term in equation (7.15), including V_{in} . The shifted version of equation (7.15) then becomes

$$\begin{aligned}
 V_R &= 0.4V_{in}e^{j0^\circ} \\
 V_L &= 0.22V_{in}e^{j90^\circ} \\
 V_C &= 1.14V_{in}e^{-j90^\circ} \\
 V_{in}e^{-j66.5^\circ} &= V_R + V_L + V_C
 \end{aligned}
 \tag{7.15}$$

where the voltage drop across the resistor is real while the inductor and the capacitor are purely imaginary.

The phasor plots for both the shifted and measured values are displayed in figure 7.4 below.

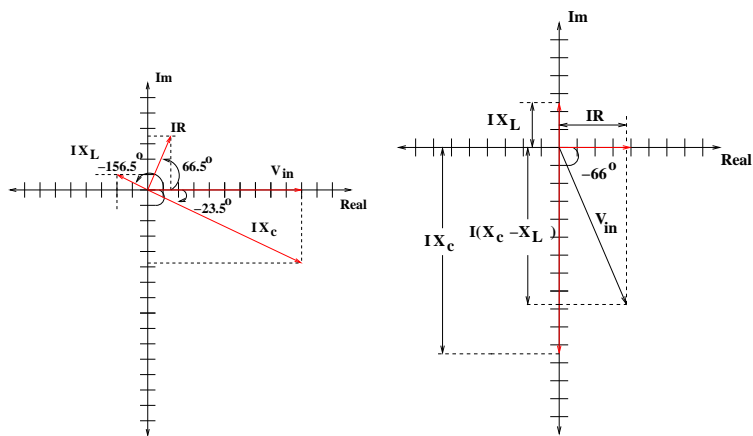


Figure 7.4: Phasor diagrams for $f_{in} = 10$ kHz.

Notice that the magnitude of the voltage drop across the capacitor is actually larger than the magnitude of the input signal.

7.3 Measurements

Obtaining measurements for V_L , V_C and V_R must be done with an oscilloscope. Channel 1 is connected to the input to display V_{in} and phase angles are all measured relative to V_{in} . Assuming the arrangement displayed in figure 7.1 the measurement of V_R can be done directly; however, the measurement of V_L and V_C cannot be done directly.

Measuring a direct voltage can only be accomplished when the circuit element measured is tied to ground. There are two solutions to find V_L and V_C . The first is to re-arrange the circuit twice placing the inductor and the capacitor where the resistor is thereby taking the output from V_L and V_C . The second is to keep the arrangement identical to that of figure 7.1, measure several points in the circuit and then match the values of those

measurements to what they should be via KVL (e.g. if KVL works ... it must be this value ...). Figure 7.5 describes what is needed for this approach.

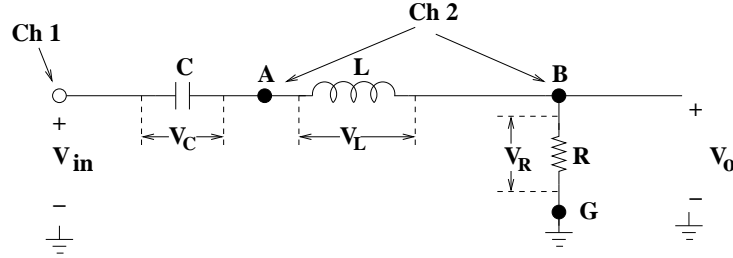


Figure 7.5: Measurement on series RLC circuit.

Notice the points, A , B and G in the circuit diagram in figure 7.5. Channel 1 of the oscilloscope always stays on the input signal. Two separate measurements are made with channel 2 at points B and A . The measurement at point B is the voltage drop across the resistor while the measurement at point A is the sum of the voltage drops across the resistor and the inductor. This can be written as

$$\begin{aligned} V_{BG} &= V_o = V_R \\ V_{AG} &= V_L + V_R \end{aligned} \quad (7.16)$$

where V_{BG} is the voltage drop across the resistor and V_{AG} is the combined voltage drop across the resistor and the inductor.

The values for the voltage drops can be found via subtraction between the 3 measurements (e.g. V_{in} on channel 1, V_{BG} and V_{AG} on channel 2). This is given by

$$\begin{aligned} V_R &= V_{BG} \\ V_L &= V_{AG} - V_{BG} \\ V_C &= V_{in} - V_{AG} \end{aligned} \quad (7.17)$$

These can be compared to values computed from KVL. Thus, pre-computing the values for V_R , V_L and V_C provides the necessary measurements.

It is worthwhile to make some general comments about phasors for the series RLC plotted in figure 7.4:

- i) the impedance of the capacitor and inductor are 180° out of phase;
- ii) the phase angle due to the capacitor is always negative while the phase angle due to the inductor is always positive;
- iii) the phase difference between the resistor and either the inductor or the resistor and capacitor will always be 90° .

The phasors plotted in figure 7.4 have been found by direct calculation. We should expect to get close the 90° phase difference between the resistor and reactive elements and/or the 180° phase difference between the capacitor and inductor. Another way to say this is¹:

$$\begin{aligned}\Delta t_C + \Delta t_I &= \frac{1}{2}T \\ \Delta t_R + \Delta t_I &= \frac{1}{4}T \\ \Delta t_R + \Delta t_C &= \frac{1}{4}T\end{aligned}\tag{7.18}$$

where T is the period of the input signal, Δt_C , Δt_I , & Δt_R are the relative time delays for the capacitor, inductor and the resistor.

You must also be aware that these relations assume ideal components. The most likely deviation will arise from the inductor at low frequencies. Thus, there will most likely be a slight deviation from $+90^\circ$ with the inductor due to the small real part. There are 4 measured values; thus errors made during the measurement of the period of the input signal and the time-differences converted into phase angles can skew the result beyond the slight deviation caused by the wire resistance of the inductor. Make an effort to be more precise when making these measurements.

7.4 Lab Instructions: Part A

- 1) Frequency Response: Measure the frequency response of the circuit in figure 7.1 using $R=5.6\text{ k}\Omega$, $L = 50\text{ mH}$ and $C = 0.001\text{ }\mu\text{F}$. Set the amplitude of V_{in} equal to $10 V_{peak}$. Note that $V_{out} = V_R$.
 - i) Make a table with the following columns: f , V_{in} , V_{out} Measured, V_{out} Calculated, $|A_v|^2$ Measured, $|A_v|^2$ Calculated, $A_v[dB]$ Measured, $A_v[dB]$ Calculated, Δt , T Measured, ϕ Measured and ϕ Calculated.
 - ii) Measure the attenuation for the following frequencies: .5 kHz, 1 kHz, 5 kHz, 10 kHz, 15 kHz, (3 points between 20-24 kHz), 25 kHz and 30 kHz. Also measure at $f = f_R$. Set the peak amplitude of the input sine wave to 10 volts.
 - iii) Complete the entries in the table.
 - iv) Plot $A_v[dB]$ Measured vs. $\log_{10}(f)$ and ϕ Measured vs. $\log_{10}(f)$. Overlay each graph with a plot of $A_v[dB]$ Calculated vs. $\log_{10}(f)$ and ϕ Calculated vs. $\log_{10}(f)$.

¹Assuming an ideal inductor !!!

- v) Identify the $3dB$ points on either side of f_R and estimate the bandwidth.
- 2) KVL/Phasor Plots: Using a $10 V_{peak}$ input sinusoid, verify KVL for frequencies of 10 kHz, 15 kHz and 25 kHz.
- i) Compute the values for V_R , V_C and V_L from X_L and X_C as was done in equations (7.9) through (7.15). Find both the rectangular and polar forms. Also compute the rectangular and polar forms for V_{BG} and V_{AG} identified in figure 7.5.
 - ii) Measure the period and amplitude of the input signal using the oscilloscope.
 - iii) Measure the amplitude and phase of V_{BG} and V_{AG} .
 - iv) Find V_R , V_L and V_C from the measurements of V_{BG} and V_{AG} .
 - v) Verify that KVL is satisfied.
 - vi) Plot the phasors based on the measurements.

7.5 Problems

- 1) Describe how a voltmeter which measures a *true rms* AC voltage could be used to demonstrate KVL.
- 2) Assume that the inductor has a wire resistance, (e.g. R_{Wire}). Rewrite equation (7.1) to include R_{Wire} . Find the values for $|A_v|^2$ in dB and ϕ for $f = f_R$.
- 3) Assuming that you have the amplitude and time differences (e.g. phase differences) measured for the resistor, capacitor and inductor is there an easier way to prove that the circuit satisfies KVL without the need for computations using phasor diagrams ?

- 4) In the section of the lab write up which provided measurement instructions it was mentioned that the impedance of the inductor might not be precisely $+90^\circ$ due to the wire resistance at low frequencies. Assuming this effect is noticeable will the phase become greater than or less than $+90^\circ$?

- 5) Given: $R = 5k\Omega$, $C = 10\mu F$, $L = 50mH$ & RLC connected in series
 - a) Find: the impedance, Z , of the series RLC at $f = 15kHz$; plot Z on the complex plane.
 - b) Find: the impedance, Z , of the series RLC at $f = 30kHz$; plot Z on the complex plane.
 - c) Find: the resonant frequency, f_R and ω_R .

7.6 Detailed Analysis of Series RLC Circuit

The series RLC circuit is a 2^{nd} -order, *linear system* which can be completely characterized by a 2^{nd} -order, *ordinary* differential equation with *constant* coefficients.

A differential equation is *any* equation which contains one or more *differentials*. The solution of a differential equation is an equation which contains no differential terms and is a function of the independent variable(s). A differential equation can be linear or non-linear (the former is of interest to us) and it can be *ordinary*, indicating one independent variable or *partial*, indicating more than one independent variable. The coefficients can also be functions of the independent variable or they can be constant. In our case, the latter is of interest.

The second order linear system provides a means of analyzing complex electrical and/or mechanical systems. Information about *stability*, *damping* and the efficiency can be related to the parameters of the system. These results provide fundamental results which predict the operation of any second order system. Moreover, the response of higher-order systems can often be characterized by multiple second-order systems.

The behavior of the series RLC circuit can be analyzed quantitatively by applying techniques in solving differential equations. Using KVL and substituting the voltage drops across each element as a function of the current we obtain,

$$V_{in} = V_C + V_L + V_R = \frac{1}{C} \int i dt + L \frac{di}{dt} + iR \quad (7.19)$$

Equation (7.19) contains both differential and integral terms, making it difficult to solve for directly. This can be simplified by differentiating through and re-arranging the terms resulting in

$$\frac{d}{dt}[V_{in}] = L \frac{d^2i}{dt^2} + R \frac{di}{dt} + \frac{1}{C}i \quad (7.20)$$

This relates the general differential equation to a specific input signal. We will show next how this circuit and earlier ones (namely the RC and RL circuits) can be described and generalized by first and second-order differential equations with constant coefficients. These differential equations are assumed to be both linear and ordinary.

7.7 1^{st} and 2^{nd} Order Differential Equations

The solutions to both 1^{st} and 2^{nd} Order Differential Equations can be expressed as exponentials. The solutions are subdivided into two classes. One

set of solutions provides the response of the system to an input of very short duration (or an input which changes over a very short duration). This is known as the *transient response*. The second solution provides the response of the system to a *specific* time-varying input signal. This is known as the *steady-state* response. The general solution of a differential equation is the sum of the transient and steady-state response. The transient response is due to changing conditions; the 'start-up', for example. The duration of the transient response is expected to die out quickly over time. The steady-state response is expected to last as long as the system input is present. To see this in a physical example, consider the shock absorber on a motorcycle. If you put enough weight on the fork to compress the shock absorber, it will bounce back up once you release it. It is possible it might bounce up, then slightly down and then stop. That is also fine. You would probably not feel safe if the fork bounced up and down for a very long time without an external force. You would expect it to stop quickly. This is analogous to the transient response. However, if you are riding the motorcycle over a continuous pattern of bumps, the shock absorber then would be expected to compress and expand in order to minimize discomfort to the rider. Once the motorcycle was on a smooth surface, the shock absorber would be expected to remain rigid. This is analogous to the steady-state response.

In general, stability of any system is found from the transient response. The response to a sudden, large input could be characterized as the transient response as well. How long the duration of the transient response depends upon the system itself. For example, a skyscraper might rock slightly in hurricane force winds; in fact, this might be incorporated in the actual design. However, you would not feel very safe if a high-rise tower continued to rock for long periods of time after a brief summer breeze. On the other hand, you do want the pendulum in a 'grandfather' clock to swing back and forth for long periods of time.

Although there is some math involved, you have been finding both the transient and steady-state responses of linear systems throughout the semester. During your investigation of RC and RL circuits in previous labs, you were really providing results for the transient and steady state responses of first-order systems. The exponential decay for the step response in the time domain was the transient response (the step was a repeating square wave). The frequency response and phase response later plotted in Bode diagrams. This was the steady-state response of those first-order systems. For the steady-state response, you measured the output amplitude and phase differences (with respect to the input waveform) for a selected range of frequencies for a *specific* input signal. In fact, in part (a) of this lab and in the lab where you measured the response of notch-filters (e.g. the twin-T filter), you were also plotting the steady-state response.

The general first-order differential equation is described by the following

expression

$$\frac{dx}{dt} + \alpha x = f(t) \quad (7.21)$$

where x is the dependent variable, t is the independent variable, α is the coefficient and $f(t)$ is the forcing function.

There are two solutions: (1) the homogeneous solution and the (2) specific solution. The homogeneous solution is found by setting $f(t) = 0$. Although this can be solved in a very straightforward manner, we will introduce the S -operator,

$$S^k = \frac{d^k}{dt^k} \quad (7.22)$$

where the S^k is substituted for the k^{th} differential.

If apply the S -operator to equation (7.21) the following result is obtained

$$(S + \alpha)x = 0 \quad (7.23)$$

assuming that $f(t)$ is zero.

The solution to equation (7.23), (e.g. $S = -\alpha$) is the exponent of the base- e , or

$$x(t) = Ce^{-\alpha t} \quad (7.24)$$

where C is a constant.

The constant C is found through initial conditions (e.g. $x(t = 0) = C$). The homogeneous solution of this first-order differential equation can be applied to the earlier output voltage and current responses obtained for the RC and RL circuits. If we substitute τ for α the same responses are easily found ($\tau = RC$ for the RC circuit and $\tau = \frac{R}{L}$ for the RL circuit).

The homogeneous differential equation for the series RLC circuit can be found from equation (7.20) by setting $V_{in} = 0$ which results in

$$\frac{d^2i}{dt^2} + \frac{R}{L} \frac{di}{dt} + \frac{1}{LC}i = 0 \quad (7.25)$$

In order to provide a more general analysis we would like to make some substitutions for the constant coefficients. Equation (7.25) can be re-written as

$$\frac{d^2i}{dt^2} + 2\alpha_1 \frac{di}{dt} + \alpha_2 i = 0 \quad (7.26)$$

where $\alpha_1 = \frac{R}{2L}$ & $\alpha_2 = \frac{1}{LC}$.

Applying the S -operator to equation (7.26) results in the following expression

$$[S^2 + 2\alpha_1 S + \alpha_2]i = 0 \quad (7.27)$$

This leaves us with a quadratic equation for S . The solution can be found by applying the quadratic formula, or,

$$S = -\alpha_1 \pm \sqrt{\alpha_1^2 - \alpha_2} \quad (7.28)$$

where the roots, S , are the natural frequencies^[3].

There are five possible solutions²[3]:

- i) **Under-damped** $\alpha_1^2 < \alpha_2$ The roots of equation (7.28) are complex conjugates. This results in a damped sinusoid.

$$\begin{aligned} v(t) &= e^{-\alpha_1 t} [C_1 \cos(\omega_d t) \pm j C_2 \sin(\omega_d t)] \\ v(t) &= C e^{-\alpha_1 t} \cos[(\omega_d) t \pm \phi] \end{aligned} \quad (7.29)$$

where $\omega_d = \sqrt{\alpha_2 - \alpha_1^2}$.

- ii) **Critically Damped** $\alpha_1^2 = \alpha_2$ The roots of equation (7.28) are real and repeated. This represents a change in behavior; the sinusoidal component is no longer part of the solution. This is a pure exponential decay.

$$v(t) = C_1 e^{-\alpha_1 t} + C_2 t e^{-\alpha_1 t} \quad (7.30)$$

- iii) **Over-damped** $\alpha_1^2 > \alpha_2$ The roots of equation (7.28) are real and distinct. This results in a pure exponential decay.

$$v(t) = C_1 e^{-\alpha_1 t} + C_2 e^{(\sqrt{\alpha_1^2 - \alpha_2}) t} \quad (7.31)$$

- iv) **Undamped** $\alpha_1 = 0$ The roots of equation (7.28) are purely imaginary and conjugates. There is no damping. This corresponds to replacing R with a short-circuit in the series RLC circuit.

$$v(t) = C_1 \cos\left(\sqrt{\alpha_2 - \alpha_1^2} t\right) \pm C_2 \sin\left(\sqrt{\alpha_2 - \alpha_1^2} t\right) \quad (7.32)$$

- v) **Unstable** $\alpha_1 < 0$

If the real part of equation (7.28) is positive then the amplitude of the solution rises exponentially. The response to an impulse or step rises uncontrollably; this is not physically realizable with the series RLC circuit but it is possible with other second-order systems.

²The differential equation is solved for $i(t)$; the output is taken at the resistor, hence $v(t) = Ri(t)$.

The cases of greatest interest are the under-damped, critically damped and over-damped transient responses. When the system is under-damped, the transient response is a damped sinusoid. This is a sine wave with the maximum amplitude decreasing as time increases. In fact, $v(t) \rightarrow 0$ as $t \rightarrow \infty$. The critically damped and over-damped transient responses are exponential decays and are very similar. The critical damping marks the change from a damped sinusoid to a pure exponential decay.

The remaining cases, undamped and unstable, are presented in order to have a complete set of transient responses. In the undamped case, $\alpha_1 = 0$, the transient response is an undamped sine wave. This is a system which will oscillate. When $\alpha_1 \neq 0$ there will always be a damping term. When $\alpha_1 < 0$, e.g. α_1 is negative, the exponential will have a positive exponent which will quickly lead to infinite values as $t \rightarrow \infty$.

7.7.1 Steady-State Response

The steady-state response is found by solving equation (7.20) for a specific function, V_{in} this is referred to as the *specific* solution. This is something we will not cover quantitatively. You should know qualitatively that the specific solution is related to the steady-state response by solving equation (7.20) for a specific input function. You should also know that the general solution of equation (7.20) is the sum of the solution of the homogeneous solution and the specific solution. Finally, you should know that the homogeneous solution leads to the transient response and the specific solution leads to the steady-state response.

7.7.2 General Transient Response

The homogeneous differential equation can be generalized. This is done in order to allow one to apply it to other types of second-order systems and also to simplify the interpretation of the resulting differential equation solutions. Equation (7.34) can be re-written as

$$\frac{d^2i}{dt^2} + 2\zeta\omega_R\frac{di}{dt} + \omega_R^2i = 0 \quad (7.33)$$

where $\zeta = \frac{R}{2}\sqrt{\frac{C}{L}}$ & $\omega_R = \frac{1}{\sqrt{LC}}$. Applying the S -operator to equation (7.26) results in the following expression using the S -operator, or,

$$[S^2 + 2\zeta\omega_RS + \omega_R^2]i = 0 \quad (7.34)$$

Once again, the solution can be found by applying the quadratic formula, or,

$$S = \omega_R \left[-\zeta \pm \sqrt{\zeta^2 - 1} \right] \quad (7.35)$$

The term ω_R is referred to as the undamped natural frequency, the term S is referred to as the natural frequencies and ζ is referred to as the damping constant. The transient response can be determined based upon the value of the damping constant. This leads to the following cases:

- i) **Under-damped** $\zeta < 1$; The roots of equation (7.35) are complex conjugates.

$$v(t) = Ce^{-\zeta\omega_R t} \cos \left[\left(\sqrt{1-\zeta^2} \right) \omega_R t + \phi \right] = Ce^{-\zeta\omega_R t} \cos (\omega_d t + \phi) \quad (7.36)$$

Note that $\omega_d = \left(\sqrt{1-\zeta^2} \right) \omega_R$.

- ii) **Critically Damped** $\zeta = 1$; The roots of equation (7.35) are real and repeated.

$$v(t) = C_1 e^{-\zeta\omega_R t} + C_2 t e^{-\zeta\omega_R t} \quad (7.37)$$

- iii) **Over-damped** $\zeta > 1$; The roots of equation (7.35) are real and distinct.

$$v(t) = C_1 e^{\omega_R(-\zeta-\sqrt{\zeta^2-1})t} + C_2 e^{\omega_R(-\zeta+\sqrt{\zeta^2-1})t} \quad (7.38)$$

- iv) **Undamped** $\zeta = 0$; The roots of equation (7.35) are purely imaginary conjugates.

$$v(t) = C_1 \cos (\omega_R) t \pm C_2 \sin (\omega_R) t \quad (7.39)$$

- v) **Unstable** $\zeta < 0$; The exponential term becomes infinite as $t \rightarrow \infty$.

This is identical to what was found in the preceding section. Figure 7.6 provides sample responses for two under-damped cases, a system near critical damping and a system which is over-damped. These responses will provide you with some insight into what the oscilloscope measurements should resemble.

7.8 Mechanical Systems

Mechanical systems can also be characterized by equation (7.33). To briefly describe how this is done we will pick the mass on a spring in figure 7.7 as an example[2].

Newton's second law and Hooke's Law can be used to describe the motion of the mass on a frictionless surface. This is expressed as

$$m \frac{d^2 x}{dt^2} = -kx \quad (7.40)$$

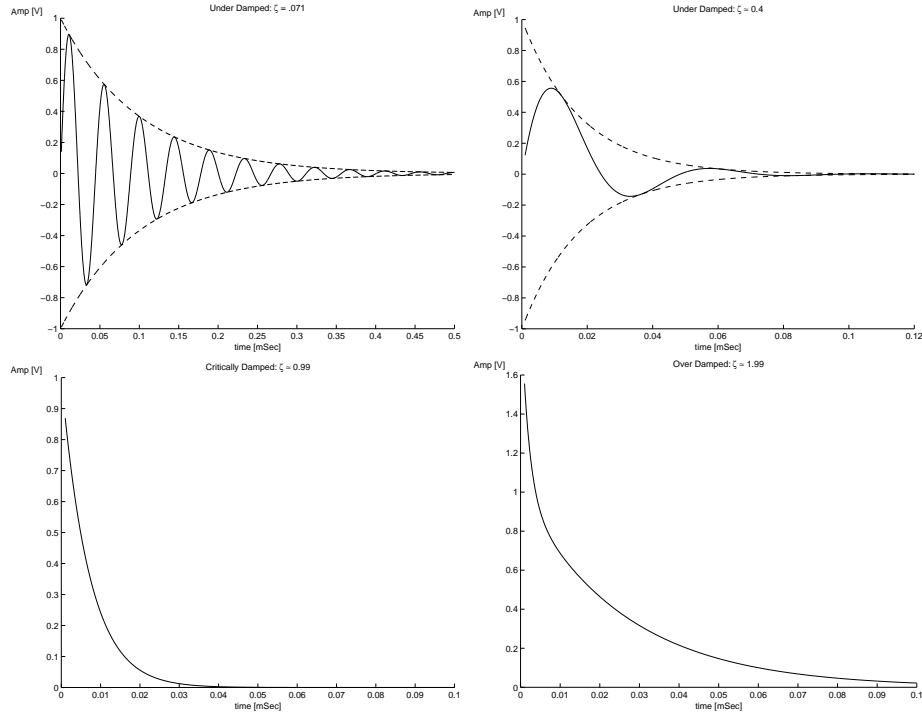


Figure 7.6: Examples of transient responses.

where m is the mass of the object connected to the spring and k is the spring constant.

Notice that in this case the spring will oscillate forever since the surface is frictionless; this is a simple harmonic oscillator. If we include a damping constant, b and a forced input, the resulting differential equation can be written as

$$\begin{aligned} F &= m \frac{d^2 x}{dt^2} + b \frac{dx}{dt} + kx \\ V &= L \frac{d^2 Q}{dt^2} + R \frac{dQ}{dt} + \frac{1}{C} Q \end{aligned} \quad (7.41)$$

where x is the displacement and Q is the charge.

The analogous differential equation for the series RLC circuit is given in the same equation to illustrate the relationship. In order to express this in terms of equation (7.33) the natural frequency and the damping constant are expressed in terms of b , k and m . This is given by

$$\begin{aligned} \zeta &= \frac{b}{2} \frac{1}{\sqrt{km}} \\ \omega_R &= \sqrt{\frac{k}{m}} \end{aligned} \quad (7.42)$$

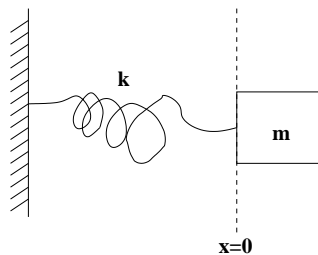


Figure 7.7: Mass on a spring.

Thus, the relationship between the mechanical system and the series RLC circuit can be found. The relationship between parameters of the series is given by [2]

$$\begin{array}{rcl}
 V & \longleftrightarrow & F \\
 Q & \longleftrightarrow & x \\
 i & \longleftrightarrow & \frac{dx}{dt} \\
 L & \longleftrightarrow & m \\
 \frac{1}{C} & \longleftrightarrow & k \\
 R & \longleftrightarrow & b
 \end{array} \tag{7.43}$$

One might ask why is it necessary to know anything about the analogy between mechanical and electrical systems. Surely, it is a topic which mechanical engineers and physicists know but the relationship to electrical engineering topics seems to be a bit misplaced.

The relationship between the second order mechanical system and the series RLC often occurs in the analysis and design of electro-mechanical sensors, such as transducers.

7.9 Measuring the Transient Response

The transient response can be measured by observing the response to a step function. This was analyzed in detail for the RC and RL circuits measured in labs 4 and 6. Since the response to a step input is required a square wave is selected because the oscilloscope is memoryless. The frequency of the square wave needs to be low enough to observe the full exponential decay.

The transient response for a 2^{nd} order system such as the series RLC circuit can be measured using an identical approach. The first step is to determine compute ω_R and ζ . This allows one to approximate the relative attenuation (f_{in} vs. f_R) and the type of decay (from the value of ζ).

7.10 Lab Instructions: Part B

1) Transient and Steady State Response: Construct the series RLC circuit displayed in figure 7.1 using $L = 50$ mH and $C = 0.001$ μF . Measure the transient response using the following values for R :

- i) $R = 1$ $k\Omega$.
- ii) $R = 5.6$ $k\Omega$.
- iii) $R = 13.36$ $k\Omega$ (5.6 $k\Omega + 12k\Omega || 22k\Omega$).
- iv) $R = 25$ $k\Omega$.

For each R value:

- i) Compute ζ , Q , ω_H , ω_L and the bandwidth, BW ; predict transient response (e.g. under-damped, over-damped or critically damped).
 - ii) Measure the transient response; record the amplitude of the output voltage across the resistor (on channel 2 of the oscilloscope) in a table with 2 columns: Δt and V_{peak} . Measure enough points to plot the transient response.
 - iii) Plot the transient response for each resistor value from the table. Label the value of ζ on each plot.
- 2) Find the frequency response of the series RLC circuit for each resistor value (except $R = 5.6$ $k\Omega$).
- i) Make a table with the following columns: f , V_{in} , V_{out} Measured, V_{out} Calculated, $|A_v|^2$ Measured, $|A_v|^2$ Calculated, $A_v[dB]$ Measured, $A_v[dB]$ Calculated, Δt , T Measured, ϕ Measured and ϕ Calculated.
 - ii) Measure the attenuation for the following frequencies: .5 kHz, 1 kHz, 5 kHz, 10 kHz, 15 kHz, (3 points between 20-24 kHz), 25 kHz and 30 kHz. Also measure at $f = f_R$. Set the peak amplitude of the input sine wave to 10 volts.
 - iii) Complete the entries in the table.

- iv) Plot $A_v[dB]$ Measured vs. $\log_{10}(f)$ and ϕ Measured vs. $\log_{10}(f)$. Overlay each graph with a plot of $A_v[dB]$ Calculated vs. $\log_{10}(f)$ and ϕ Calculated vs. $\log_{10}(f)$.

- v) Identify the $3dB$ points on either side of f_R and estimate the bandwidth.

7.11 Problems

- 1) Rewrite equations (7.33) through (7.38) in terms of R , L and C .
- 2) Find the conditions for under-damped, critically damped and over-damped solutions in terms of R , L and C .
- 3) What is the general relationship between ζ in the transient response and the Q of the frequency response ?

References

- [1] A. J. Diefenderfer, *Principles of Electronic Instrumentation*. W.B. Sanders, 1972.
- [2] A. M. Portis and H. D. Young, *Berkeley Physics Laboratory*. Hickok Teaching Systems, 1977.
- [3] M. Van Valkenburg, *Network Analysis*. Prentice Hall, 1974.

Chapter 8

Simple Op Amp Circuits

Lab Objectives:

- §1. Gain a practical understanding of basic op amp configurations.
- §2. Understand non-ideal effects and how to compensate for them.
- §3. Draw analogies between active and passive circuits.

Pre Lab:

1. Read this lab.
2. Read the data sheet on the LM741/747 op amp.
3. Review DC op amp theory from ELE212.

8.1 Introduction

This lab provides an introduction to simple op amp circuits. It is assumed at this point that you have covered the basics of DC operational amplifiers in your circuit theory course. The practical implementation of the simple circuits discussed will require additional passive devices and modifications to the intended circuits.

These differences occur because the op amp is truly a non-linear circuit which is configured to approximate the behavior of a linear amplifier. There are limitations which will often prevent one from directly realizing a particular circuit.

A good designer will make himself/herself familiar with the data sheet for the op amp selected. The data sheet provides the pin configuration, power supply requirements and parameters which describe the non-ideal effects of a given op amp. Thus, the data sheet will allow one to find the limitations of a given amplifier. Some of these non-ideal effects will be discussed in the following section.

8.2 The Non-Ideal Op Amp

The op amps used in practice are non-ideal; the non-ideal effects are measured and parameterized. Their values are supplied by the vendor in a data sheet. Some of these parameters are:

- i) **Finite Open Loop Gain** The amplifier open loop gain is actually finite. Moreover, its value will fall as $freq \rightarrow \infty$. A simple description of the open loop gain as a function of frequency is provided in figure 8.1.

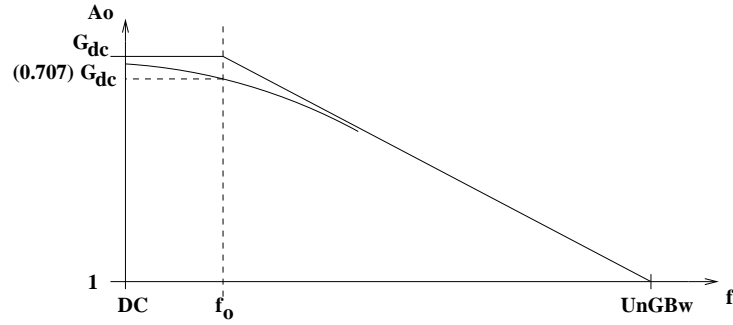


Figure 8.1: Typical open loop frequency response of an op amp.

The frequency response of the open loop amplifier in figure 8.1 approximates the actual behavior using a first order RC low pass filter with a large gain. Thus, at DC, the gain will be greater than unity; the gain is G_{dc} and this is the maximum value. At the cut-off frequency, f_o , the gain value becomes $\frac{G_{dc}}{\sqrt{2}}$ on a linear plot. This is equivalent to the $3dB$ point. The gain then falls at $20dB$ per decade or $6dB$ per octave which is also equivalent to the roll off of a simple RC low-pass filter. The open loop gain also drops to a value of 1 on the linear scale; this is also shown in figure 8.1. The frequency for which the amplifier gain is 1 on a linear scale (or $0dB$ on a Bode plot) is known as the *Unity-Gain Band Width*. This is also known as the *gain-bandwidth product* since it is defined as $G_{dc}f_o$. As the amplifier gain falls, the input impedance falls and the DC analysis techniques fail since they are only valid for $G, R_{in} = \infty$.

The 741 op amp has a DC gain of 200,000. At a frequency of $1kHz$, the gain drops to approximately 1,000. The unity-gain bandwidth is approximately $1MHz$.

- ii) **DC Offset Voltage** The simple DC model of the op amp is based on an ideal gain stage. One property of an ideal gain stage is that when the DC input voltage is zero then the DC output voltage is also assumed to be zero. The DC gain of a typical operation amplifier is on the approximately 10^5 while the power supply is approximately 10 volts. Thus, the minimum input voltage which can saturate the amplifier is $100\mu V$.

The 741 op amp has a typical DC offset voltage of 1-2 mV and a maximum DC offset voltage of $5mV$; more than enough to saturate the open loop amplifier. This is similar to most operation amplifiers. Thus, operation without feedback leads to saturation and instability. This makes it especially difficult to construct reliable DC circuits such as temperature sensors; it also makes it difficult for one to measure the open loop gain directly.

- iii) **Noise** Op amps have broad-band noise, that is, noise which is uncorrelated with frequency which extends across the usable bandwidth. The units of this noise is given in $\frac{\mu V}{\sqrt{Hz}}$ because its power scales proportionally with the bandwidth. This noise-figure is typically named input-referred noise. This means that the noise at the output is amplified along with input signals.
- iv) **Distortion** A non-linear effect present in every amplifier is distortion. For a sine-wave input, the output of an ideal linear amplifier would result in a sine wave scaled by the gain. In actual amplifiers sine-wave like waveforms occur at frequencies which are integer multiples of the input sine-wave frequency.
- v) **Common Mode Rejection Ratio (CMRR)** The gain of the common mode signal. The difference between the voltages at the inverting and non-inverting inputs is amplified by the open loop gain. The sum of these voltages, V_{cm} , which is the common-mode input would be non-zero if correlated noise were present. Thus, V_{cm} must be attenuated. $CMRR \rightarrow 0$ as $f \rightarrow \infty$.
- vi) **Rise Time** The rise time is defined as the time it takes for the output voltage to rise from 10% of its maximum value up to 90% of its maximum value. Rise time is measured for an op amp with feedback (e.g. a close loop configuration). This is shown graphically in figure 8.2.

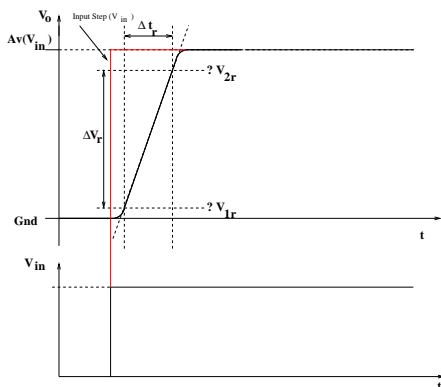


Figure 8.2: Rise time measurement of the response to a step input.

- vii) **Slew Rate** The slew rate is the rate in $\frac{V}{\mu\text{sec}}$ of the output voltage response to a step input. If Δt_R is the rise time, then the slew rate can be expressed as

$$\text{SlewRate} = \frac{\Delta V_R}{\Delta t_R} \quad (8.1)$$

where ΔV_R and Δt_R are shown graphically in figure 8.2.

8.3 The Inverting Amplifier

The inverting amplifier is shown in figure 8.3. Since it has feedback, its operation is stable.

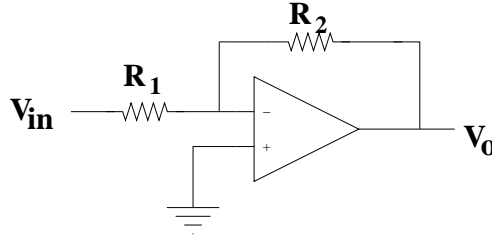


Figure 8.3: Inverting amplifier.

The amplifier gain is given by

$$A = \frac{V_o}{V_{in}} \approx -\frac{R_2}{R_1} \quad (8.2)$$

where R_2 and R_1 are the feedback and input resistors. Note that the \approx can be replaced by an $=$ when R_{in} and A_o are assumed to be large values; however the accuracy of this approximation is best at DC and it will deteriorate at higher frequencies. The usable operating frequency range is determined by the bandwidth and settling of the particular operational amplifier selected. In every case as the frequency is increased, the open loop gain will be reduced and the input resistance between the inverting and non-inverting input will also be reduced. This means that the assumption of equality between the inverting and non-inverting inputs is only acceptable at lower frequencies. As the input resistance of the operational amplifier is reduced the virtual ground is no longer a good assumption.

Even if we only operate at DC, there are still limitations. The input voltage must be kept low enough to maintain the following result

$$|V_o| < |V_{supply}| \quad (8.3)$$

which implies that

$$\frac{R_2}{R_1}|V_{in}| < |V_{supply}| \quad (8.4)$$

otherwise, the circuit will saturate at voltage close to or equal to the positive or negative supply voltage (sometimes referred to as the *supply rail* or the *rails*). Thus, unlike passive circuits, the input voltage of a circuit realized with an operational amplifier is restricted in amplitude in order to maintain linearity (e.g. equivalence to the linear circuit describing the operation). There is a restricted linear operating voltage range.

In order to maintain the DC operating assumptions, the operating frequency is also restricted; however, operation in a region where the bandwidth and settling effects are noticed does not necessarily mean that the operation will be non-linear, despite violating the DC assumptions.

8.4 The Integrator

The expression in equation (8.2) can be further generalized to include reactances or even impedances. There can be entire circuits lumped into a feedback term (e.g. R_2) and the feed-forward term (e.g. R_1). For example, the inverting amplifier frequency response can be altered by replacing R_2 and R_1 with complex impedances, Z_2 and Z_1 ,

$$A_v = \frac{V_o}{V_{in}} = -\frac{Z_2}{Z_1} \quad (8.5)$$

where Z_2 and Z_1 replace the resistors in figure 8.3.

The utility of the operational amplifier can be illustrated by realizing that Z_2 and Z_1 can each represent a series circuit, a parallel circuit or even a combination of series and parallel circuits. The expression for the transfer function given by equation (8.5) will hold as long as the R_{in} and A_o are large values. Thus, operation at frequencies where these assumptions do not hold must be avoided.

A simplified example comes from making the feedback purely reactive, e.g. $Z_2 = jX_2$ by replacing R_2 with a capacitor. This circuit is shown in figure 8.4.

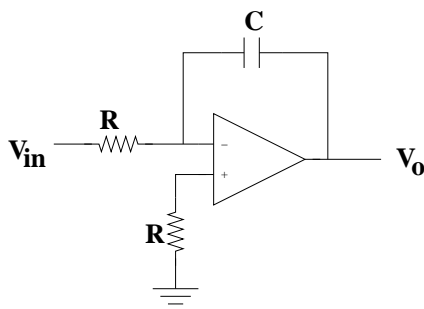


Figure 8.4: Pure integrator.

It is easy to show that

$$\frac{V_{in}}{R} \approx -C \frac{dV_o}{dt} (\approx \text{dueto finite gain}) \quad (8.6)$$

which leads to

$$V_o = -\frac{1}{RC} \int V_{in} dt \quad (8.7)$$

This circuit is known as a *pure* integrator since the output voltage will ALWAYS be integral of the input voltage. The transfer function of the pure integrator is given by

$$A_v(\omega) = \frac{j}{\omega RC} \quad (8.8)$$

Notice that the pure integrator differs from the RC low pass filter in two major areas. First, the gain of the RC low pass filter approaches unity for $f \ll f_o$ and is exactly equal to 1 for a DC input. The pure integrator gain is ∞ at DC, based upon equation (8.8), which means that it is ALWAYS unstable at DC. Second, the phase response of the RC low pass filter is zero at DC and becomes -90 degrees for $f \gg f_o$ while the phase response of the pure integrator is ALWAYS 90 degrees from equation (8.8). Since the RC low pass filter is only an integrator for $f \gg f_o$, problems with DC stability are avoided. Thus, even with very little knowledge of linear systems theory, one can see that a the DC stability problem must be addressed in the practical implementation of a pure integrator.

The DC stability problem will manifest itself into the circuit in figure 8.4 even if the designer decides to eliminate DC inputs by adding a blocking capacitor. Suppose that a blocking capacitor is placed in the input path? This will definitely prevent a DC offset from the signal generator. The problem is that any operational amplifier has an internal DC source; a non-zero offset voltage[1]. This effective of the offset voltage can be by separating the DC components from AC components. If we assume that the input voltage is composed of a DC voltage and AC voltages, then by superposition we can duplicate the circuit in figure 8.4 (one for DC inputs and one for non-DC inputs). If we then add the outputs we will (by superposition) obtain an output equivalent to that obtained from the single circuit composed of DC and non-DC inputs. Superposition is of course violated if it can be shown that the output of either circuit (DC or AC) becomes saturated (because it then becomes non-linear and superposition only holds for linear circuits). Non-linearity is undesirable because the output voltage will no longer be an amplitude-scaled, phase-shifted version of the input signal. The output signal will become quite distorted. Stated less formally, the *game's over* if the amplifier output *saturates* (e.g. flattens due to reaching the supply rails). Also note that when the mathematical expression for the output voltage becomes ∞ , this is equivalent to the output voltage of the real circuit saturating at or near the supply rails. Since we do not have infinite voltage supplies available, the output voltage reaches a value limited by the power supply. It is important to see that this is not a desired result.

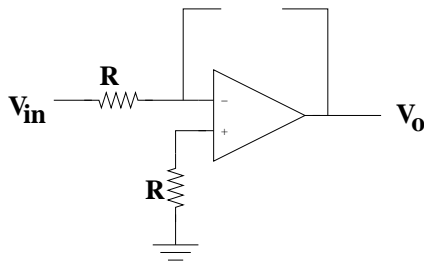


Figure 8.5: DC model of the pure integrator.

The DC model of the pure integrator shown in figure 8.5.

Notice that at DC, the feedback capacitor no longer provides feedback. At DC, the input sees the full open loop gain of the operational amplifier; this is a problem even if V_{in} is tied to the analog ground.

To see this, let's examine common operational amplifier specifications. Typically, the DC gain is on the order of 10^5 . The power supply voltages are on the order of $10V$. Offset voltages are approximately $1 - 10mV$. Thus, even in the best case, $10^{-3}x A_o < V_{Supply}$ the offset voltage alone will likely saturate the amplifier even for a gain of 10^4 and an offset voltage of $1mV$. The output voltage will be at $10V$ which means the supply voltage must be larger. In a protoboard implementation with a direct connection to a signal generator the offset of the signal generator will often add to the DC offset of the amplifier. In addition, low frequency noise components can also cause problems since the gain at low frequencies is still very high. Thus, there is a high sensitivity to noise in the low frequency range. The most common source of low frequency noise will be the power supply. While the low frequency noise pickup can be cleaned up with a better implementation of the board and improved bypassing of the supply voltages, the DC instability and the low frequency sensitivity will make it very difficult to obtain an undistorted sine wave from the pure integrator under the best possible conditions.

These problems can be overcome by modifying the circuit to create a *flattened* response at low frequencies. This can be accomplished by adding a resistor, R_2 , in the feedback path in parallel with the capacitor. Thus, a stable DC operating point is obtained. This is shown in figure 8.6.

The frequency response and phase response of the circuit in figure 8.6 are

$$\begin{aligned} A_v(\omega) &= -\left(\frac{R_2}{R_1}\right) \frac{1}{1+j\omega R_2 C} \\ \phi(\omega) &= 180 - \tan^{-1}(\omega R_2 C) \end{aligned} \quad (8.9)$$

which is similar to the RC low pass filter with a programmable gain (e.g. the gain at DC is set by $\frac{R_2}{R_1}$); the 3dB cutoff frequency is set by R_2 and C . Note however, that due to the inverting amplifier configuration that the transfer function is multiplied by a -1 ; this means that the phase response is shifted by 180 degrees.

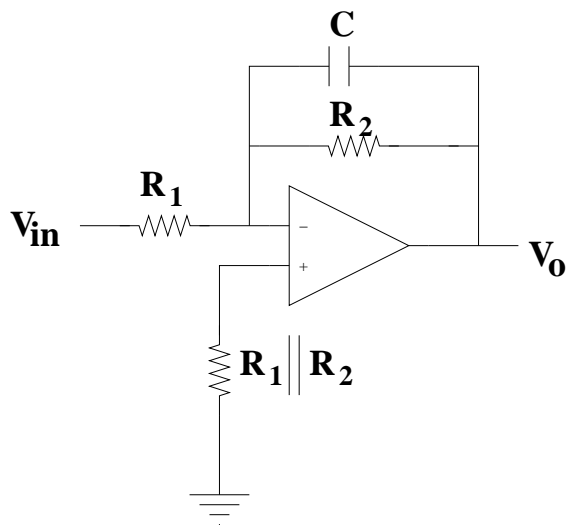


Figure 8.6: Active, first-order low pass filter.

At DC, the circuit in figure 8.6 is equivalent to an inverting amplifier. Since the circuit is derived from the inverting amplifier thus the negative sign causes the 180 degree phase shift.

8.5 The Differentiator

The circuit in figure 8.7 is known as the *pure* differentiator. This circuit can be derived from the pure integrator by swapping the position of the resistor and the capacitor, placing the resistor in the feedback loop.

The circuit in figure 8.7 will always yield

$$v_o = -RC \frac{dV_{in}}{dt} \quad (8.10)$$

hence, the name pure differentiator.

The transfer function of the pure differentiator is given by

$$Av(\omega) = -j\omega RC \quad (8.11)$$

It does not take much insight to realize that a circuit with this transfer function should always be avoided. Even without formal system theory one can intuitively assume that the stability of ANY linear system (which includes linear circuits) depends upon the gain remaining finite as the frequency approaches ∞ . Simply stated, in order to achieve stability the gain of a system must be *bounded* over ALL frequencies since instability is a very *bad* thing. Equation (8.11) demonstrates that the gain is infinite when ω approaches infinity.

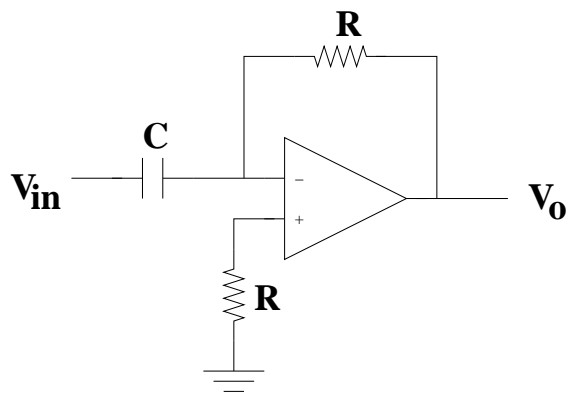


Figure 8.7: Pure differentiator.

There are practical problems with the pure differentiator. It will be extremely sensitive to high frequency noise pickup. Noise is always present in op amps. Input referred noise is amplified proportional to the rate of change at the input. Sudden changes in the input noise are amplified[1]. Since the gain is not bounded, the amplifier can easily saturate for noise at frequencies high enough to provide sufficient gain to reach the supply rails (e.g. the supply voltages). High frequency noise sources are difficult to compensate for unless the entire circuit is placed inside a conductive enclosure and properly grounded. Even with external noise sources eliminated, the internal noise of the amplifier, which is present at even high frequencies, will make reliable operation impossible.

In order to provide a more practical approach to the implementation of a differentiator, let's have a look at the RC high pass filter. The transfer function of the RC high pass filter is given by

$$A_v(\omega) = \frac{j\omega RC}{1 + j\omega RC} \quad (8.12)$$

Notice what happens as ω approaches infinity; the magnitude of the transfer function approaches 1 ! Thus, the gain is definitely bounded. What is the difference between this transfer function and equation (8.11) ? The answer: the addition of a denominator term which is equal to the numerator at high frequencies. The trade off is that the RC high pass filter is ONLY a differentiator at low frequencies because the gain is bounded.

Now, we need only work backward to find an implementation in an active circuit which will provide a transfer function with a denominator term. This results in the circuit shown in figure 8.8.

The frequency response and phase response are given by

$$\begin{aligned} A_v(\omega) &= -\left(\frac{R_2}{R_1}\right) \frac{j\omega R_1 C}{1 + j\omega R_1 C} \\ \phi(\omega) &= -90 - \tan^{-1}(\omega R_1 C) \end{aligned} \quad (8.13)$$

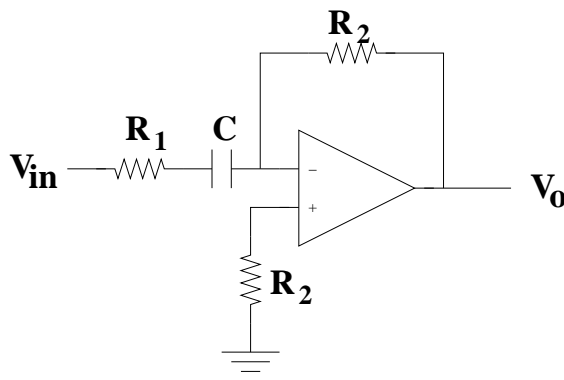


Figure 8.8: Active, first-order high-pass filter.

Equation (8.13) is equivalent to the RC high-pass filter with a programmable gain $\frac{R_2}{R_1}$; the 3dB cutoff frequency is given by R_1 and C . At high frequencies the response is flat.

It is worthwhile to compare the active HPF in figure 8.8 the passive RC HPF (from chapter 4). Notice that the simple RC high-pass filter phase response is 90 degrees at DC and approaches 0 degrees when $f \gg f_o$. The phase response of the RC HPF is equal to that of the phase response of the RC LPF shifted by 90 degrees (due to the j in the numerator). The HPF in figure 8.8 is merely the RC HPF with a shift of 180 degrees due to the negative sign picked up from the inverting amplifier configuration. The additional 180 degree shift leaves us with a phase response which ranges from -90 degrees at DC and approaches -180 degrees when $f \gg f_o$. The -90 is actually 270 degrees; however, it is renamed to -90 degrees because our phase co-ordinate system ranges from 0 to 180 degrees and 0 to -180 degrees. This is known as *wrap-around*. Also note that 180 degrees = -180 degrees since they are at the same point on the real axis.

8.6 Lab Instructions

For each part of the lab use the LM741 or LM747 op amp. The power supplies should be set to $\pm 15V$. You are expected to know how to configure the oscilloscope for the measurements of the low-pass and high-pass filters.

- 1) Inverting Amplifier: Construct the inverting amplifier shown in figure 8.3. Measure the V_o vs. V_{in} for the following DC input voltages: .01V, .03V, .1V, .3V, 1V, 3V, 5V, 10 V and 12 V.
Perform these measurements for each of the values for R_1 and R_2 :

- i) $R_1=10k\Omega$, $R_2=10k\Omega$.

ii) $R_1=5k\Omega$, $R_2=10k\Omega$.

iii) $R_1=1k\Omega$, $R_2=10k\Omega$.

Make a table with two columns: V_{in} and V_o . Plot V_o vs. V_{in} for each pair of R_1, R_2 values.

2) Low Pass Filter: Construct the low pass filter and set it up as shown in figure 8.6. Use $R_1=1k\Omega$, $R_2=10k\Omega$ and $C = .1 \mu F$.

i) Make a table with the following columns: f , V_{in} , V_o Measured, V_o Calculated, $|A_v|^2$ Measured, $|A_v|^2$ Calculated, $A_v[dB]$ Measured, $A_v[dB]$ Calculated, Δt , T Measured, ϕ Measured and ϕ Calculated.

ii) Measure the attenuation for the following frequencies: 10 Hz, 100 Hz, 1 kHz, 3 kHz, 10 kHz, 30 kHz and 100 kHz. Also measure at $f = f_o$. Set the peak amplitude of the input sine wave to 1 volt.

iii) Complete the entries in the table.

iv) Plot $A_v[dB]$ Measured vs. $\log_{10}(f)$ and ϕ Measured vs. $\log_{10}(f)$. Overlay each graph with a plot of $A_v[dB]$ Calculated vs. $\log_{10}(f)$ and ϕ Calculated vs. $\log_{10}(f)$.

v) Compute the cutoff frequency. Identify the $3dB$ point extrapolated from the measurements and overlay this with the computed value.

3) High Pass Filter: Construct the high pass filter and set it up as shown in figure 8.8. Use $R_1=8.2k\Omega$, $R_2=82k\Omega$ and $C = .001 \mu F$.

i) Make a table with the following columns: f , V_{in} , V_o Measured, V_o Calculated, $|A_v|^2$ Measured, $|A_v|^2$ Calculated, $A_v[dB]$ Measured, $A_v[dB]$ Calculated, Δt , T Measured, ϕ Measured and ϕ Calculated.

ii) Measure the attenuation for the following frequencies: 100 Hz, 1 kHz, 10 kHz, 100 kHz and 300 kHz. Also measure at $f = f_o$. Set the peak amplitude of the input sine wave to 1 volt.

iii) Complete the entries in the table.

iv) Plot $A_v[dB]$ Measured vs. $\log_{10}(f)$ and ϕ Measured vs. $\log_{10}(f)$. Overlay each graph with a plot of $A_v[dB]$ Calculated vs. $\log_{10}(f)$ and ϕ Calculated vs. $\log_{10}(f)$.

v) Compute the cutoff frequency. Identify the $3dB$ point extrapolated from the measurements and overlay this with the computed value.

8.7 Problems

- 1) Find expressions for the phase response of the pure integrator in figure 8.4 and the pure differentiator in figure 8.7. (Assume ideal op amps).
- 2) What happens to the high-pass filter at the high frequencies ? Explain it.

References

- [1] W. D. Stanley, *Operational Amplifiers with Linear Integrated Circuits*. New York: Macmillan, 3rd ed., 1994.

Chapter 9

Laboratory Problem: S-E Filter

Lab Objectives:

- §1. Characterize a 2^{nd} -order active filter by applying concepts of the general 2^{nd} -order system.
- §2. Assemble a 2^{nd} -order active filter.
- §3. Characterize the resonant frequency and Q-point as a function of critical components.

Pre Lab:

1. Read this carefully.
2. Read the data sheet on the LF353 op amp.
3. Review applicable lab write ups in ELE215.
4. Review material 2^{nd} -order systems in ELE212 & ELE215.

9.1 Introduction

Your project assignment is to construct and analyze the 2^{nd} -order active filter described in this write up[1, 2, 3]. You are to apply the lessons you have learned in ELE215 and ELE212. These lessons include the proper techniques for the construction and characterization of active and passive filters, the tabular description of the frequency and phase response and the Bode plot for the frequency and phase response.

We have found the attenuation and/or gain for numerous circuits, A_v . This is often referred to as the transfer function. The transfer function can be expressed in the time domain where time is the independent variable or it can be expressed in the frequency domain where ω is the independent variable. We know that the transfer function of a linear system can be found from the differential equation which describes it. We have worked out this problem in great detail many times for the 1^{st} -order RC, RL and active RC filters (all 1^{st} -order linear systems). We have also worked on this problem for the 2^{nd} order RLC filter.

We know from our earlier work that the transfer function derived from the differential equation is the linear sum of the transient and steady state response

of the system. The transient response is measured in the time domain using a step input. The steady state response is measured in the time domain using a sine-wave input. The frequency response can be found from the steady state response by plotting the amplitude and phase vs. the frequency. We have always done this using the Bode plot.

Passive RLC filters are always linear and thus, characterizing them as a linear system is always correct. As long as a system is linear superposition holds. Active filters approximate linear behavior when they are working within the linear operating range of the amplifier. The most critical condition to retain linear operation is bounding the input signal amplitude. Care must be taken to keep it within the limits of the power supply rails (e.g. ± 15 volts for our purposes). The frequency response of the op amp does not affect the linearity¹, because it can be characterized as a linear system (e.g. there is only a filtering effect) but the approximate linear characterization of the op amp could change dramatically at high frequencies. At high frequencies it becomes a *different* linear system².

9.2 Filter Description

The filter proposed by Sedra & Espinoza is shown in figure 9.1. This filter realizes a high-Q, resonant bandpass filter.

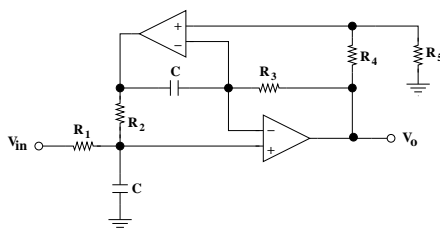


Figure 9.1: S-E filter[2].

Notice that there are no inductors in this circuit. The only passive components are resistors and capacitors. This is possible with operational amplifiers. The reactive response of the inductor is often replaced by operational amplifiers, resistors and capacitors because inductors are quite large and bulky. Also, inductors display many undesirable non-ideal effects at low frequencies.

The magnitude and phase response of this filter are given by
and

¹this assumes that the amplifier does not slew; e.g. the setting is *linear*

²same reasons as 2

9.3 Part Placement Assembly

The circuit for this project will use an 8 pin dual op amp. The schematic diagram in figure 9.2 is annotated with the pin numbers. There are two power supply connections not shown in the figure. The +15 and -15 Volt supplies are connected to pins 8 and 4, respectively.

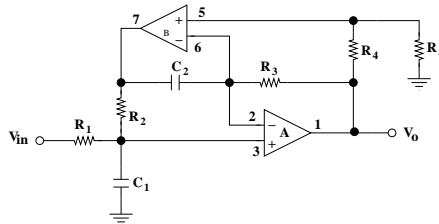


Figure 9.2: S-E filter[2].

Prior to wire-wrapping or the fabrication of a printed circuit board the designer needs to work out the part placement and wiring. This has been done for you and the resulting placement and wiring are shown in figure 9.3. Notice that there is a top view and a bottom view. Since this circuit will be realized on a wire-wrap board the bottom view shows all of the wiring.

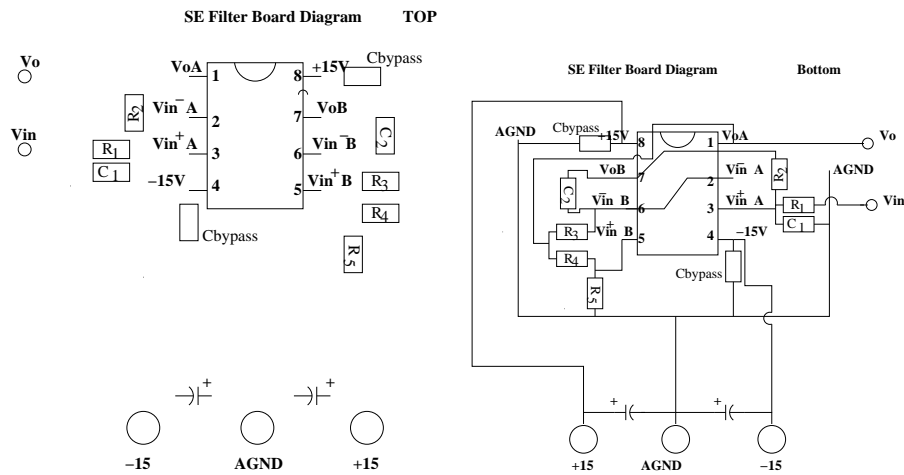


Figure 9.3: S-E filter board part placement & wiring (Top & Bottom).

9.4 Instructions

For the filter depicted in figure 9.1, follow the instructions and document the work in a lab report. You will use the LF353 op amp. The power supplies should be set to $\pm 15V$ (the power supply connections are not show in in figure 9.1).

1) Construct the circuit in figure 9.1. Use the following values:

- i) $R_1=22k\Omega$,
- ii) $R_2=1k\Omega$,
- iii) $R_3=1k\Omega$,
- iv) $R_4=1k\Omega$,
- v) $R_5=1k\Omega$,
- vi) $C=0.0068 \mu F$.

2) Measure the transient response and make a table with the following columns: V_{in} , V_o and t . Select at least 15 values; I expect to be able to reliably see the response from the values selected.

3) Make a detailed plot on a linear scale of the transient response.

4) Find the following parameters:

- i) ω_d ,
- ii) α ,
- iii) and ζ .

You will need to apply the appropriate waveform and select the frequency to enable you to see the transient response.

5) Measure the frequency response. Make a table with the following columns: f , V_{in} , V_o , $|A_v|^2$, $A_v[dB]$, Δt , T , and ϕ .

You will be expected to select the frequencies. I expect that you make 20 measurements for the frequency response. I also expect at least 5 measurements around the resonant frequency.

6) Plot $A_v[dB]$ vs. $\log_{10}(f)$ and ϕ vs. $\log_{10}(f)$.

7) Find the following parameters:

- i) Maximum gain,
- ii) ω_R ,

- iii) and Q ,
- 8) For a sine wave at f_R , make a table with the following columns: $|V_{in}|$, $|V_o|$. Select 10 values; the range of $|V_o|$ should be from 500 mV up to 15 V.
- 9) Plot $|V_o|$ vs. $|V_{in}|$ using this data.
- 10) Do the following:
 - i) Input a square wave at the resonant frequency.
 - ii) Plot the signal on the scope (Amplitude vs. time).
 - iii) Mathematically describe the signal based on the measurements.
 - iv) Compare the amplitude of the output signal to that of the input square wave.
 - v) Gradually lower the frequency of the input square wave.
 - vi) What happens ? At the first interval where you again see a waveform, plot the waveform seen on the scope (Amplitude vs. time). RECORD THIS FREQUENCY !!!
 - vii) Mathematically describe the signal based on the measurements.
 - viii) Compare the amplitude of the output signal to that of the input square wave.
 - ix) Repeat (v) through (viii) again for the next lower frequency.
- 11) Repeat part 10 using a sawtooth waveform as the input signal.
- 12) Replace R_1 with a 5.6 k Ω resistor. Repeat parts 1 through 7.

9.5 Questions

- 1) Is the system critically damped, over-damped or under-damped or neither ?
- 2) What is the *linear* amplitude range for signal at resonance ?
- 3) What the components of the square wave based upon the output signals measured in part 10 ?
- 4) What the components of the sawtooth waveform based upon the output signals measured in part 10 ?
- 5) From the measurements in parts 1 through 7 what can be determined from the relationship between R_1 and the Q and R_1 and the damping ?

- 6) Is there a relationship between Q and ζ ? For just this case or for all 2^{nd} -order systems?

References

- [1] A. G. Sadasiv, “ELE 215 Lab: Op amps - part 2.” University of Rhode Island, no date.
- [2] A. S. Sedra and J. L. Espinoza, “Sensitivity and frequency limitations of biquadratic active filters,” *IEEE Trans. on Circuits and Systems*, vol. CAS-22, pp. 122–130, February 1975.
- [3] R. Schaumann, M.A.Soderstrand, and K. Laker, *Modern Active Filter Design*. IEEE Press, 1981.