CALIFORNIA STATE UNIVERSITY
NORTHRIDGE

ELECTRICAL ENGINEERING FUNDAMENTALS LABORATORY EXPERIMENTS 1-12 ELECTRICAL ENGINEERING ECE 240L LABORATORY MANUAL

Prepared
By: Benjamin Mallard

Department of Electrical and Computer Engineering

(Revised 12/11)

Introduction

The main objective of this manual is to relate the theory of circuit analysis and practice. Each laboratory experiment consists of the "equipment needed", theory, preliminary calculations and procedure. Students are required to review the theory and complete the preliminary calculations before the lab period and perform the actual experiment. The theory is presented in brief and is to assist students with basic equations and information needed to complete the calculations and perform the measurements. The last experiment in this manual is optional. It is intended for students to gain design experience in using the theories, practices, and measurement techniques used in the prior experiments.

ECE 240 EXPERIMENTS

1) LABORATORY INSTRUMENTS AND REPORTS
2) OSCILLOSCOPES
3) DC CIRCUITS
4) COMPUTER SIMULATION
5) NETWORK THEOREMS
6) OPERATIONAL AMPLIFIERS
7) FIRST ORDER CIRCUITS
8) SECOND ORDER CIRCUITS
9) IMPEDANCE AND ADMITTANCE
10) FREQUENCY RESPONSE
11) PASSIVE FILTERS
12) DESIGN EXPERIMENT

EXPERIMENT ONE
LABORATORY INSTRUMENTS AND REPORTS

1) EDUCATIONAL LABORATORY VIRTUAL INSTRUMENTATION SUITE (ELVIS)
2) BREADBOARD STRUCTURE AND LAYOUT
3) POWER SUPPLIES
4) GROUNDING
5) DIGITAL MULTIMETERS
6) OSCILLOSCOPES
7) FUNCTION GENERATORS
8) LABORATORY REPORTS

<u>ELVIS FEATURES AND LAYOUT</u>

Insert the Prototyping Board (breadboard) into slots on top of the Benchtop Workstation.

Find and identify the following features on the Benchtop Workstation (Front Panel):

1) Prototyping Board Power Button

2) System Power Indicator Light

3) Communications Bypass/Normal Switch

4) Variable Power Supply Switches and Knobs

5) Function Generator:
    - **i)** Manual Select Switch
    - ii) Waveform Select Switch
    - iii) Frequency Select Knob
    - iv) Fine Frequency Adjustment Knob
    - v) Amplitude Adjustment Knob

6) Digital Multimeter (DMM):
    - i) Input Current Measurement Terminal (HI)
    - ii) Input Current Measurement Terminal (LOW)
    - iii) Input Voltage Measurement Terminal (HI)
    - iv) Input Voltage Measurement Terminal (LOW)

7) Oscilloscope (SCOPE):
    i) Channel A Input (CH A)
    ii) Channel B Input (CH B)
    iii) External Trigger Input (TRIGGER)

Find and identify the following features on the Prototyping Board:

1) Analog Pins:
    i) Differential Analog Inputs (6)
    ii) Analog Sense Input
    iii) Analog Ground

2) Oscilloscope Pins:
    i) Oscilloscope Signal Inputs
    ii) Oscilloscope Trigger Input

3) Programmable Function Pins:
    i) Programmable Functions Inputs (5)
    ii) Scan Clock (Scanclk)
    iii) Reserved (External Strobe Pin)

4) Digital Multimeter (DMM) Pins:
    i) 3 Wire Transistor Measurements (3-Wire)
    ii) Input Current Measurement Pin (+)
    iii) Input Current Measurement Pin (-)
    iv) Input Voltage Measurement Pin (+)
    v) Input Voltage Measurement Pin (-)

5) Analog Outputs (2)
    i) DAC0
    ii) DAC1

6) Function Generator
    i) Output Signal (FUNC_OUT)
    ii) Synchronization Signal (SYNC_OUT)
    iii) Amplitude Modulation Input (AM_IN)
    iv) Frequency Modulation Input (FM_IN)

7) User Configurable I/O (8):
    i) Banana Connector A (BANANA A)
    ii) Banana Connector B (BANANA B)
    iii) Banana Connector C (BANANA C)
    iv) Banana Connector D (BANANA D)
    v) BNC Signal 1 (BNC 1+)
    vi) BNC Signal 1 Ground or Reference (BNC 1-)
    vii) BNC Signal 2 (BNC 2+)
    viii) BNC Signal 2 Ground or Reference (BNC 2-)

8) Variable Power Supplies:
    i) Positive Output of 0V to 12V (SUPPLY+)
    ii) Power Supply Ground or Common (GROUND)
    iii) Negative Output of 0V to 12V (SUPPLY-)

9) DC Power Supplies:
    i) Fixed +15V Output (+15V)
    ii) Fixed -15V Output (-15V)
    iii) Power Supply Ground or Common (GROUND)
    iv) Fixed +5V Output (+5V)

Although there are more features on the Prototyping Board, the connections, pins, and terminals listed above are the only items of importance for the ECE240 Laboratory.

BREADBOARD STRUCTURE AND LAYOUT

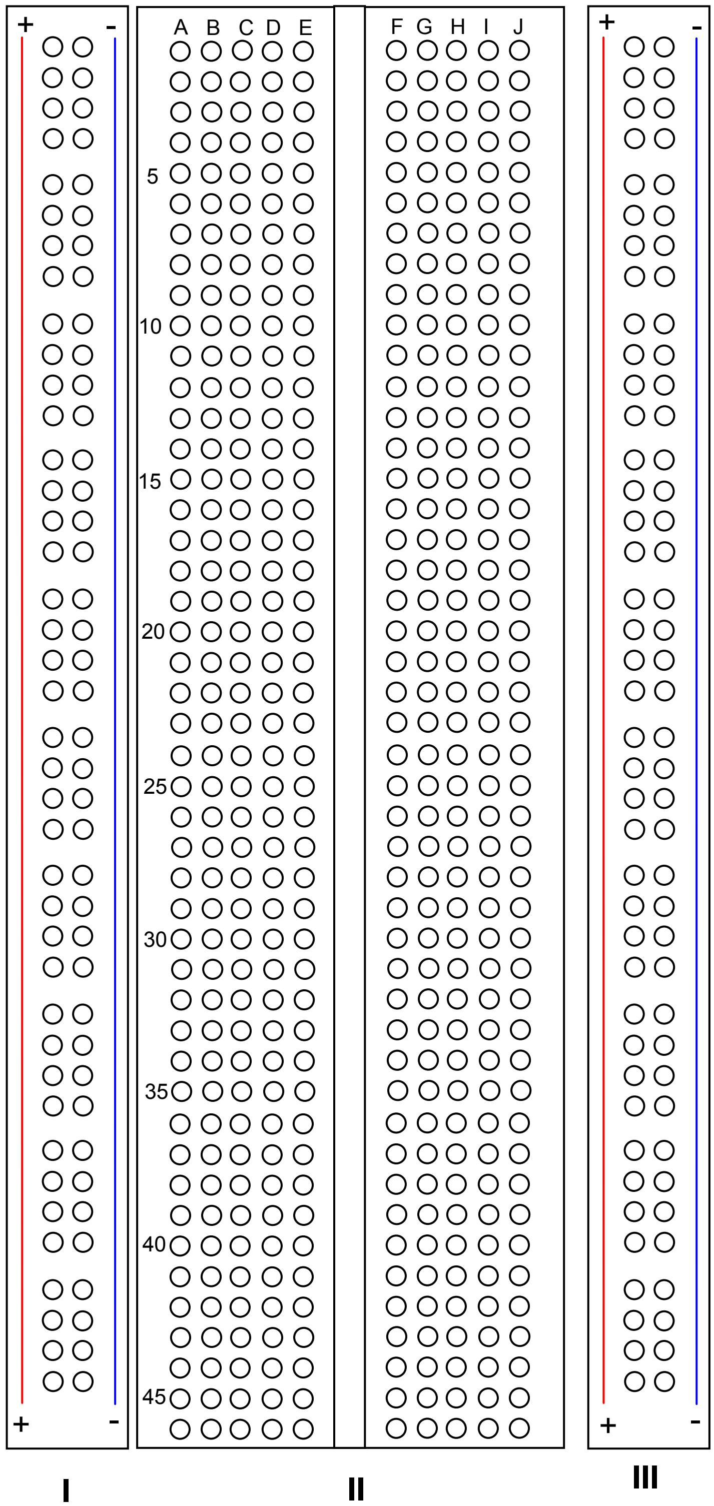


Figure 1.1

The diagram shown on the previous page is that of the circuit construction area on the prototyping board. In the laboratory, this area is used to construct a preliminary circuit according to the circuit schematic or wiring diagram. An explanation on its use is as follows.

SECTIONS I and III

The holes in these sections are connected vertically. Moreover, the column of holes between the '+' signs are connected. These are the holes adjacent to the red vertical stripe. Likewise the holes located between the '-' signs are connected. These holes are adjacent to the vertical blue line. There is no electrical connection of these holes horizontally. It is suggested that the holes in this section be reserved for direct connections to laboratory equipment. This enables the user to distributed common connections throughout the circuit that is suggestively constructed in section II.

SECTION II

The holes at this location are electrically connected horizontally. Each row of holes is connected on each side of the groove or trough. The groove isolates one row of holes from the other horizontally. There is no electrical connection of any of these holes in the vertical direction. Each column of holes in this section is not connected. Since there are more holes in this section, construction of the circuit to be analyzed or designed is reserved for this area.

Electrical components are placed in holes where other connections can be made. An example is shown below.

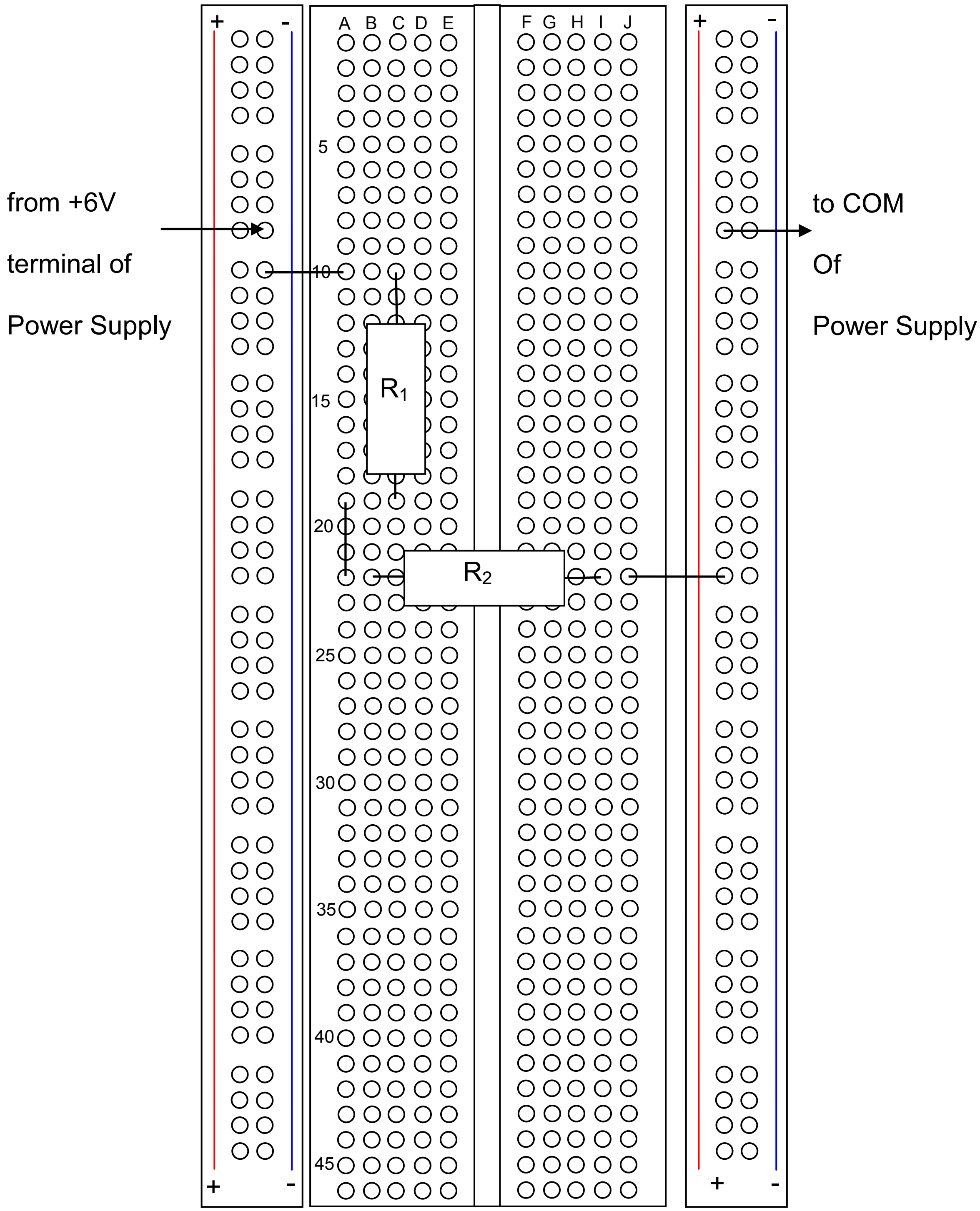


Figure 1.2

Schematic (circuit diagram)

A schematic is a drawing of a circuit comprised of symbols and lines representing the connection scheme. In figure 1.3, the circuit connection shown in figure 1.2 is illustrated.

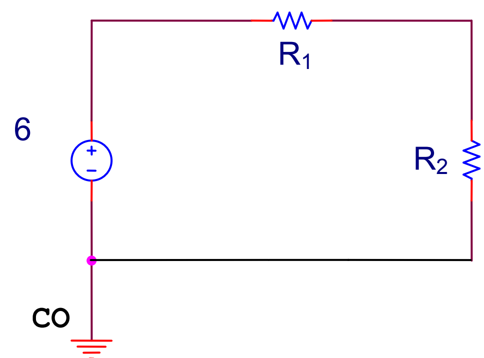


Figure 1.3

Circuit Layout

The diagram shown below depicts what is called a circuit layout. This type of diagram shows physical components, the terminals associated with them, and a wiring scheme showing how each component of the circuit is connected. In the diagram below, the triple power supply is connected in series with resistors R_1 and R_2. This depiction is the associated circuit layout of the circuit shown schematically above in figures 1.2 and 1.3. The series loop terminates on the COM (ground) terminal of the power supply thus completing the circuit.

All power supplies have a "power on" switch that allows electricity to flow into the component from the wall or bench supply. Locate this switch and become familiar with its locality. The power supply also has indicator lights, LEDs, and numerical displays which provide information regarding over voltage and short circuit current protection, and values of dialed voltage and current respectfully. Locate these features and become familiar with them.

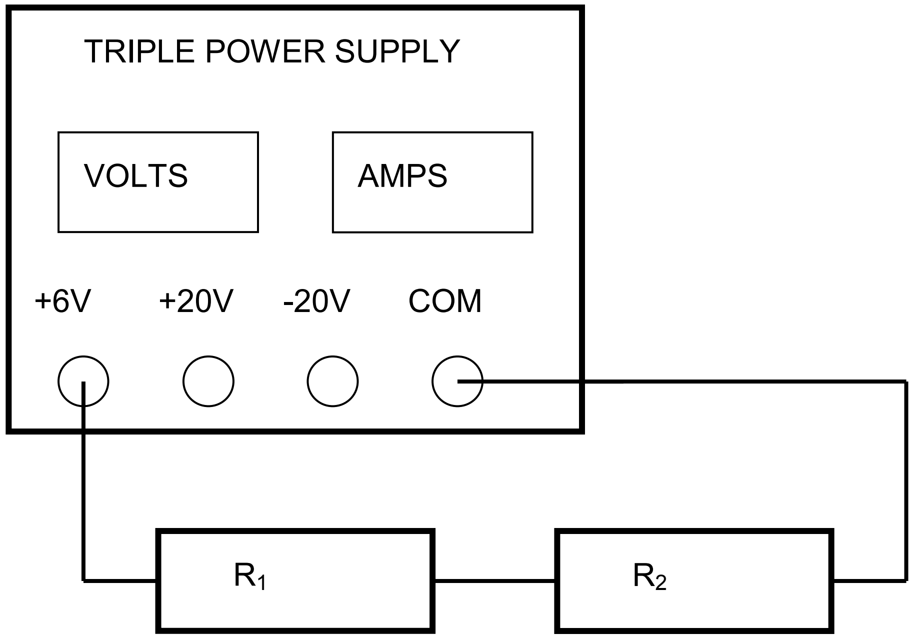


Figure 1.4

GROUNDING

The concept of grounding is quite simple. Although some may think grounding is a concept whereby a path is chosen for electrons (or charge carriers in general) to disappear into the physical or earth ground, nothing is farther from the truth. The ground point in any circuit is a reference point or connection from which all voltages are measured. The ground node in a circuit by convention has a voltage of zero. Therefore the voltage measured at any other node registers a measurement of the voltage “difference” from that node to the reference or ground node.

In establishing the ground node in a circuit, typically this node is located at the power source of the circuit. On all power supplies this is labeled as the COM socket or connector as mentioned previously. After the ground node has been designated in a circuit schematic, this node should physically be the COM connection on the power supply of the circuit. If more than one power supply is used, then all COM connectors of each power supply should be connected together.

PROCEDURE

1. Turn the power supply off. Make sure there is nothing connected to the output voltage terminals.

2. Connect one banana lead to the +6V output of the power supply and the other end to the voltage input of a digital multimeter. Connect another banana lead to the COM terminal of the power supply and the other end to the COM terminal of the digital multimeter.

3. On the power supply, turn all voltage adjustment knobs all the way to the left.

4. Turn on the power supply and digital multimeter. Make sure the voltage display on the power supply and the digital multimeter reads approximately 0 volts. The amp display on the power supply should read 0 amps.

5. Slowly turn the voltage adjustment knob to the right until the digital multimeter displays +6V.

6. Turn the power supply off.

7. Connect the power supply to the circuit illustrated in the circuit diagram above. Make sure the COM terminal of the power supply is connected to R_2.

8. Turn on the power supply.

9. Measure voltages using the digital multimeter at each resistor lead and record them.

10. Turn off the power supply.

11. Repeat steps 1 through 10 with power supply voltage outputs of +5V, +3V, and +1V.

12. Turn on the computer and select the NI ELVIS icon.

13. From the NI ELVIS menu on the monitor, select the DIGITAL MULTIMETER.

14. Turn off the Benchtop Workstation power supply. Verify that the power indicator light is off. Connect a wire from the 0V to 12V (SUPPLY+) output pin of the prototype board to the HI input voltage measurement pin of the digital multimeter. Connect another wire from the Power Supply Ground or Common (GROUND) pin of the prototype board to the LOW input voltage measurement pin of the digital multimeter.

15. On the front panel of the Benchtop Workstation, turn all voltage adjustment knobs to the zero volt setting.

16. Turn on the Benchtop Workstation power switch. Make sure the voltage display on the DMM window reads approximately 0 volts. The amp display should read 0 amps. Select the NULL icon on the DMM virtual instrument to reset the reading to zero volts if necessary.

17. Slowly turn the voltage adjustment knob to the right until the digital multimeter displays +6V.

18. Repeat step 17 for +5V, +3V, and +1V.

DIGITAL MULTIMETER

The digital multimeter or DMM is used to measure electrical quantities of voltage, current, and resistance. For voltage and current, the meter measures both DC and AC quantities. For AC voltages and currents, the meter measures what is typically called RMS quantities. These measurements are internally calculated by the meter using the following formulas for sinusoidal voltages and currents.

$$V_{RMS} = \frac{V_P}{\sqrt{2}}, \qquad I_{RMS} = \frac{I_P}{\sqrt{2}} \qquad (1.1)$$

V_P is the peak voltage and I_P is the peak current of an AC waveform or signal.

Resistance is measured in units of ohms and kilohms. The following procedure explains how these measurements should be performed.

1. Insert one banana lead into the volt/kΩ socket on the front of the multimeter. Insert another banana lead into the COM socket.
2. Place alligator clips on the ends of both leads and connect them to opposite leads of the resistor to be measured.
3. Select the kΩ measurement push-in button on the front of the multimeter.
4. Beginning with the scale selected at the lowest range of measurement, push in the scale buttons until a stable reading appears on the display.
5. Record the measurement.

Measuring resistance using the ELVIS is performed as fellows:

1. Place the resistor to be measured into an appropriate pair of holes on the prototyping board.
2. Connect a wire from one lead of the resistor to the + input current measurement pin. Connect another wire from the other lead of the resistor to the – input current measurement pin.
3. In the DMM window of the ELVIS software, select the ohmmeter measurement feature.
4. Using the measurement scale selection, find the setting that provides the largest resolution as possible.
5. Record the measurement.

OSCILLOSCOPES

The oscilloscope is a laboratory instrument that allows viewing of time-dependent signal voltages. Each bench is supplied with an analog and a digital oscilloscope. The analog oscilloscope provides a visual representation of a signal on a voltage versus time axis. This axis is viewed on the display portion of the instrument. From this display, the experimenter is able to measure parameters such as:

- Peak voltage
- Frequency
- Period of oscillation
- Time constants
- Phase angle

The digital oscilloscope allows the user to measure the same quantities with the added bonus of selected menus that automatically measure and display them as well. The following procedure allows the user to become familiar with these features.

ANALOG OSCILLOSCOPE

1. Locate the power button and turn it on.

2. Locate the two channel modules on the oscilloscope (channel 1 and channel 2).

3. On both channels, select the ground reference position.

4. Observe the Voltage Sensitivity knobs for channel 1 and channel 2. Record the settings for each knob position.

5. Locate the Time Base module. Record the settings for each knob position.

DIGITAL OSCILLOSCOPE

1. Locate the power button and turn it on.

2. Locate the two channel modules on the oscilloscope (channel 1 and channel 2).

3. On both channels, select the ground reference position.

4. Observe the Voltage Sensitivity knobs for channel 1 and channel 2. Record the settings for each knob position.

5. Locate the Time Base module. Record the settings for each knob position.

6. Locate and observe the menu buttons on the right side of the oscilloscope. Record the different options for each button.

On the ELVIS, select the OSCILLOSCOPE virtual instrument and perform the same steps described for the DIGITAL OSCILLOSCOPE.

FUNCTION GENERATORS

Like the power supply, function generators are also a voltage source for any passive circuit. In order to see the output voltages from the function generator an oscilloscope must be used. The following are knobs on the function generator which provide certain functions.

Amplitude – peak voltage of the output waveform
Frequency – periodicity of the output waveform
Sine – sine waveform
Square – square waveform
Triangular – triangular waveform
Offset – DC bias level
Multiplier – frequency range selector

1. Connect a double BNC cable to the OUT terminal of the function generator and the other end to the channel 1 input on the oscilloscope.

2. Turn on the function generator and the oscilloscope.

3. Select the SINE waveform on the function generator.

4. Adjust the AMPLITUDE and FREQUENCY knobs on the function generator until a sine wave appears on the oscilloscope screen.

5. Turn the OFFSET knob on the function generator and record your observation.

6. Select different MULTIPLIER settings on the function generator and record your observation.

Repeat steps 1 through 6 selecting the SQUARE and TRIANGULAR waveforms.

On the ELVIS system, select the FUNCTION GENERATOR virtual instrument and perform the same steps as listed above.

LABORATORY REPORTS

The laboratory report is the most important document in this class. The contents of this report should not only tell the reader what the experimenter did, but also allow the reader to duplicate the experiment in complete detail. The aim of the laboratory report is to provide information into the examination of a theory or formula in order to confirm its validity or to expose deviations from its exactness. In addition it also contains details on the process of experimentation. A generic structure is presented here as a guide to help the student develop sound investigative reporting skills.

INTRODUCTION

All reports should begin with an introduction that describes what the experiment is about and why certain items are being investigated (e.g. – Ohm's Law, Thevenin's Theorem, etc.). It should be stated here what the outcome or results of the experiment would be. This section should only be a few sentences which should be written in a manner to interest or entice the reader to peruse the document.

PROCEDURE(S) or METHOD(S)

The student should use his or her own words in this section to describe the strategy of how the experiment or investigation was performed. In this section a brief history may be included to educate the reader on what past procedures were used to conduct the same or similar experiments. This section of the report should also include a list of major lab equipment that was used in the experiment. Circuit schematics or diagrams, connection layouts including equipment, is also useful in describing how an item was tested or examined.

RESULTS and/or OBSERVATIONS

This section should be the most voluminous part of the report. It is extremely important to relate and document every measurement and observation here. To organize these data tables, charts, graphs, etc. may be used. Software tools such as spreadsheets and data bases are commonly used but not required.

DISCUSSION

This is the section where the student earns his or her grade essentially. Here the components of the previous section are explained, validated, or not. It is very important for the student to express his or her true thoughts on how the experiment provided added knowledge into the field of electrical engineering or not. If the results are erroneous, provide reasons/justifications for errors and state how to produce the correct results.

CONCLUSION

This section can be combined with the previous section if desired. It is meant to provide a summary of the outcome and results of the experiment in order to leave a last impression on the reader.

EXPERIMENT TWO
OSCILLOSCOPES

EQUIPMENT NEEDED:

1) Analog Oscilloscope
2) Function Generator
3) Resistors
4) ELVIS

THEORY AND PRELIMINARY PREPARATION

Review the oscilloscope exercises from experiment 1. Your instructor will give a brief lecture on the use of the analog, digital, and ELVIS oscilloscopes.

Procedure

1) Set the frequency dial of the function generator to 1 kHz. Push the waveform select button to output a sine wave. Display the sine wave on channel 1 of the analog oscilloscope. Set the voltage sensitivity to 5V/DIV. Adjust the amplitude knob on the function generator to produce a 2 volt peak (4 volts peak-to-peak) waveform on the oscilloscope.

   a) Sketch the display to scale after setting the time base on the oscilloscope to 1 ms/div, 0.1ms/div, and 10μs/div. Be sure to label the voltage and time axes with accurate divisions.

   b) Set the time base on the oscilloscope so that one or two cycles of the sine wave is visible. Measure the period of the waveform. Record this as T_{meas}. Calculate the frequency of oscillation. Record this as f_{meas}. Compare this calculation with the frequency setting of the function generator.

   c) Calculate the percentage error between the frequency setting of the function generator f_{dial}, and f_{meas} using the following formula:

$$\%\,error = \left(\frac{f_{meas} - f_{dial}}{f_{dial}}\right) \times 100\% \qquad (2.1)$$

   d) Measure the voltage of the sine wave using the digital multimeter. Connect the sine wave output to the AC voltage input of the digital multimeter. Record this measurement as the RMS voltage of the sine wave.

e) From the results of part d, calculate the conversion factor K using the following formula:

$$K = \frac{V_{peak}}{V_{RMS}} \qquad (2.2)$$

2) Repeat all of step 1 for a 2V peak, 1 kHz square wave.

3) Construct the circuit shown in figure 2.1. Using the analog oscilloscope, set the sensitivity knobs on channels one and two to the same setting. Select the display mode setting of X-Y. Connect VL to channel two and VS to channel one.

a) Sketch the display to scale.

b) From the straight line displayed on the oscilloscope, measure the slope m.

c) Calculate the theoretical value of m using the following formula: $m = \frac{V_L}{V_S}$

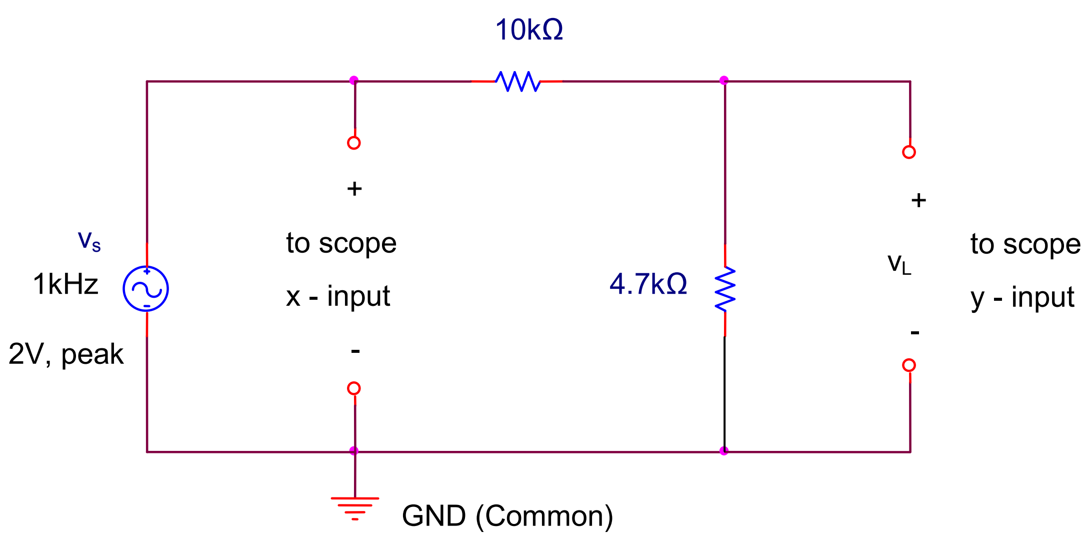


Figure 2.1

EXPERIMENT THREE
DC CIRCUITS

EQUIPMENT NEEDED:

1) DC Power Supply
2) DMM
3) Resistors
4) ELVIS

THEORY

Kirchhoff's Laws:

Kirchhoff's Voltage Law: The algebraic sum of the voltages around any closed path is zero.

$$\sum_{i=1}^{N} v_i = 0 \qquad (3.1)$$

Kirchhoff's Current Law: The algebraic sum of the currents at any node is zero.

$$\sum_{i=1}^{N} i_i = 0 \qquad (3.2)$$

Series Circuits:

In a series circuit the current is the same through all the elements.

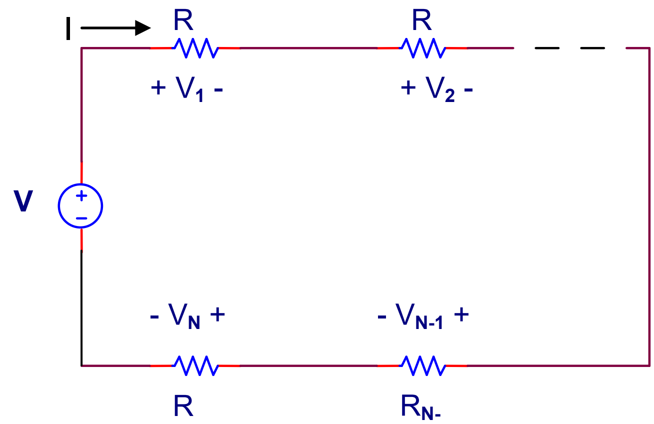


Figure 3. 1

The total series resistance R_S is given by

$$R_S = R_1 + R_2 + \cdots + R_{N-1} + R_N \qquad (3.3)$$

and

$$V_S = IR_S \qquad (3.4)$$

The Kirchhoff's voltage law indicates that:

$$V_S = V_1 + V_2 + \cdots + V_{N-1} + V_N \qquad (3.5)$$

The voltages across resistors can be obtained by multiplying the current by the corresponding resistors.

$$\left.\begin{aligned} V_1 &= IR_1 \\ V_2 &= IR_2 \\ &\vdots \\ V_{N-1} &= IR_{N-1} \\ V_N &= IR_N \end{aligned}\right\} \rightarrow \quad I = \frac{V_S}{R_S} \quad \rightarrow \begin{cases} V_1 = \left(\dfrac{R_1}{R_S}\right)V_S \\ V_2 = \left(\dfrac{R_2}{R_S}\right)V_S \\ \vdots \\ V_{N-1} = \left(\dfrac{R_{N-1}}{R_S}\right)V_S \\ V_N = \left(\dfrac{R_N}{R_S}\right)V_S \end{cases} \qquad (3.6)$$

The last expressions of equation 3.6 are known as voltage division.

Parallel Circuits:

In a parallel circuit the voltage is the same across all the elements.

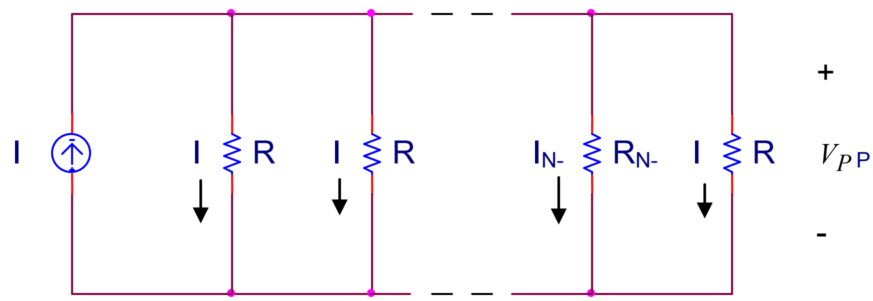


Figure 3. 2

The total parallel resistance, *Rp* is given by

$$\frac{1}{R_P} = \frac{1}{R_1} + \frac{1}{R_2} + \cdots + \frac{1}{R_{N-1}} + \frac{1}{R_N} \qquad (3.7)$$

and

$$V_P = I_P R_P \qquad (3.8)$$

Kirchhoff's current law states:

$$I_P = I_1 + I_2 + \cdots + I_{N-1} + I_N \qquad (3.9)$$

The current through the branch resistors can be obtained by dividing the terminal voltage V_P by the corresponding branch resistance, R. therefore:

$$\left.\begin{aligned} I_1 &= \frac{V_P}{R_1} \\ I_2 &= \frac{V_P}{R_2} \\ &\vdots \\ I_{N-1} &= \frac{V_P}{R_{N-1}} \\ I_N &= \frac{V_P}{R_N} \end{aligned}\right\} \rightarrow \; V_P = I_P R_P \; \rightarrow \left\{\begin{aligned} I_1 &= \left(\frac{R_P}{R_1}\right) I_P \\ I_2 &= \left(\frac{R_P}{R_2}\right) I_P \\ &\vdots \\ I_{N-1} &= \left(\frac{R_P}{R_{N-1}}\right) I_P \\ I_N &= \left(\frac{R_P}{R_N}\right) I_P \end{aligned}\right. \qquad (3.10)$$

The last expressions of equation 3.10 are known as current division.

The reciprocal of resistance is known as conductance. It is expressed in the following equations:

$$G = \frac{1}{R} \qquad (3.11)$$

and

$$G_P = \frac{1}{R_P} \qquad (3.12)$$

This expression can be used to simplify equations 3.12 as shown below.

$$\left.\begin{array}{l} I_1 = G_1 V_P \\ I_2 = G_2 V_P \\ \vdots \\ I_{N-1} = G_{N-1} V_P \\ I_N = G_N V_P \end{array}\right\} \rightarrow \quad V_P = \frac{I_P}{G_P} \quad \rightarrow \left\{\begin{array}{l} I_1 = \left(\frac{G_1}{G_P}\right) I_P \\ I_2 = \left(\frac{G_2}{G_P}\right) I_P \\ \vdots \\ I_{N-1} = \left(\frac{G_{N-1}}{G_P}\right) I_P \\ I_N = \left(\frac{G_N}{G_P}\right) I_P \end{array}\right. \qquad (3.13)$$

where

$$G_P = \frac{1}{R_1} + \frac{1}{R_2} + \cdots + \frac{1}{R_{N-1}} + \frac{1}{R_N} \qquad (3.14)$$

If only two resistors make up the network, as shown next

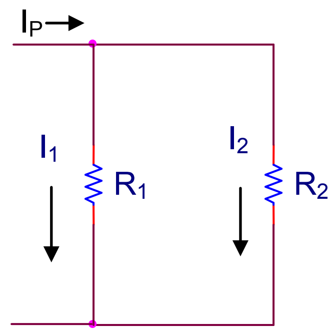


Figure 3.3

then the current in branches 1 and 2 can be calculated as follows:

$$I_1 = \left(\frac{G_1}{G_P}\right) I_P \qquad (3.15)$$

$$G_1 = \frac{1}{R_1} \qquad (3.16)$$

and

$$G_P = \frac{1}{R_1} + \frac{1}{R_2} \quad \rightarrow \quad G_P = \frac{R_1 + R_2}{R_1 R_2} \qquad (3.17)$$

$$\therefore \quad I_1 = \left(\frac{1}{R_1}\right)\left(\frac{R_1 R_2}{R_1 + R_2}\right) I_P \quad \rightarrow \quad I_1 = \left(\frac{R_2}{R_1 + R_2}\right) I_P \qquad (3.18)$$

In a similar fashion it can be shown that

$$I_2 = \left(\frac{R_1}{R_1 + R_2}\right) I_P \qquad (3.19)$$

(Note how the current in one branch depends on the resistance in the opposite branch)

But, if the network consists of more than two resistors - say four

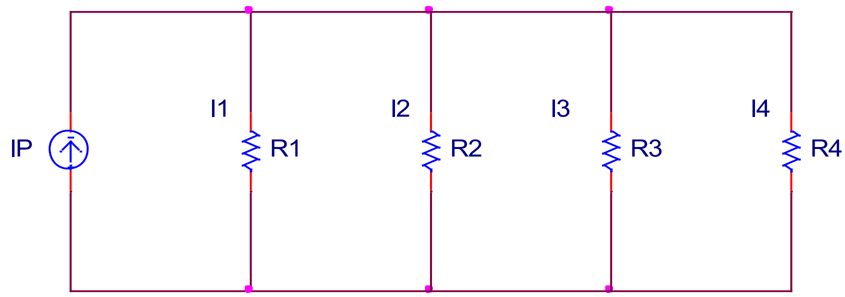


Figure 3.4

Then the calculation or branch currents using individual resistance becomes complex as demonstrated next, e.g.,

$$I_3 = \left(\frac{R_P}{R_3}\right) I_P \qquad (3.20)$$

$$\frac{1}{R_P} = \frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} + \frac{1}{R_4} \quad \to \quad \frac{1}{R_P} = \frac{R_2R_3R_4 + R_1R_3R_4 + R_1R_2R_4 + R_1R_2R_3}{R_1R_2R_3R_4} \qquad (3.21)$$

so that

$$I_3 = \left(\frac{1}{R_3}\right)\left(\frac{R_1R_2R_3R_4}{R_2R_3R_4 + R_1R_3R_4 + R_1R_2R_4 + R_1R_2R_3}\right) I_P \qquad (3.22)$$

and

$$I_3 = \left(\frac{R_1R_2R_4}{R_2R_3R_4 + R_1R_3R_4 + R_1R_2R_4 + R_1R_2R_3}\right) I_P \qquad (3.23)$$

By using conductances, the above is simplified to

$$I_3 = \left(\frac{\frac{1}{R_3}}{\frac{1}{R_1} + \frac{1}{R_2} + \frac{1}{R_3} + \frac{1}{R_4}} \right) I_P \qquad (3.24)$$

and is easily accomplished with a hand calculator.

As the above demonstrates, when using current division, always use conductances and avoid using resistances in the calculation for all parallel networks with more than two resistors.

Series - Parallel Circuits

The analysis of series -parallel circuits is based on what has already been discussed. The solution of a series-parallel circuit with one single source usually requires the computation of total resistance, application of Ohm's law, Kirchhoff's voltage law, Kirchhoff's current law, voltage and current divider rules.

Preliminary Calculations:

Be sure to show all necessary calculations.

I. The resistors used in this lab all have 5% tolerances. This is denoted by the gold band. Calculate the minimum and maximum values for resistances with nominal values of 1kΩ and 2.7kΩ. Enter the values in Table 3.1.

2. Assume that the two resistors of problem 1 are used in the circuit of Figure 3.5. Calculate v_1, v_2, and I when R_1 and R_2 take on their minimum and maximum values and enter in Table 3.2.

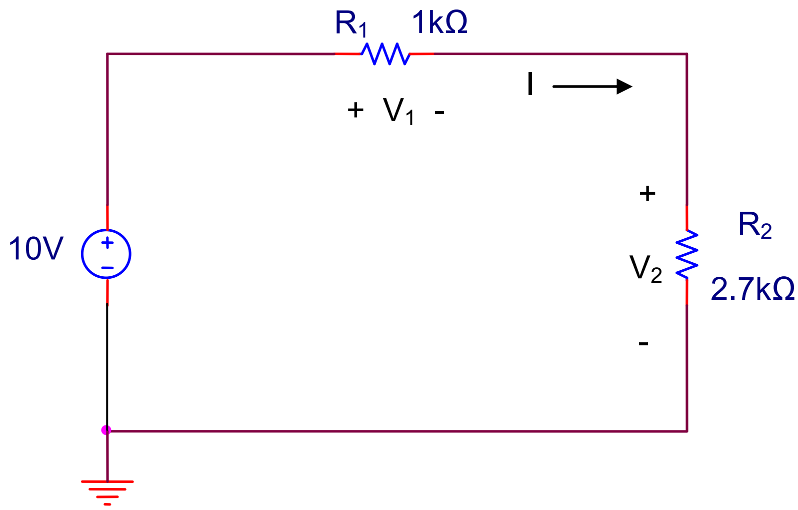


Figure 3.5

3. From your calculations in 2, record the maximum and the minimum possible values of I, v_1, and v_2 that you should see in the circuit in Table 3.3. Also, calculate and record the value of these variables when R_1 and R_2 are at the nominal values. What is the maximum % error in each of the variables possible due to the resistor tolerances?

4. For the circuit of Figure 3.6 calculate the resistance between nodes:

a. a and b ($R_{a\text{-}b}$)

b. a and c ($R_{a\text{-}c}$)

c. c and d ($R_{c\text{-}d}$)

Enter your results in Table 3.4

Hint: Part c cannot immediately be reduced using series and parallel combinations.

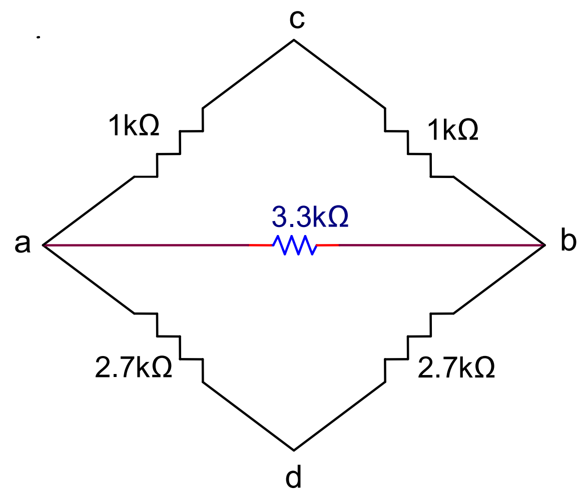


Figure 3.6

5. Use voltage division to calculate V_1 and V_2 for the circuit in Figure 3.7. Enter your results in Table 3.5.

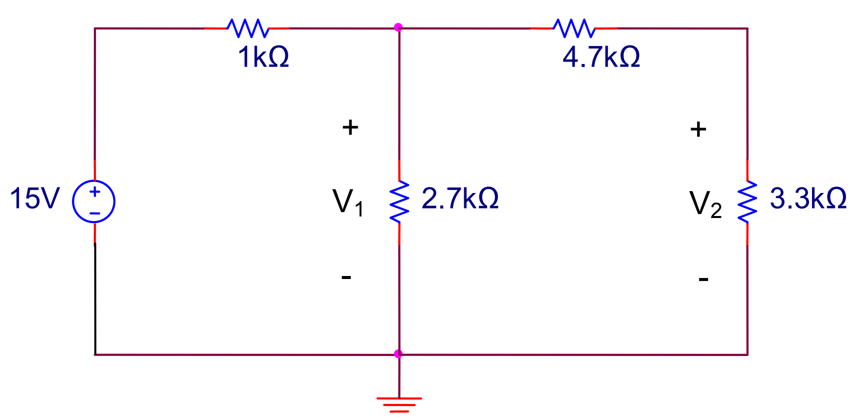


Figure 3.7

6. For the circuit in Figure 3.8, if R = 1k ohm, calculate I. Use current division to calculate I_R. Enter your results in Table 3.6. Repeat for R = 2.7k and 3.3k ohms.

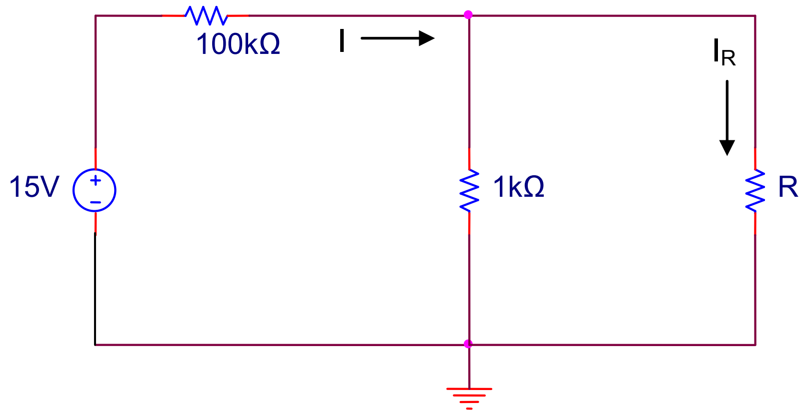


Figure 3.8

7. For the circuit in Figure 3.9, calculate each of the variables listed in Table 3.7.

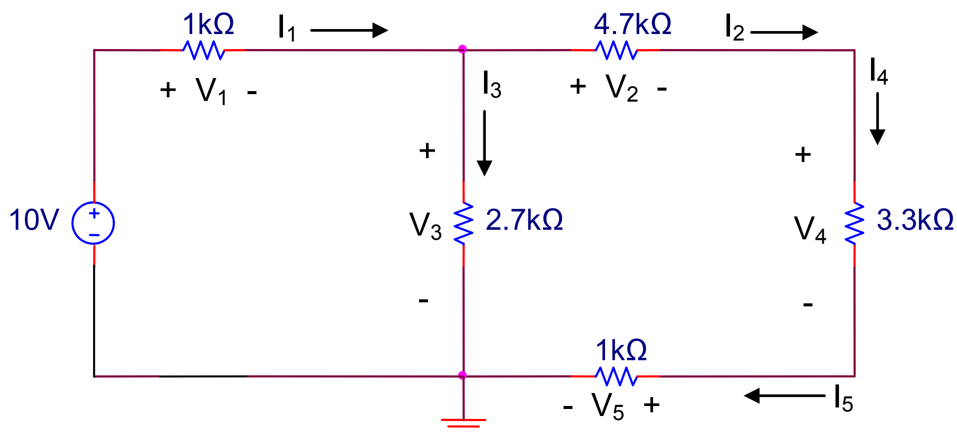


Figure 3.9

Procedure

l. Place a wire between the two measuring terminals of the ohmmeter and adjust the measurement reading to zero ohms. Obtain a 1kΩ and 2.7kΩ resistor and measure their values with the ohmmeter. What is the % error as compared to their nominal values? Enter your results in Table 3.1.

2. Construct the circuit in Figure 3.5. Measure V_1 and V_2 using the DMM only. Calculate I from your measurements. What is the % error as compared to their nominal values? Enter your results in Table 3.3.

3. Construct the circuit of Figure 3.6. Use an ohmmeter to measure the resistances listed in Table 3.4. Calculate the % error.

4. Construct the circuit of Figure 3.7. Measure V_1 and V_2 using the DMM only. Calculate the % error. Enter your results in Table 3.5.

5. Construct the circuit of Figure 3.8. Find I and I_R for R = 1kΩ, 2.7 kΩ, and 3.3 kΩ by measuring the appropriate voltages using the DMM only and applying Ohm's Law. Enter your results in Table 3.6. Note that I is approximately constant. Why?

6. Construct the circuit of Figure 3.9. Using the DMM, measure each of the variables listed in Table 3.7, and calculate the % error for each. Verify that KVL holds for each of the 3 loops in the circuit. Verify that KCL holds at each node. What can be said about $I_2 + I_3$ and I_1?

Table 3.1

Rnominal	Rmin	Rmax	Rmeas	% error
1kΩ				
2.7kΩ				

Table 3.2

	R_1,min R_2, min	R_1, max R_2, min	R_1, min R_2, max	R_1, max R_2,max
I				
V_1				
V_2				

Table 3.3

	max	min	nom	max % error	meas	% error
V_1						
V_2						
I						

Table 3.4

Resistance	Calculated	Measured	% error
R_{ab}			
R_{ac}			
R_{cd}			

Table 3.5

	Calculated	Measured	% error
V_1			
V_2			

Table 3.6

R	I, calc	I_R, calc	I, meas	I_R, meas
1kΩ				
2.7kΩ				
3.3kΩ				

Table 3.7

PARAMETER	CALCULATED	MEASURED	% ERR
V1			
V2			
V3			
V4			
V5			
I1			
I2			
I3			
I4			
I5			

EXPERIMENT FOUR
COMPUTER SIMULATION

EQUIPMENT NEEDED:

1) PSPICE Computer Program
2) ELECTRONICS WORKBENCH (optional)

THEORY & PRELIMINARY PREPARATION

Review set procedures for PSPICE or ELECTRONICS WORKBENCH with your instructor.

Procedure:

DC Analysis. Analyze the circuit in Figure 4.1 using PSPICE. In addition to the node voltages, have the program find the currents i_1 and i_2.

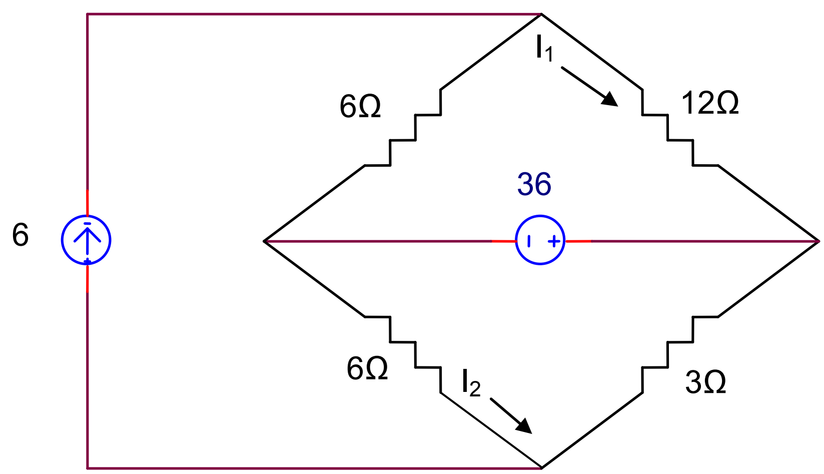


Figure 4.1

Transient Analysis. Analyze the circuit in Figure 4.2 using PSPICE

a. Assume $v_i(t)$ is a 1KHz square wave (15 volts, peak) as shown. Write a PSPICE program to plot the first 2 cycles of v_o (t).

b. Repeat a, but now plot the 10th and 11th cycles of $v_o(t)$.

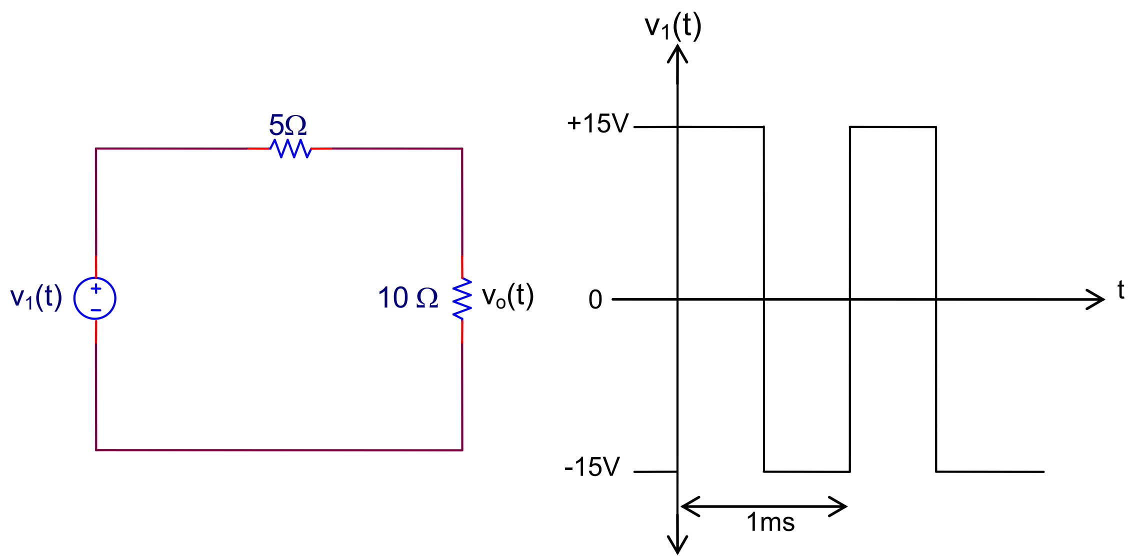


Figure 4.2

Repeat step 2a when vi(t) is a sine wave of peak voltage 15V and a frequency of 1kHz.

EXPERIMENT FIVE
NETWORK THEOREMS

EQUIPMENT NEEDED:

1) DC power supply
2) DMM
3) Resistors
4) ELVIS

THEORY

Thevenin's Theorem:

Any two-terminal linear resistive circuit can be replaced by an equivalent circuit with a voltage source and a series resistor.

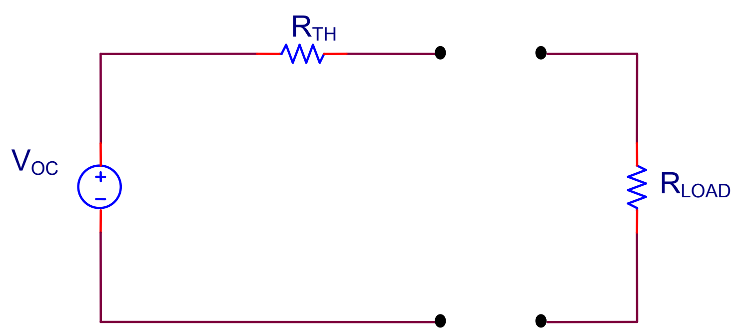


Figure 5.1 – Thevenin Circuit

The voltage source is denoted by V_{OC} (open circuit voltage), and the resistor by R_{Th}(Thevenin resistor). The objective is to evaluate V_{OC} and R_{Th} . The procedure of obtaining V_{OC} and R_{Th} is stated below for the circuits containing independent sources only.

a) Remove the portion of the circuit external to which the Thevenin's equivalent circuit is to be found.

b) Compute the voltage across the open-loop terminals. This voltage is V_{OC}.

c) Eliminate all the sources and compute the resistance across the open-loop terminal. This resistance is R_{Th}. A voltage source is eliminated by replacing it with a short circuit and a current source is eliminated by replacing it with an open circuit.

d) Draw the Thevenin equivalent circuit by placing the load resistor across the open-loop terminal (Figure 5.1).

Norton's Theorem:

Any two-terminal linear resistive circuit can be replaced by an equivalent circuit with a current source and a parallel resistor.

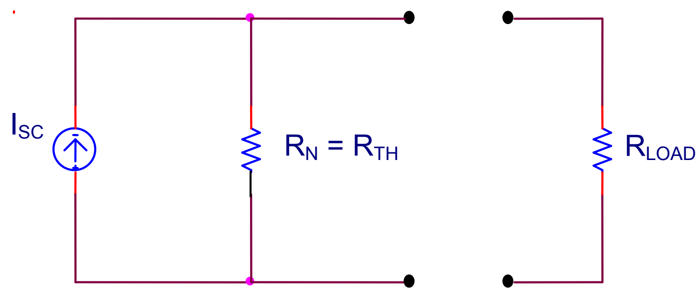


Figure 5.2 – Norton Circuit

The current source is denoted by I_{SC} (short circuit current), and the resistor by R_N (Norton resistor). The value of R_N is the same as R_{Th}. The objective is to evaluate I_{SC} and R_N. The simplest way is to convert the voltage source of Figure 5.1 to the current source of Figure 5.2. This indicates:

$$I_{SC} = V_{OC}/R_{TH} \qquad R_N = R_{TH} \qquad (5.1)$$

The general procedure of obtaining I_{SC} and R_N is stated below for circuits containing independent sources only.

a) Remove the portion of the circuit external to which the Norton equivalent circuit is to be found.

b) Place a short across the open-loop terminals and evaluate the current through this portion of the circuit. This current is I_{SC}.

c) Eliminate all sources and compute the resistance across the open-loop terminals. This resistance is R_N, which is identical to R_{Th}.

d) Draw the Norton equivalent circuit by placing the load across the open-loop terminals (Figure 5.2).

The procedure of obtaining R_{Th} (R_N) for the circuits containing dependent sources is different than outlined above. In such cases R_{Th} is obtained by

$$R_{TH} = V_{OC}/I_{SC} \qquad R_{TH} = R_N \qquad (5.2)$$

Maximum Power Theorem

Figure 5.3 shows a Thevenin circuit with a load resistor R_L. The maximum power theorem states that for the load resistor R_L to dissipate the maximum power, its value must be equal to the Thevenin resistance, that is:

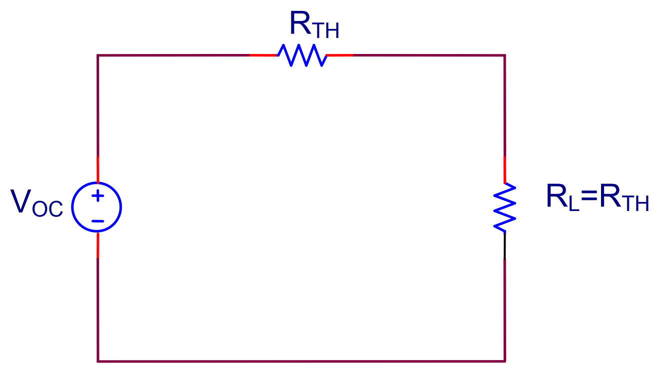


Figure 5.3

In such a case the power dissipated by R_L is maximum and is given by the following equation

$$P_{\max(load)} = \frac{V_{OC}^2}{4R_{TH}} \qquad (5.3)$$

Superposition Theorem

In any linear resistive circuit containing two or more independent sources, the current through or voltage across any element is equal to the algebraic sum of the currents or voltages produced independently by each source.

The procedure of using the superposition theorem is stated below:

a) Eliminate all independent sources except one.

b) Obtain the currents and/or voltages desired. Record the proper directions of currents and polarities of voltages.

c) Repeat steps a) and b) for all the other independent sources in the circuit.

d) Combine the results algebraically.

Preliminary Calculations:

1. Find the Thevenin and Norton Equivalents for the circuit to the left of a-b in figure 5.4.

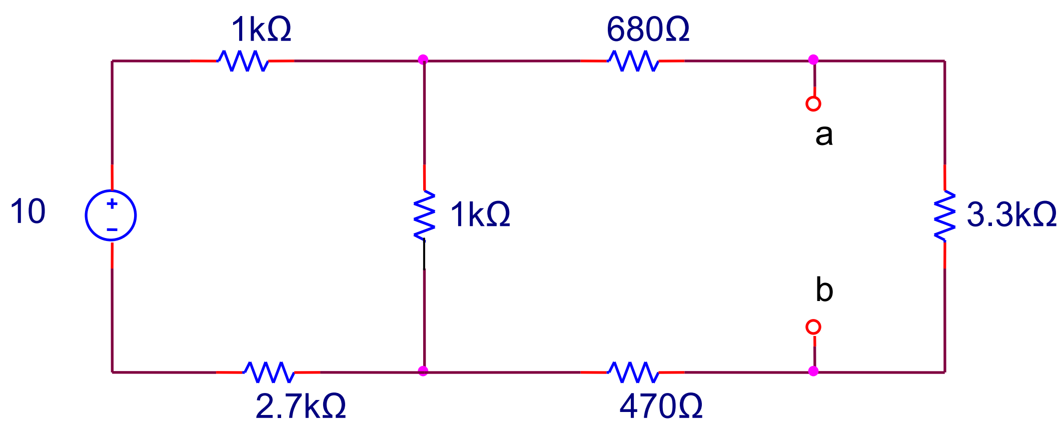


Figure 5.4

2. Using the maximum power transfer theorem, find the value of R which will result in the maximum power being delivered to R (see Figure 5.5).

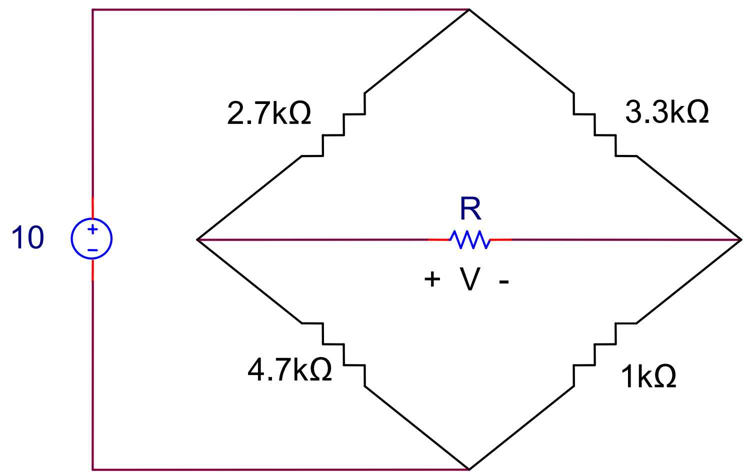


Figure 5.5

3. For the circuit in Figure 5.6, use superposition to determine V_1 and V_2.

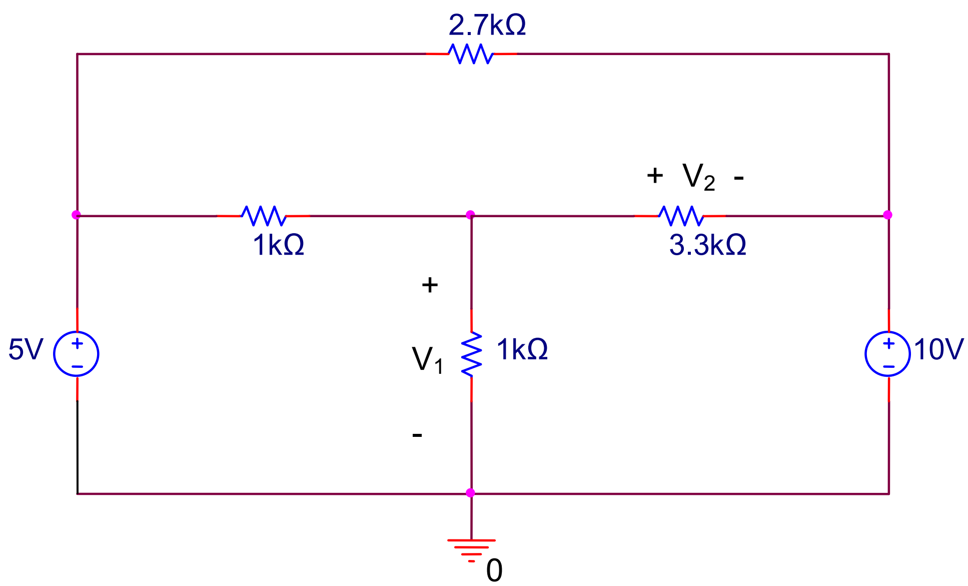


Figure 5.6

Procedure

1. Construct the circuit of Figure 5.4. Measure the voltage, V_{ab}.
2. Remove the 3.3kΩ resistor and measure V_{oc}. Compare your measurement to the value calculated in the prelab.

3. Place a short circuit between the terminals a-b and measure the current, I_{sc}, flowing through this short circuit. Compare to the value calculated in the pre-lab.

4. Remove the 10 volt source and replace it with a short circuit. Use an ohmmeter to measure R_{th} to the left of a-b. Compare to the value calculated in the pre-lab.

5. Using the measured values of V_{oc} and R_{th} (using closest available standard values) construct the Thevenin equivalent of the circuit to the left of a-b. Add the 3.3kΩ resistor across a-b and measure the voltage across it. Compare to the value measured in step 1.

6. Construct the circuit in Figure 5.5. Using values of R in the range from 100Ω to 10kΩ, measure V and calculate P, the power delivered to R. Plot P vs. R. Be sure to take several readings for R values in the neighborhood of the value calculated in part 2 of the pre-lab.

7. Construct the circuit of Figure 5.6 and measure V_1 and V_2.

8. Remove the 5 volt source and replace it with a short circuit. Measure V_1 and V_2.

9. Replace the 5 volt source and remove the 10 volt source and replace it with a short circuit. Measure V_1 and V_2 again.

10. Use your results from steps 7-9 to verify the principle of superposition. Also, compare these values to those obtained in step 3 of the pre-lab.

11. Use SPICE to calculate V_{ab} for the circuit in Figure 5.4. Compare your results to your measurements in step 1 of the procedure.

l2. Use SPICE to find V_{oc} and I_{sc} for the circuit in Figure 5.4. (Note: you will need to modify the circuit to do this.)

l3. Using the value of R calculated in step 2 of the pre-lab, use SPICE to calculate V in Figure 5.5. Compare this value to your measurement.

14. Use SPICE to analyze the circuit in Figure 5.6 (with both sources in). Compare your results to those obtained in the pre-lab.

EXPERIMENT SIX
OPERATIONAL AMPLIFIERS

EQUIPMENT NEEDED:

1) Oscilloscope
2) DC power supply
3) Function Generator
4) DMM
5) Resistors
6) ELVIS

THEORY

Operational Amplifier

The symbol for an operational amplifier is shown in Figure 6.1. The operational amplifier (op amp) has many terminals. These terminals include inverting input, noninverting input, output, dc power supply with positive, negative and ground, frequency compensation, and offset null terminals. For the sake of simplicity Figure 6.1 shows the two input terminals and the output terminal

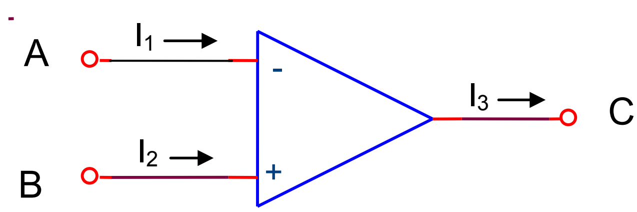


Figure 6.1

Operational amplifiers are usually available in the form of integrated circuits. The most important characteristics of op amps are stated below:

d) Currents into both input terminals are zero

$$I_1 = 0 \qquad I_2 = 0 \tag{6.1}$$

e) Voltage between the input terminals is zero

$$V_{AB} = 0 \tag{6.2}$$

f) The output current $I_3 \neq 0$, hence KCL does not apply here, that is

$$I_1 + I_2 \neq I_3 \tag{6.3}$$

For more information on the op amps refer to the sections on op amps in your textbook.

Preliminary Calculations:

For the circuit in Figure 6.2, find v_o as a function of v_i and R_F.

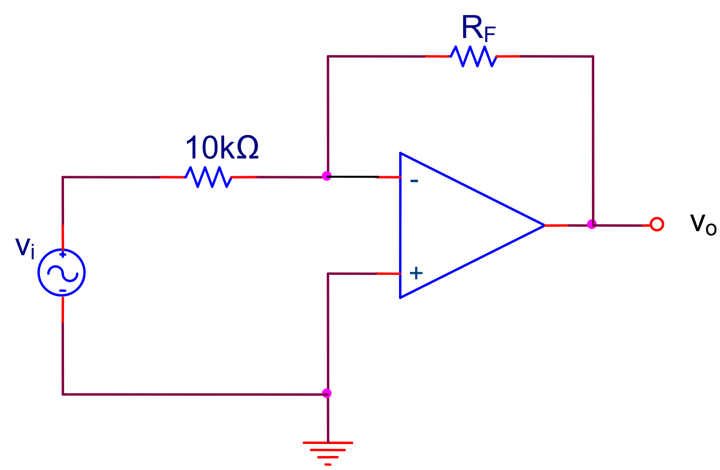


Figure 6.2

2. Repeat 1 for the circuit of Figure 6.3.

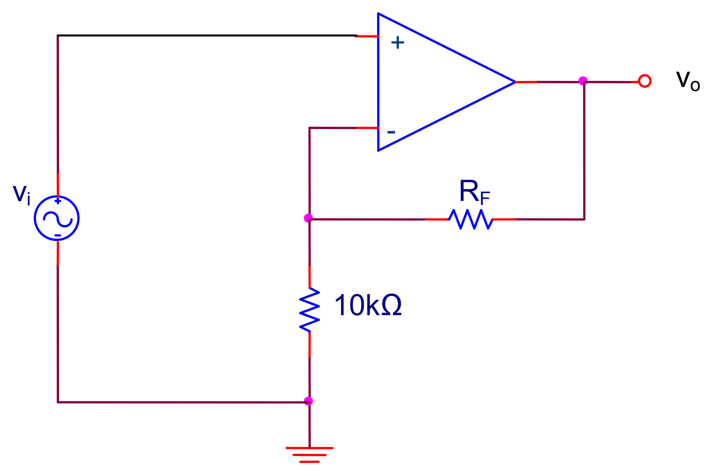


Figure 6.3

3. Calculate the voltage V_{ab} across the 6.2k ohm resistor in Figure 6.4.

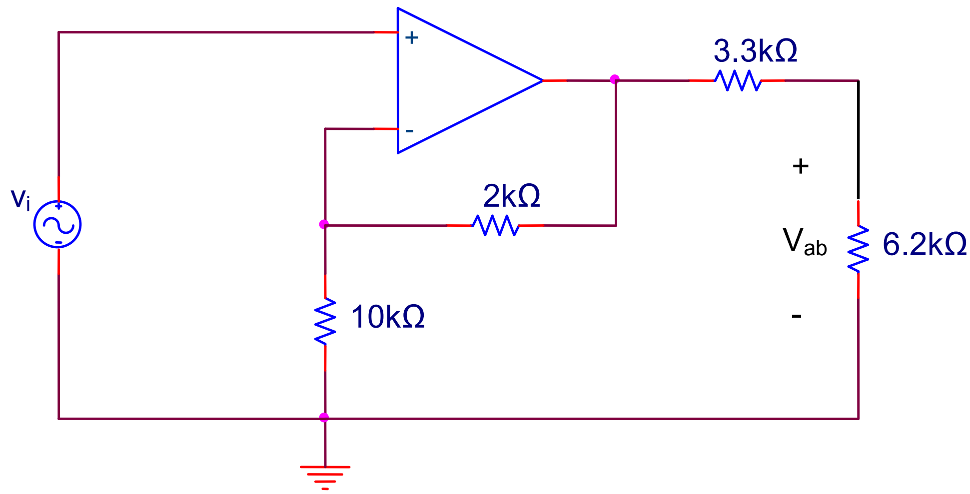


Figure 6.4

Procedure:

Throughout this experiment, connect the OP AMP to the d.c. biasing circuit shown below. Apply 30V from the VPOS to VNEG terminals of the supply or 15V from VPOS to ground and -15V from VNEG to ground. These voltages are required for proper operation of the device. The reasons for this will be explained in an electronic course. Pins 2 & 3 are the inverting and non-inverting inputs of the OP AMP and Pin 6 is the output.

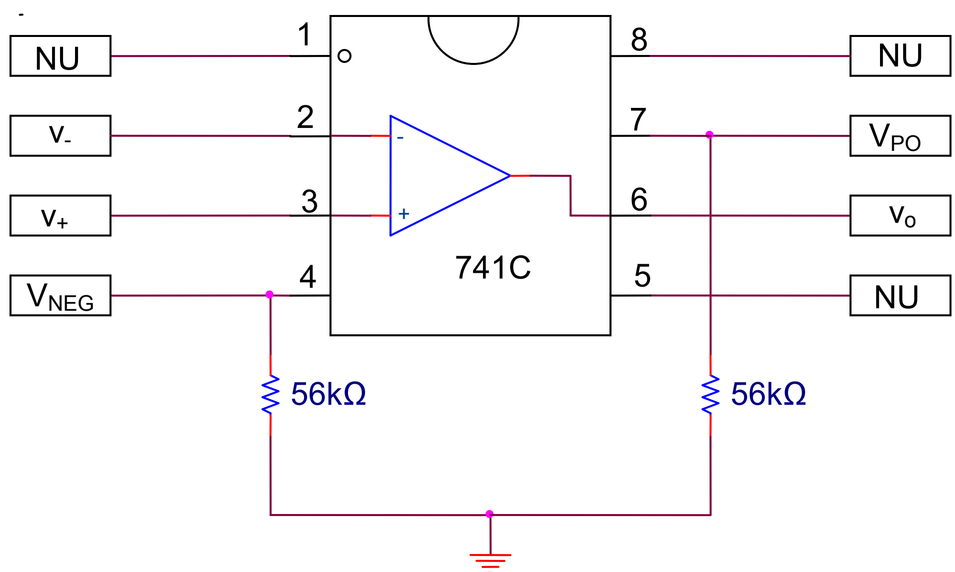


Figure 6.5 – Top View of 741C Operational Amplifier with biasing circuit

Table 6.1 – Pin out of 741C Operational Amplifier

PIN NO.	NAME	DESCRIPTION
1	NU	NOT USED
2	v_-	INVERTING INPUT
3	v_+	NON-INVERTING INPUT
4	V_{NEG}	NEG. DC PS VOLTAGE
5	NU	NOT USED
6	v_o	OUTPUT VOLTAGE
7	V_{POS}	POS. DC PS VOLTAGE
8	NU	NOT USED

1. Construct the circuit in Figure 6.2. Set R_F = 10k ohms, and apply a 0.3V peak, 1kHz sine wave at V_i. Use your oscilloscope to observe V_i and V_o simultaneously. Record the peak value of V_o. Compare it to the value calculated in the pre-lab. What is the phase difference between V_o and V_i?

2. Repeat 1 for R_F = 27k ohms and 100k ohms.

3. Repeat parts 1 and 2 for the circuit of Figure 6.3.

4. Replace R_F in Figure 6.3 with a short circuit. What is the peak value of V_o?

5. Construct the circuit of Figure 6.4. Measure the voltage V_{ab} with an oscilloscope. Compare your result to that obtained in the pre-lab.

EXPERIMENT SEVEN
FIRST ORDER CIRCUITS

EQUIPMENT NEEDED:

1) Oscilloscope
2) Function Generator
3) Resistors, Capacitors, Inductors
4) ELVIS

THEORY

Voltage-Current Relationship for a Capacitor:

The voltage across a capacitor and the current through the capacitor are related as follows:

$$v_C(t) = \frac{1}{C}\int_{-\infty}^{t} i_C(t')dt' + v_C(-\infty) \qquad (7.1)$$

$$i_C(t) = C\frac{dv_C(t)}{dt} \qquad (7.2)$$

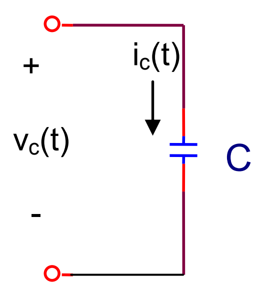


Figure 7.1

Equation (7.1) can be written as (if $v_C(-\infty)$=0)

$$v_C(t) = \frac{1}{C}\int_{-\infty}^{0} i_C(t')dt' \quad + \quad \frac{1}{C}\int_{0}^{t} i_C(t')dt' \qquad (7.3)$$

$$with \quad V_0 = \frac{1}{C}\int_{-\infty}^{0} i_C(t')dt' \quad \rightarrow \quad v_C(t) = V_0 + \frac{1}{C}\int_{0}^{t} i_C(t')dt' \qquad (7.4)$$

If the capacitor has no initial voltage ($V_O = 0$), then equation (7.4) reduces to:

$$v_C(t) = \frac{1}{C}\int_0^t i_C(t')dt' \qquad (7.5)$$

It is also clear that, when the voltage v(t) across the capacitor is constant, the current through the capacitor is zero. Under such a condition the capacitor can be replaced by an open circuit. This occurs for the case of DC input, steady state condition.

Voltage-Current Relationship for an Inductor:

The voltage across an inductor and the current through the inductor are related as follows:

$$v_L(t) = L\frac{di_L(t)}{dt} \qquad (7.6)$$

$$i_L(t) = \frac{1}{L}\int_{-\infty}^t v_L(t')dt' \qquad (7.7)$$

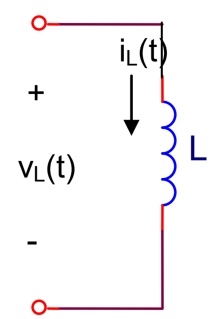


Figure 7.2

Equation (7.7) can be written as

$$i_L(t) = \frac{1}{L}\int_{-\infty}^{0} v_L(t')dt' \quad + \quad \frac{1}{L}\int_{0}^{t} v_L(t')dt' \qquad (7.8)$$

$$with \quad I_0 = \frac{1}{L}\int_{-\infty}^{0} v_L(t')dt' \quad \rightarrow \quad i_L(t) = I_0 + \frac{1}{L}\int_{0}^{t} v_L(t')dt' \qquad (7.9)$$

If the inductor has no initial current (I_0 = 0), then equation (7.9) reduces to:

$$i_L(t) = \frac{1}{L}\int_{0}^{t} v_L(t')dt' \qquad (7.10)$$

It is also clear that, when the current $i_L(t)$ through the inductor is constant, the voltage across the inductor is zero. Under such a condition the inductor can be replaced by a short circuit. This occurs for the case of DC input, steady state condition.

Simple RC and RL Circuits:

The differential equation of a simple RC and RL circuits (first order differential equation) can be obtained by application of KVL or KCL and using equations (7.1) and (7.2) for the capacitor and equations (7.6) and (7.7) for the inductor.

For the case of a simple RC circuit the time constant τ in seconds is defined as;

$$\tau = RC \qquad (7.11)$$

For the case of a simple RL circuit the time constant τ in seconds is defined as

$$\tau = L/R \qquad (7.12)$$

The terms τ = RC and τ = L/R appear in the solutions of simple RC and RL circuits respectively. It is important to remember that for the case of DC input, it takes approximately 5τ (5RC) for the capacitor to become fully charged, and 5τ (5L/R) for the inductor to become fully fluxed or energized, assuming initially there is no voltage across the capacitor and no current flowing through the inductor.

For the DC input, steady state solution, the capacitor is replaced by an open in the RC circuit and inductor is replaced by a short in the RL circuit.
For further information, review the sections on RC and RL circuits in your textbook.

Preliminary Calculations:

1.A voltage source can be modeled as an ideal voltage source in series with a resistance, R_{int} as shown in figure 7.3.

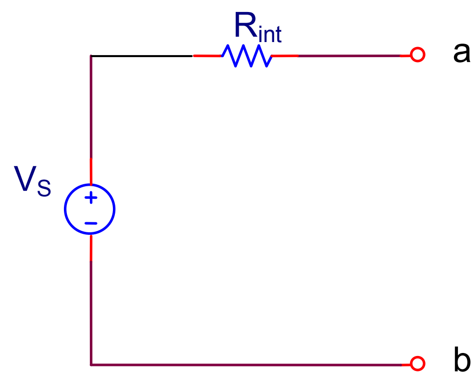


Figure 7.3

The internal resistance of the voltage source (R_{int}), can be determined by placing a known resistance of R across the open terminals of the circuit in figure 7.3. In doing so, the following procedure allows the student to measure this internal resistance and use it in circuits that are energized by voltage sources.

If:
$V_{oc} = V_{ab}$, when a-b is open-circuited, and
$V_R = V_{ab}$, when a resistance, R, is connected between a and b
Show that

$$R_{int} = \frac{R(V_{OC} - V_R)}{V_R} \qquad (7.13)$$

2. At $t = 0$, the switch in Figure 7.4 is moved from position 1 to position 2. Find and sketch, $v_C(t)$, $t \geq 0$.

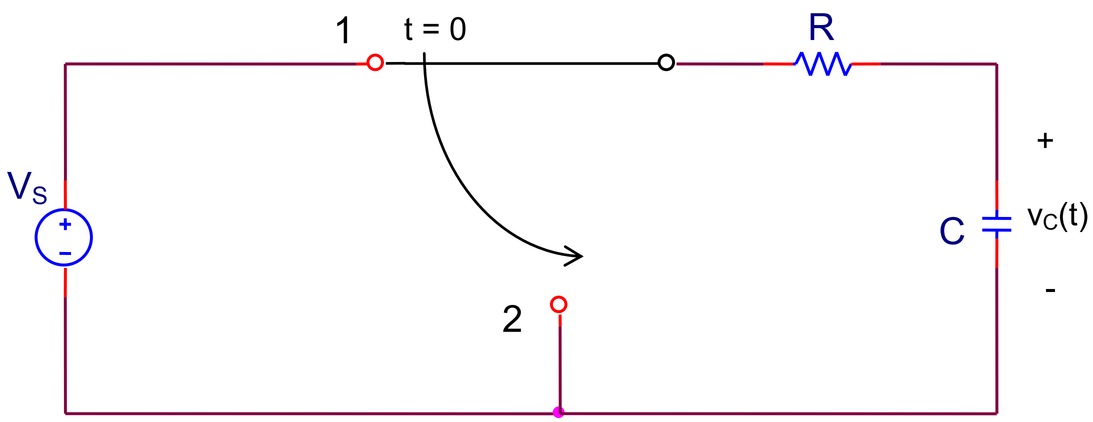


Figure 7.4

3. An "actual" inductor can be modeled as a resistor in series with an ideal inductor. If the circuit in Figure 7.5 is constructed with a one volt d.c. source, show that:

$$R_{dc} = \left(\frac{R}{1 - V_L}\right) V_L \qquad (7.14)$$

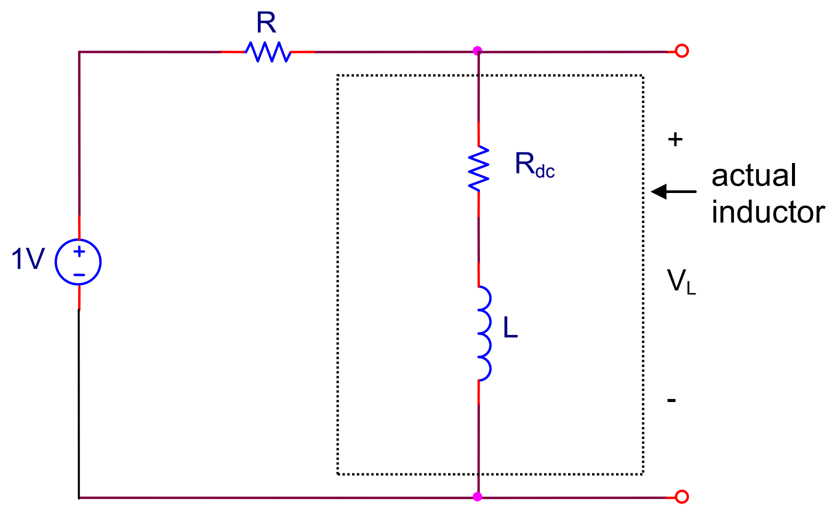


Figure 7.5

4. Assume the actual inductor in Figure 7.6 has a dc resistance, R_{dc} and an inductance, L. Find and sketch, $V_R(t)$, the voltage across the resistor.

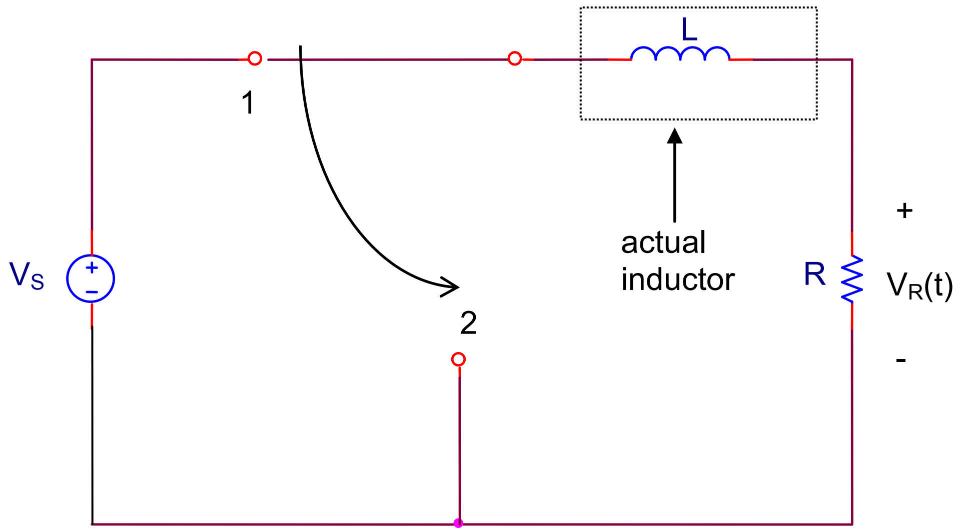


Figure 7.6

Procedure:

1. Use the technique described in part 1 of the pre-lab to determine R_{int} for your square wave generator, when the generator is set to 2kHz. Use values of R that give V_R readings in the range of 1/3 V_{OC} - 2/3 V_{OC}. Be sure to include this as a part of your total resistance in the remainder of this and other labs. You should recheck the value if you vary the frequency later in the experiment.

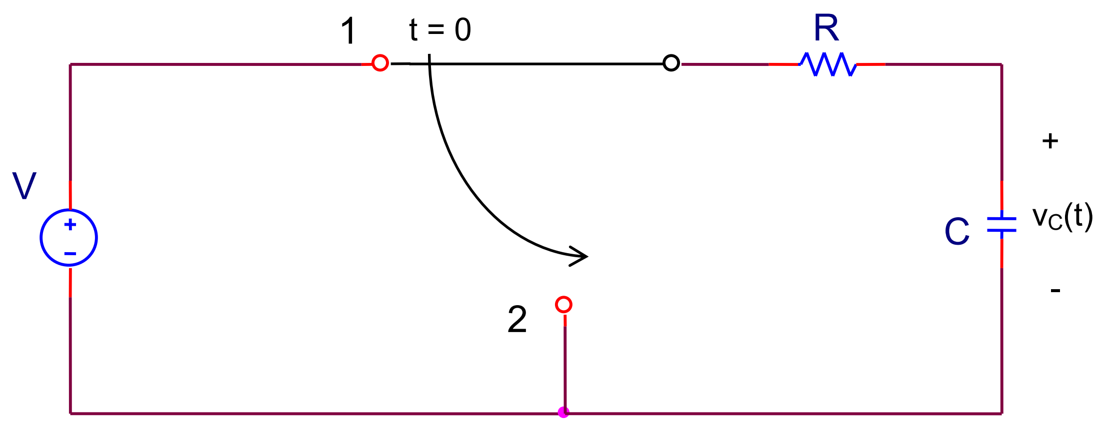


Figure 7.7

The natural response of RC, RL and RLC circuits can be demonstrated with the use of a square wave function generator. Consider Figure 7.7. The switch is left in position 1 for a "long time" so that the capacitor charges to V volts. When the switch is moved to position 2 (at t = 0) the capacitor discharges and we have the natural response of the RC circuit. The circuit in Figure 7.8 provides the same result repeatedly for the ease of viewing on the oscilloscope as long as T/2 is long compared to the circuit time constant, RC.

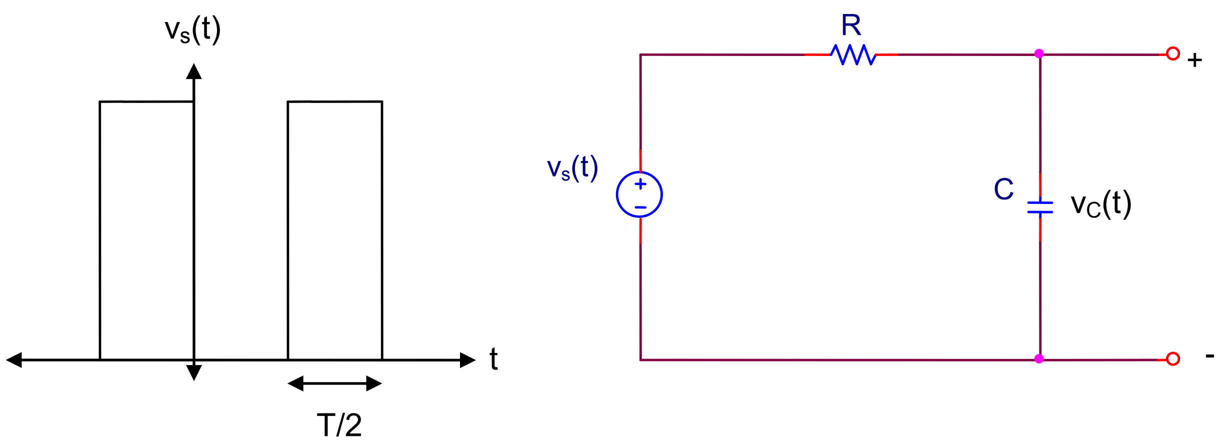


Figure 7.8

2. Construct the circuit in Figure 7.8. Use R = 470Ω and C = 0.1μF. Set the function generator for a 1 volt peak-to-peak square wave of 2 KHz. Use the oscilloscope to observe, v_C (t). Sketch the portion of the waveform in which the capacitor is discharging. Use your sketch to determine the time constant of the circuit. Compare this to the theoretical time constant (be sure to include the internal resistance of the source).

3. Repeat part 2 for R = 6.8kΩ and C = 0.1μF. Adjust the frequency of the function generator to obtain an adequate display.

4. Using the procedure described in part 3 of the pre-lab, find R_{dc} for a 68mH inductor.

5. Construct the circuit of Figure 7.9. Use R = 680Ω and L = 68mH. Set the amplitude of $v_s(t)$ to a 1 volt peak-to-peak square wave. Adjust the frequency of the function generator for an adequate display of the resistor voltage as it decreases. Sketch the voltage and determine the time constant of your circuit from the sketch. Compare this value to the calculated time constant. Be sure to include the effects of R_{int} and R_{dc} into your calculations.

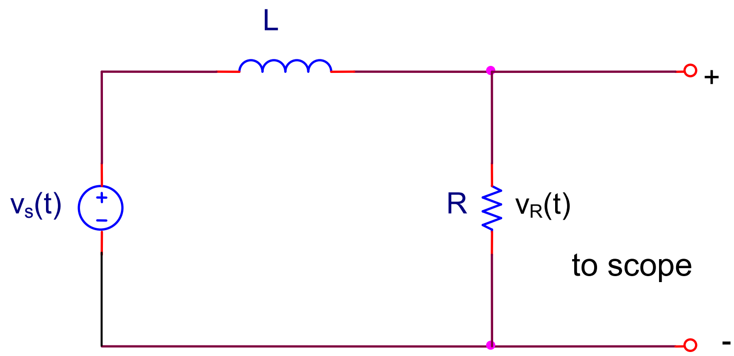


Figure 7.9

6. Using the above procedure, find (experimentally) the dc resistance and the inductance of the unknown inductor supplied by your lab instructor.

7. Use PSPICE[1] (TRANSIENT ANALYSIS) to plot $v_C(t)$, $t > 0$, for the circuit in Figure 7.7. Compare your output to the sketch obtained in the lab. Do this for C = 0.1μF and R = 470Ω and 6.8kΩ ohms.

8. Use PSPICE[1] to obtain the resistor voltages that you obtained in parts 5 and 6 of the procedure. Compare them to your experimental results also.

[1]NOTE: It is not necessary to use the pulse input here. You can analyze the circuit (for $t \geq 0$) as it is shown in Figures 7.6 & 7.7.

EXPERIMENT EIGHT
SECOND ORDER CIRCUITS

EQUIPMENT NEEDED:

1) Oscilloscope
2) Function Generator
3) Resistors, Capacitors, Inductors
4) ELVIS

THEORY

The Differential Equation - 2nd Order

The differential equations of second order circuits are obtained by application of KVL and/or KCL and using equations (7.1), (7.2), (7.6) and (7.7) that were discussed in the Theory of Experiment 7. If the application of KVL or KCL results in an integro-differential equation, the derivative of both sides of the equation should be taken to convert the equation into a differential equation.

Consider a general second order differential equation

$$a\frac{d^2y}{dt^2} + b\frac{dy}{dt} + cy = 0 \qquad (8.1)$$

$$\text{with roots} \qquad r_{1,2} = -\frac{b}{2a} \pm \frac{\sqrt{b^2 - 4ac}}{2a} \qquad (8.2)$$

Assume a, b, and c are positive. Three cases are considered here.

Case I $b^2 - 4ac > 0 \rightarrow$ "Overdamped" condition.

In this case, r_1 and r_2 are real, distinct and negative. This results in solutions of (8.1) in the form of decaying exponentials: $y = A_1e^{r_1t} + A_2e^{r_2t}$ Coefficients A_1 and A_2 are determined using the initial conditions of the system or circuit.

Case II $b^2 - 4ac = 0 \rightarrow$ "Critically Damped" condition.

In this case, r_1 and r_2 are real, equal and negative ($r = r_1 = r_2$). This results in solutions of (8.1) in the form of following equation: $y = A_1e^{rt} + A_2te^{rt}$ or $y = e^{rt}(A_1 + A_2t)$

Case III $b^2 - 4ac < 0 \quad \rightarrow$ "Underdamped" condition.

In this case, r_1 and r_2 are complex. This results in solutions of (8.1) in the form of the following equation: $y = e^{\alpha t}\left(A_1 \cos \omega_d t + A_2 \sin \omega_d t\right)$ where:

$$\alpha = -\frac{b}{2a} \qquad (8.3)$$

$$\omega_d = \frac{\sqrt{4ac - b^2}}{2a} \qquad (8.4)$$

The term α is known as the damping factor and ω_d is the damped frequency.

Consider the graph of an underdamped case shown in figure 8.1 where:

$$v(t) = Ae^{\alpha t} \cos(\omega_d t + \theta) \qquad (8.5)$$

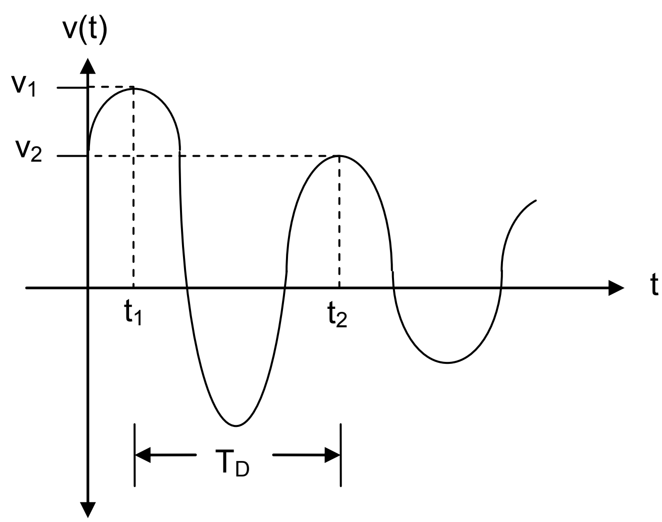


Figure 8.1

Using figure 8.1, the following items can be determined:

$$v_1 = Ae^{\alpha t_1}\cos(\omega_d t_1 + \theta) \qquad (8.6)$$

$$v_2 = Ae^{\alpha t_2}\cos(\omega_d t_2 + \theta) \qquad (8.7)$$

Dividing (8.6) by (8.7) yields:

$$\frac{v_1}{v_2} = e^{\alpha(t_2 - t_1)} \qquad (8.8)$$

where $t_2 - t_1 = T_D \quad \rightarrow \quad T_D = \frac{2\pi}{\omega_d}$ $\qquad (8.9)$

Substituting (8.9) into (8.8) and taking the natural logarithm of both sides results in:

$$\ln\left(\frac{v_1}{v_2}\right) = \frac{2\pi\alpha}{\omega_d} \qquad (8.10)$$

$$\alpha = \frac{\omega_d}{2\pi}\ln\left(\frac{v_1}{v_2}\right) \qquad (8.11)$$

Preliminary Calculations:

1. For the circuit in Figure 8.2 , write a differential equation for v(t) (in terms of R, L, and C). If L = 68mH and C = 0.01μF, what range of values for R correspond to over-, under-, and critically damped cases?

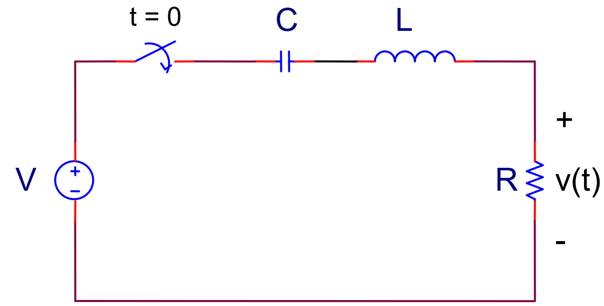


Figure 8.2

2. Repeat 1 for the circuit in Figure 8.3 (same L and C).

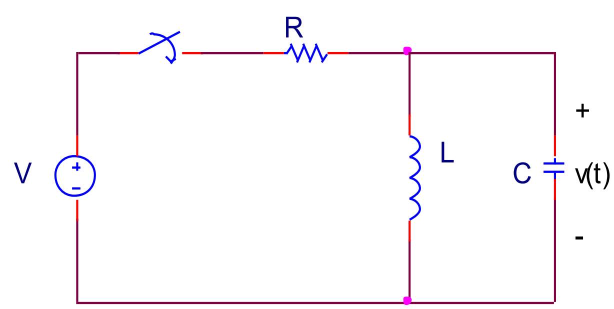


Figure 8.3

Procedure:

1. The circuit of Figure 8.4 can be used to observe the voltage, v(t) for the circuit of Figure 8.2. As in the previous experiment, the square wave generator is used in place of the dc source in series with a switch.

a. Use the oscilloscope to observe and sketch v(t) (while the capacitor is charging) for R = 6.8kΩ. Vary the frequency of the square wave so that you observe v(t) until it reaches its steady state value.

b. Repeat a. for R = 560Ω.

c. For each case, state whether the circuit is over-, under-, or critically damped. For the underdamped case (s) calculate the theoretical damping factor, α, and the damped frequency, ω_d. Compare these to your experimental values. (Don't forget to include the internal resistance of the square wave generator.)

d. Using PSPICE plot v(t) versus t for each of the three circuits, compare them to your experimental results. Recall that the circuit you are analyzing is equivalent to that shown in Figure 8.2 (zero initial conditions). You do not need to use the pulse input for PSPICE.

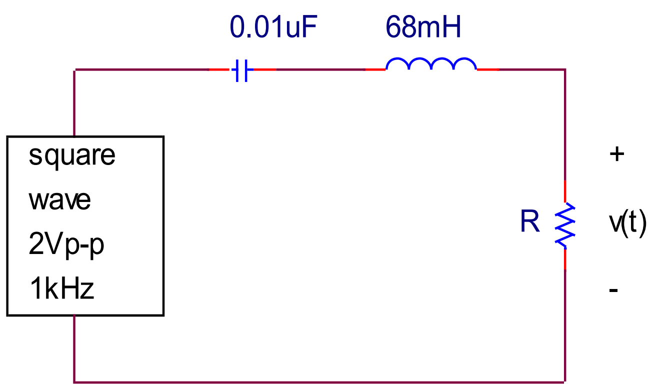


Figure 8.4

2. Repeat step 1 for the circuit in Figure 8.5 (used to measure v(t) for the circuit of Figure 8.3).

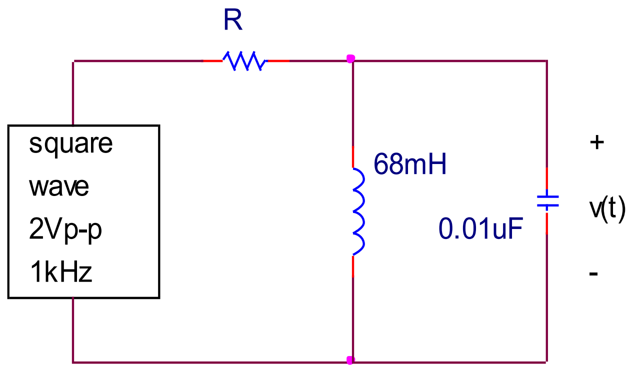


Figure 8.5

EXPERIMENT NINE
IMPEDANCE AND ADMITTANCE MEASUREMENT

EQUIPMENT NEEDED:

1) Oscilloscope
2) Function Generator
3) Resistors, Capacitors, Inductor
4) ELVIS

THEORY

A simple way of solving for steady state solution of circuits with sinusoidal input is to convert the voltages and currents to phasor notation. The voltage v(t) and current i(t) can be converted to phasor notation as follows:

$$v(t) = V\cos(\omega t + \theta) \qquad (9.1)$$

$$i(t) = I\cos(\omega t + \phi) \qquad (9.2)$$

The ratio of the phasor voltage to the phasor current is defined as the impedance.

$$Z \equiv \frac{V}{I} \qquad \angle Z = \theta - \phi \qquad (9.3)$$

In general impedance is complex. The real part of impedance is known as resistance and the imaginary part as reactance.

$$Z = R + jX \qquad (9.4)$$

Reactance for an inductance and a capacitor are given below in (9.5) and (9.6) respectively.

$$X_L = \omega L \qquad (9.5)$$

$$X_C = -\frac{1}{\omega C} \qquad (9.6)$$

The ratio of the phasor current to the phasor voltage is defined as the admittance.

$$Y \equiv \frac{I}{V} \qquad \angle Y = \phi - \theta \tag{9.7}$$

Like impedance, admittance is also complex. The real part of admittance is known as conductance and the imaginary part as susceptance.

$$Y = G + jS \tag{9.8}$$

Susceptance for an inductance and a capacitor are given below in (9.9) and (9.10) respectively.

$$S_L = -\frac{1}{\omega L} \tag{9.9}$$

$$S_C = \omega C \tag{9.10}$$

All the rules of circuit analysis that were covered for the case of pure resistive circuits apply here. The exception is that all voltages and currents are phasors, and resistors are replaced by the impedances.

Kirchhoff's Laws:

Kirchhoff's Voltage Law: The algebraic sum of the voltages around any closed path is zero.

$$\sum_{n=1}^{N} V_n = 0 \qquad (9.11)$$

Kirchhoff's Current Law: The algebraic sum of the currents at any node is zero.

$$\sum_{n=1}^{N} I_n = 0 \qquad (9.12)$$

Series Circuit:

A series circuit in terms of phasors and impedance is shown in Figure 9.1. The current I in the series circuit is the same through all elements and other rules are similar to those of resistive circuit.

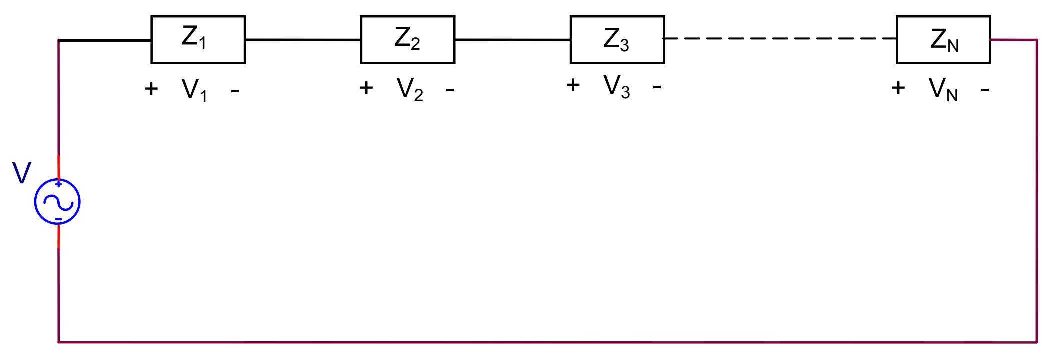


Figure 9.1

KVL: $V = V_1 + V_2 + V_3 + \cdots + V_N$ (9.13)

Total Impedance: $Z_S = Z_1 + Z_2 + Z_3 + \cdots + Z_N$ (9.14)

Voltage Divider Rule:

$$V_1 = \left(\frac{Z_1}{Z_S}\right)V$$

$$V_2 = \left(\frac{Z_2}{Z_S}\right)V$$

$$V_3 = \left(\frac{Z_3}{Z_S}\right)V \qquad (9.11)$$

$$\vdots$$

$$V_N = \left(\frac{Z_N}{Z_S}\right)V$$

Parallel Circuit:

A parallel circuit in terms of phasors and admittances is shown in Figure 9.2. The voltage V across parallel elements is the same and other rules are similar to those of resistive circuit.

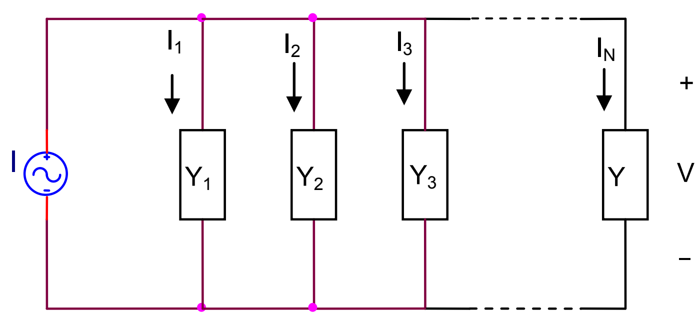


Figure 9.2

$$\text{KCL} \; : \; I = I_1 + I_2 + I_3 + \cdots + I_N \qquad (9.12)$$

Total Admittance: $$Y_P = Y_1 + Y_2 + Y_3 + \cdots Y_N \qquad (9.13)$$

Current Divider Rule:

$$I_1 = \left(\frac{Y_1}{Y_P}\right)V$$

$$I_2 = \left(\frac{Y_2}{Y_P}\right)V$$

$$I_3 = \left(\frac{Y_3}{Y_P}\right)V \qquad (9.14)$$

$$\vdots$$

$$I_N = \left(\frac{Y_N}{Y_S}\right)V$$

<u>Phase Measurement</u>

In general phasor voltages and current as well as impedances are complex. Hence measuring the phase becomes as important as the amplitude. The input voltage source can be considered as having zero degree phase shift and the phase of other voltages and/or currents in a circuit are measured with respect to the input. There are actually two methods of measuring the phase difference between two sinusoids and these methods are explained below.

Method 1 - Dual Trace

Consider two sinusoids having frequency $\omega = 2\pi f$. For the ωt axis, the difference between peaks is ϕ, measured in radians, and this is the difference in phase between v_1 and v_2. (Note that v_2 lags v_1.) However, on an oscilloscope, the axis is time t.

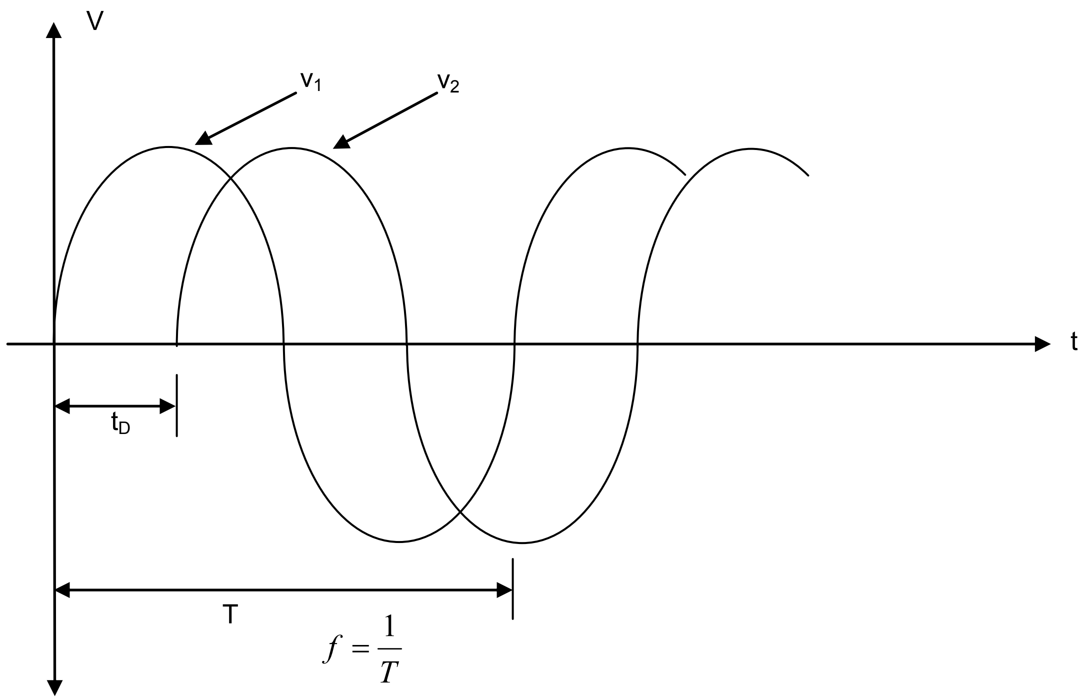


Figure 9.3

Since: $$t_D = \frac{\phi}{\omega} \qquad (9.15)$$

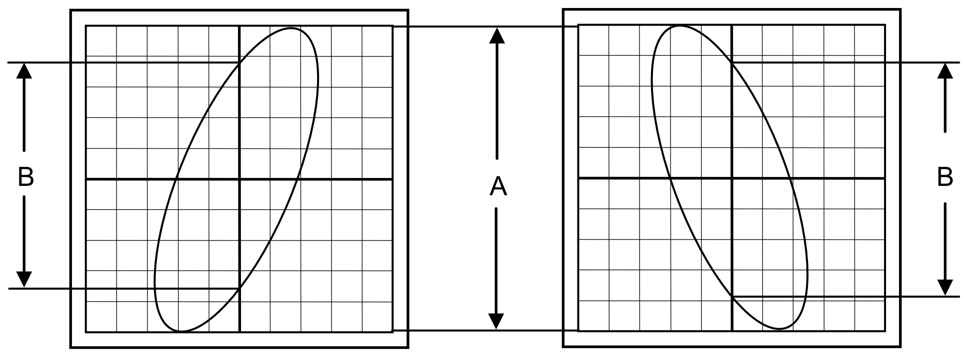
then $$\phi = \omega t_D \quad \rightarrow \quad \phi = \frac{2\pi t_D}{T}\ radians \qquad (9.16)$$

The phase angle can also be expressed in degrees:

$$\phi = 2\pi\left(\frac{t_D}{T}\right)\left(\frac{360^\circ}{2\pi}\right) \quad \rightarrow \quad \phi = \left(\frac{t_D}{T}\right)(360^\circ) \qquad (9.17)$$

Method 2 - Lissajous Pattern

An alternative procedure for measuring phase is to apply one sinusoid to the vertical axis y of a scope and one to the horizontal axis x. The resulting ellipse, called a Lissajous pattern, is shown. (Figure 9.4)



0º < φ < 90º or
270º < φ < 360º

90º < φ < 180º or
180º < φ < 270º

Figure 9.4

$$\phi = \sin^{-1}\frac{B}{A} \qquad (9.18)$$

Preliminary Calculations:

1. For the circuit of Figure 9.5, calculate Z_1, Z_2, and Z_S, the impedances of the series R-L, the series R-C, and the entire circuit, respectively.

a. f = 1kHz C = 0.1μF

b. f = 1kHz C = 1μF

c. f = 500Hz C = 0.1 μF

2. For the three cases in part 1, calculate the phasors V_1, V_2, and I (corresponding to v_1 (t), v_2 (t), and i(t), respectively).

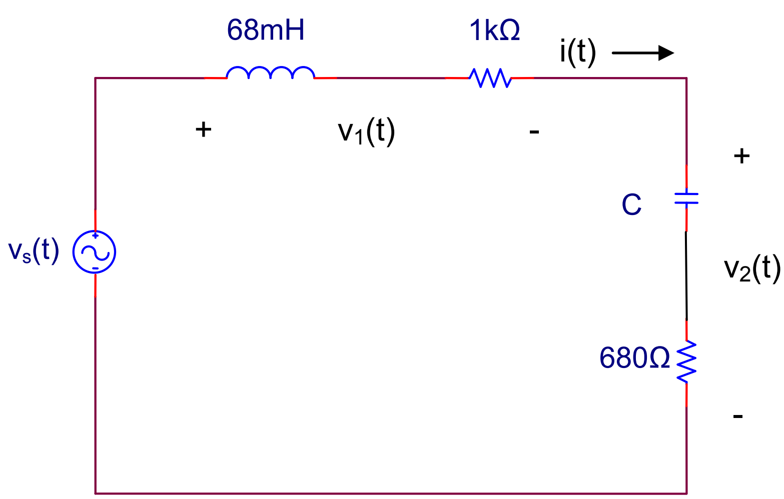


Figure 9.5

Procedure:

1. Construct the circuit of Figure 9.5. Set the function generator for a 1V peak sinusoid with a frequency of 1kHz. Set C = 0.1μF. Use the oscilloscope to measure the magnitude of the phasors v_1 and v_2.

2. Obtain the magnitude and phase angle of the current phasor, I, by observing the voltage across the 680Ω resistor.

3. Draw a phasor diagram with V_1*, V_2, V_S, and I. Include both the measured and theoretical values. Show that $V_S = V_1 + V_2$ (graphically.)

4. Using the measured values of V_1, V_2, V_S, and I, calculate the impedances (mag. and phase) Z_1, Z_2, and Z_S (as defined in pre-lab). Show, graphically, that $Z_S = Z_1 + Z_2$. Compare the calculated (experimental) values to those obtained in the pre-lab.

5. Repeat 1-4 for f = 1kHz and C = 1μF.

6. Repeat 1-4 for f = 500Hz and C = 0.1μF.

*Recall that the grounds in the 2 oscilloscope inputs are common. Keep this in mind so that you do not ground out part of your circuit.

EXPERIMENT 10
FREQUENCY RESPONSE

EQUIPMENT NEEDED:

1) Oscilloscope
2) Function Generator
3) Resistors, Capacitors, Inductor
4) ELVIS

THEORY

In the previous experiment, phasor voltages and currents were computed and measured for a sinusoidal input at frequencies of 500Hz and 1kHz. Since impedances of inductors and capacitors are dependent on frequency of the input, the phasor voltages and currents are also frequency dependent.

The frequency response considers the input-output relation in terms of amplitude and phase for a range of desired frequencies. As an example consider a phasor series circuit shown in Figure 10.1.

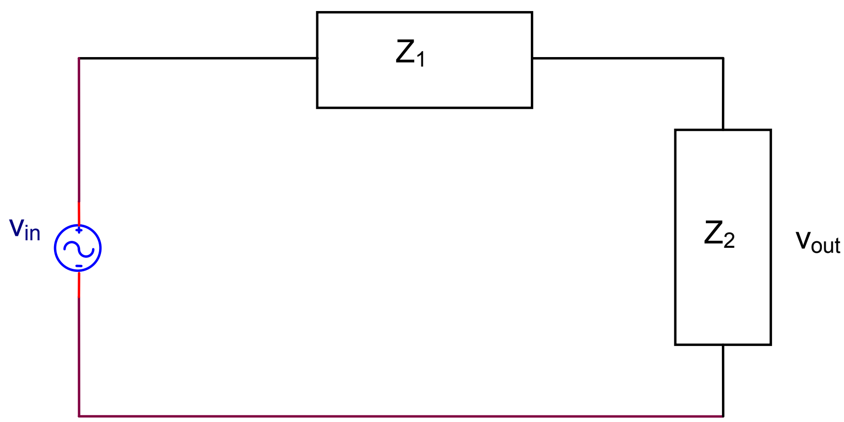


Figure 10.1

The system transfer function H(jω) is defined as

$$H(j\omega) \equiv \frac{v_{out}}{v_{in}} \qquad (10.1)$$

Using voltage division: $v_{out} = \left(\frac{Z_2}{Z_1 + Z_2}\right) v_{in} \quad \rightarrow \quad H(j\omega) = \frac{Z_2}{Z_1 + Z_2}$ (10.2)

In general H(jω) is complex, that is

$$H(j\omega) = |H(j\omega)| \angle \phi(j\omega) \qquad (10.3)$$

The term |H (jω)| is known as the amplitude response and φ (jω) is known as the phase response. Both |H (jω)| and φ (jω) are dependent on the input frequency ω.

Preliminary Calculations:

1. For the circuit in Figure 10.2
   a. Find an expression for H (jω) = V_2/V_1, [V_2 and V_1 are the phasors for the voltages $v_2(t)$ and $v_1(t)$ respectively].
   b. Sketch | H (jω)| versus ω, indicating the peak value and half power point.
      Note: the half power point is the frequency at which |H (jω) | is reduced to 1/√2 of its peak value.
   c. Sketch arg H (jω) versus ω.

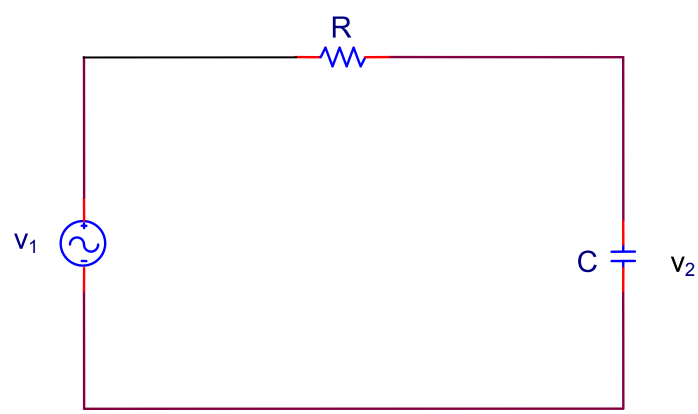


Figure 10.2

2. Repeat part 1 for the circuit of Figure 10.3.

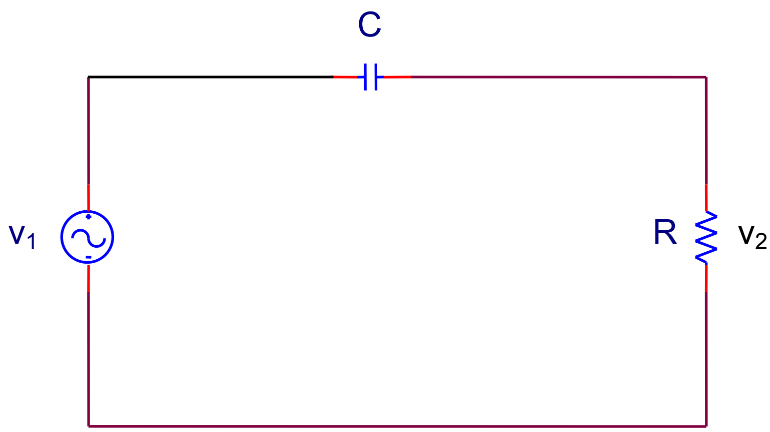


Figure 10.3

3. Repeat part 1 for the circuit of Figure 10.4.

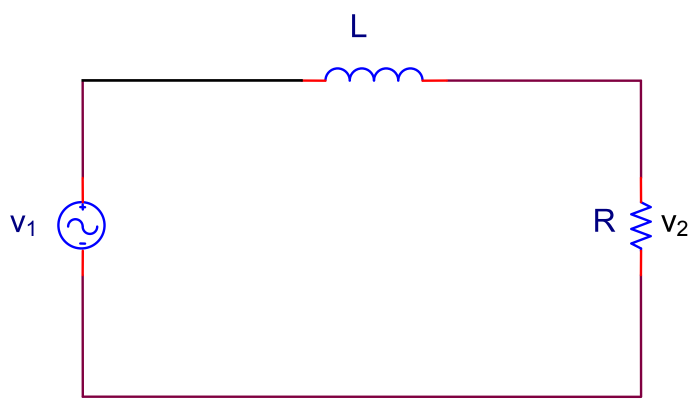


Figure 10.4

Procedure:

1. Construct the circuit of Figure 10.2, with R = 5.1kΩ and C = 0.01μF. Set the sine wave generator for a peak amplitude of 1 volt. Vary the frequency from 10 Hz to 100 kHz and use the oscilloscope to measure the magnitude and phase angle (with respect to the input) of v_2. Make sure that the input remains at 1V as you vary the frequency. Use your measurements to plot the magnitude and phase angle versus frequency. Note: Since v_1 = 1V and φ_1=0º, observing v_2 (jω) is the same as observing H (jω).

2. Determine the experimental half power point from your magnitude plot. Compare it to the theoretical value obtained in the pre-lab.

3. Use the SPICE AC analysis to plot the magnitude and phase angle of V_2 versus frequency. Compare these to your experimental results.

4. Repeat 1-3 for the circuit of Figure 11.3 with R = 5.1kΩ and C = 0.01μF.

5. Repeat 1-3 for the circuit of Figure 11.4 with R = 3.3kΩ ohms and L = 68mH.

EXPERIMENT 11
PASSIVE FILTERS

EQUIPMENT NEEDED:

1) Oscilloscope
2) Function Generator
3) Resistors, Capacitors, Inductors
4) ELVIS

THEORY

Frequency response and the system transfer function $H(j\omega)$ were discussed in the theory of the previous experiment. The definitions of resonant frequency ω_r, bandwidth BW and quality factor Q are presented here for a second order RLC network. Consider the amplitude response of a network (Figure 11.1)

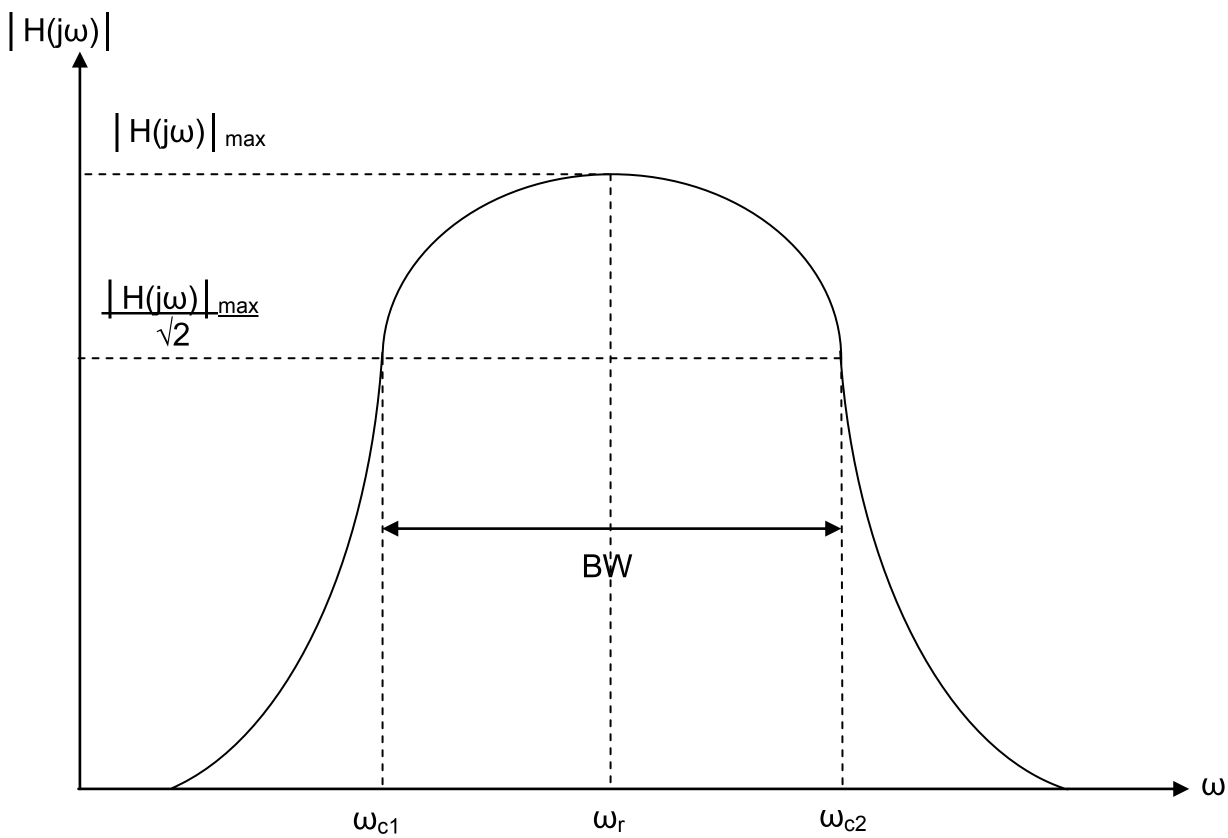


Figure 11.1 – Amplitude Response

Resonant Frequency: The frequency at which the response amplitude is maximum is known as the resonant frequency. This frequency is denoted by ω_r.

Cutoff Frequency: The frequency (frequencies) at which the response amplitude is $1/\sqrt{2}$ of maximum is (are) known as the cutoff frequency (frequencies). Figure 11.1 shows two such frequencies denoted by ω_{c1} and ω_{c2}. Sometimes these frequencies are referred to as corner frequencies or half-power frequencies.

Bandwidth:

The width of the frequency between the cutoff frequencies ω_{c1} and ω_{c2} is known as the bandwidth. That is,

$$BW = \omega_{c2} - \omega_{c1} \qquad (11.1)$$

Quality Factor:

The quality factor Q is a measure of the sharpness of peak in a resonant circuit, and is defined as:

$$Q = \frac{\omega_r}{BW} \qquad (11.2)$$

Hence smaller Q means larger bandwidth and larger Q means smaller bandwidth. In general the amplitude response $|H(j\omega)|$ is not symmetrical about the resonant frequency. Normally for a large value of Q (>5) the amplitude response $|H(j\omega)|$ is symmetrical, in which case we can write

$$\omega_{c1} = \omega_r - \frac{BW}{2} \qquad (11.3)$$

and

$$\omega_{c2} = \omega_r + \frac{BW}{2} \qquad (11.4)$$

<u>Example</u>

As an example consider the parallel RLC network shown in Figure 11.2.

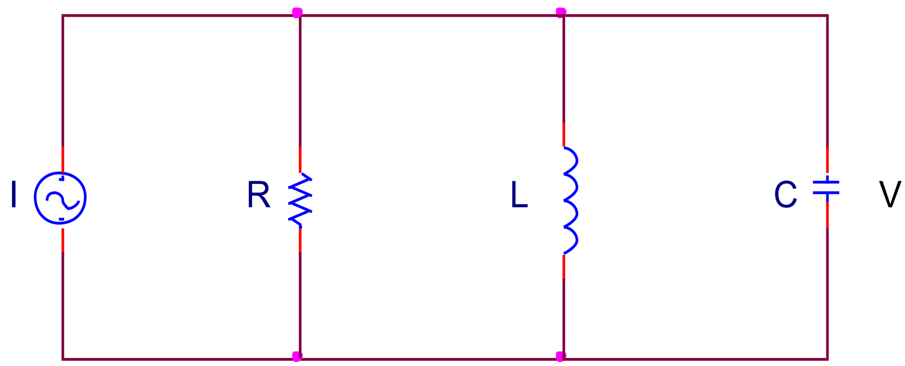


Figure 11.2

Let us define the transfer function H (jω) as V (jω) / I (jω), which is the total impedance.

Hence:

$$H(j\omega) = \frac{1}{\frac{1}{R} + j\left(\omega C - \frac{1}{\omega L}\right)} \qquad (11.5)$$

The amplitude response |H (jω)| is maximum if:

$$\omega C - \frac{1}{\omega L} = 0 \qquad (11.6)$$

The cutoff frequencies occur when

$$\left|H\left(j\omega_{c1,c2}\right)\right| = \frac{1}{\sqrt{2}}\left|H\left(j\omega\right)\right|_{MAX} \qquad (11.7)$$

Therefore:

$$\frac{R}{\sqrt{2}} = \frac{1}{\sqrt{\left(\frac{1}{R}\right)^2 + \left(\omega C - \frac{1}{\omega L}\right)^2}} \qquad (11.8)$$

Solving (11.8) results in:

$$\omega_{c1} = -\frac{1}{2RC} + \sqrt{\frac{1}{LC} + \frac{1}{4R^2C^2}} \qquad (11.9)$$

and

$$\omega_{c2} = \frac{1}{2RC} + \sqrt{\frac{1}{LC} + \frac{1}{4R^2C^2}} \qquad (11.10)$$

The bandwidth BW is evaluated by:

$$\omega_{c2} - \omega_{c1} = \frac{1}{RC} \qquad (11.11)$$

and the quality factor Q is

$$Q = \omega_r RC \qquad (11.12)$$

The above equations were obtained for a parallel RLC circuit. Similar procedure should be used to obtain ω_r, BW and Q for any other second order RLC network with sinusoidal input.

Preliminary Calculations

1. a. Derive an expression for $H(j\omega) = v_2 (j\omega) / v_1 (j\omega)$ for the circuit of Figure 11.3.

   b. Sketch $|H (j\omega)|$ vs. ω.

   c. What is the resonant frequency of this circuit?

   d. What is the bandwidth?

   e. What is the Q?

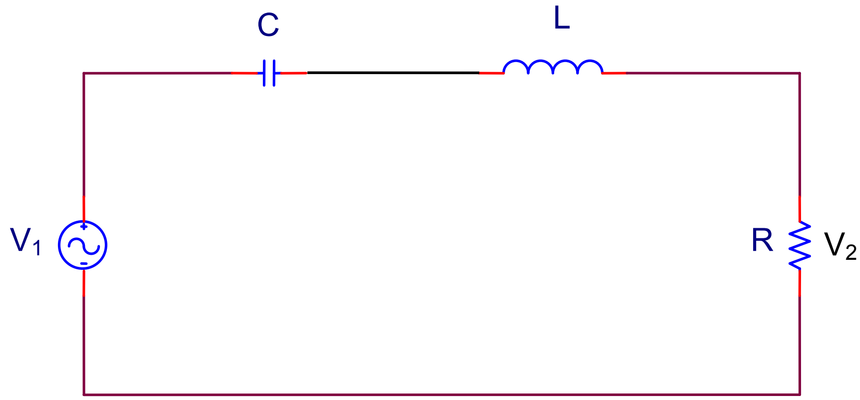


Figure 11.3

2. Repeat part 1 for the circuit of Figure 11.4.

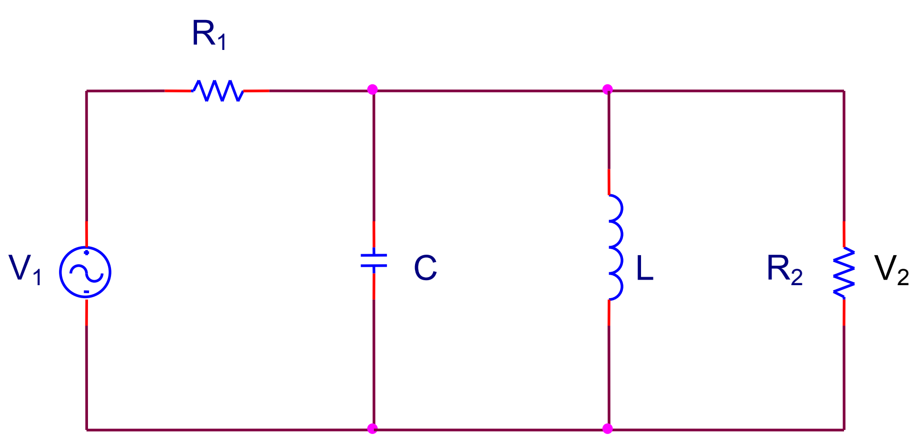


Figure 11.4

Procedure:

1. a. Construct the circuit of Figure 11.3 with R = 5.1kΩ, L = 68mH, C = 0.01μF. Set the sine wave generator for a 1V peak sinusoid. Vary the frequency from 10Hz to 100kHz and use the oscilloscope to measure the magnitude of V_2. Make sure that the input voltage remains constant throughout the measurements. Plot the magnitude of V_2 vs. frequency on semi-log graph paper (frequency on log scale).
   b. Use the above plot to find the resonant frequency of the circuit. How does it compare to the theoretical value obtained in the pre-lab?
   c. Use your graph to determine the experimental bandwidth of the circuit. How does this value compare to the value in the pre-lab?
   d. Use the AC analysis of SPICE to plot the magnitude of V_2 versus frequency. How do your results compare to those obtained in part a?

2. Repeat part 1 for R = 2kΩ ohms.

3. Repeat part 1 for the circuit of Figure 11.4 with R = 15kΩ, L = 68mH, and C = 0.01μF.

4. Repeat part 3 for R = 5.1kΩ.

EXPERIMENT 12
CIRCUIT DESIGN EXPERIMENT

EQUIPMENT NEEDED:

1) Oscilloscope
2) Function Generator
3) DMM
4) Resistors, Capacitors, Inductors
5) ELVIS

Circuit I

Choose values for resistors R_1, R_2, R_3, and R_4 in figure 12.1 to produce the voltages shown in the schematic.

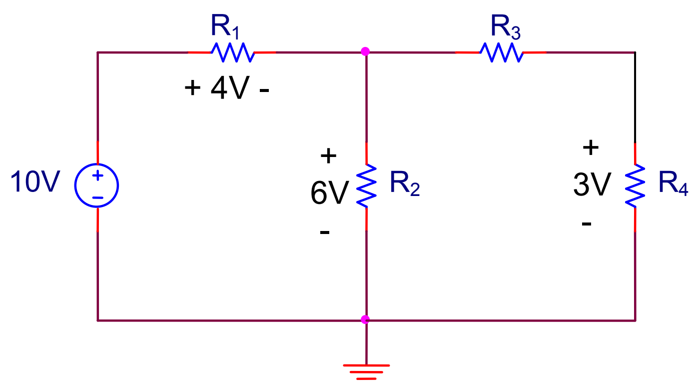


Figure 12.1

Circuit II

For the circuit shown in figure 12.2, choose resistor values that will maximize the power dissipated in the 2.7kΩ resistor.

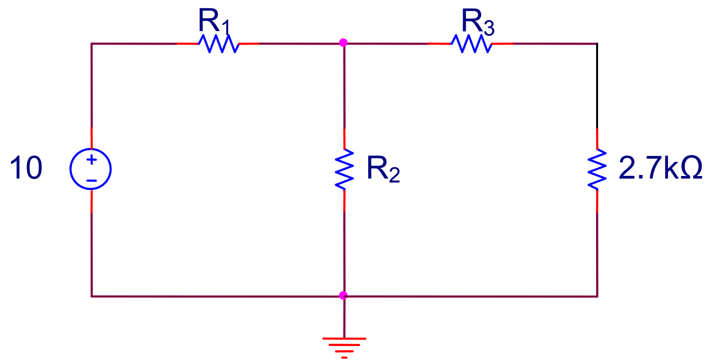


Figure 12.2______________________

Circuit III

Using the circuit shown in figure 12.3, choose values for R and C that will duplicate the wave-form shown in figure 12.4.

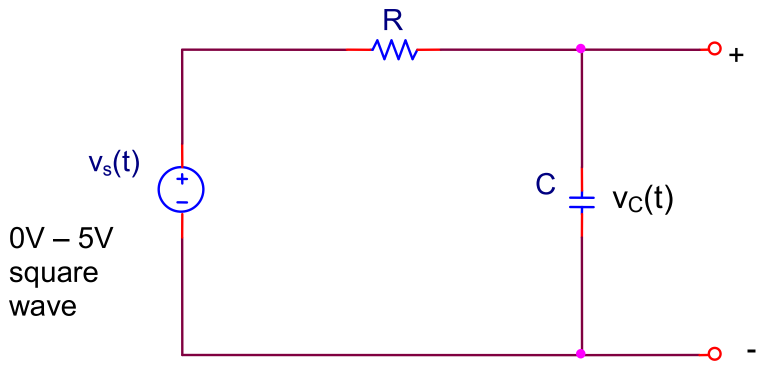


Figure 12.3

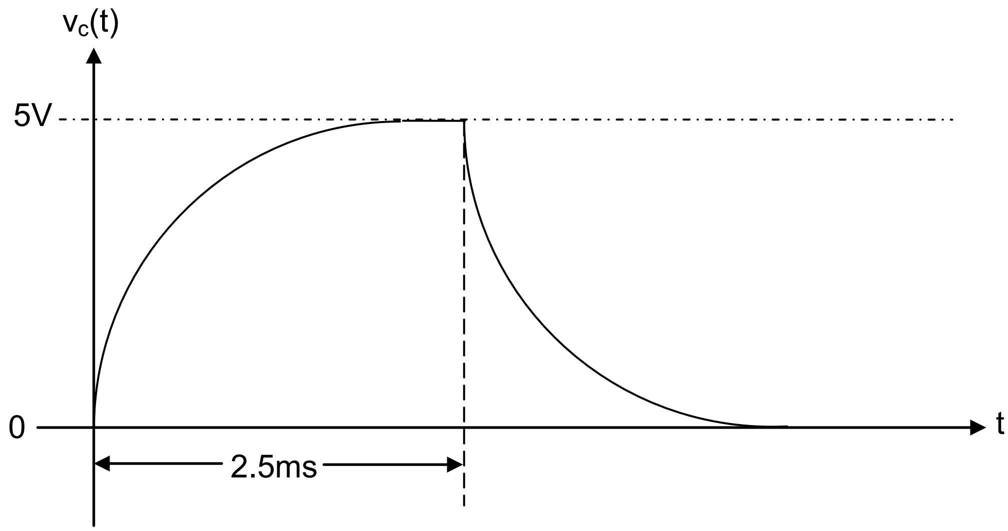


Figure 12.4

APPENDIX I

Resistor
Color Coding

RESISTOR VALUES

Resistors are available in certain standard values. The 5% tolerance resistors available in this lab have values of:

$$A \times 10^b$$

where A = 10, 11, 12, 13, 15, 16, 18, 20, 22, 24, 27, 30, 33, 36, 39, 43, 47, 51, 56, 62, 68, 75, 82, 91, 100
and b = 1, 2, 3, 4, 5 or 6
(i.e. 2K ohms and 2.2K ohms are available, but 2.1K ohms is not). The value of a resistor is read in the following way:

4-BAND RESISTORS

±10%
± 5%   BANDS 1 2 Multiplier Tolerance

Band 1		Band 2		Multiplier		Resistance	
<u>1st Digit</u>		<u>2nd Digit</u>				<u>Tolerance</u>	
Color	Digit	Color	Digit	Color	Multiplier	Color	Tolerance
Black	0	Black	0	Black	1	Silver	±10%
Brown	1	Brown	1	Brown	10	Gold	± 5%
Red	2	Red	2	Red	100	Brown	± 1%
Orange	3	Orange	3	Orange	1,000		
Yellow	4	Yellow	4	Yellow	10,000		
Green	5	Green	5	Green	100,000		
Blue	6	Blue	6	Blue	1,000,000		
Violet	7	Violet	7	Silver	0.01		
Gray	8	Gray	8	Gold	0.1		
White	9	White	9				