Experiments with
Electric Circuits

by Sid Antoch

ZAP Studio Instructional Publication

Experiments with Electric Circuits

ISBN-13: 978-1-935422-11-2
ISBN-10: 1-935422-11-1

Trademark Information

Pictures, drawings, and diagrams by Sid Antoch
Edited by Anna Antoch

Cover photos by NASA
Hubble Space Telescope, May 2009.
International Space Station's solar array panels, August 2008.

ZAP Studio LLC PO Box 1150 Philomath, OR 97370

www.zapstudio.com

Introduction

Sid Antoch has taught electrical engineering circuits courses at Portland State University and Portland Community College, and electrical engineering technology courses at Portland Community College and Tektronix. This lab book is in part the result of this experience.

Experiments in this manual are intended for the laboratory component, or as a supplement to the laboratory component of an electrical engineering electric circuits course. The experiments emphasize the use of spreadsheets and simulation software. Several of the experiments refer to the specific application of a Tektronix TDS type oscilloscope to acquire experimental data. These may be omitted or modified if the lab does not have TDS type oscilloscopes.

Analysis suggestions are provided for each experiment. These may be used as is or may be modified by the instructor according to the lab curriculum requirements. Experiments may be selected according to the accompanying textbook and course emphasis.

Equipment List

Digital Oscilloscope with 10X probes, Function Generator, Digital Multi-Meter.
Solder-less Breadboard. Power Supply: 0-volts to 6-volts and 0-volts to ±15-volts.
Phase Tripler (refer to appendix 2). Computer with word processor, spreadsheet, simulation, and data acquisition software.

Parts List

Resistors: 8.2, 10, 47, four-100, 220, 330, three-390, 470, 680, 750, 820, three-1K, 1.2k.
Resistors: two-1.5k, 1.8K, 2.2K, 3.3K, 3.9K, two-4.7K, 6.8K, 8.2K, 10K, 12K, 15K, 18K, 22K.
Resistors: 33K, two-100K, two 150K, 1 Meg, 2.2 Meg, all ¼ watt, 5%.
Trim Pots: 100 Ω and 10 kΩ, one turn, breadboard mountable, ¼ or ½ watt.
Capacitors: 1 nF, two-10 nF, 22 nF, 47nF, two-100 nF, 1 μF (non-polarized), 5%.
Inductors: 10mH, three-100 mH (50mA),
Light bulb: Type 2182 (14 V, 80 mA).
Analog meter, 1 mA < 1000 Ω (or 100 μA < 10,000 Ω).
Transformer: 12.6 VAC center tapped.
Audio Transformer, 1000 Ω, center tapped, to 8 Ω, 200 mW.
Loudspeaker, 8 Ω, 200 mW minimum, (2 to 4 inch).
Op-amp: LM741 or uA741
Light Emitting Diode (LED).
Transistor: 2N3904.

Contents

Experiment 1: Electrical Resistance and the Resistor

Introduction

Ohm's law is the most fundamental equation in electric circuit analysis. It states that the amount of electric current flowing in a circuit is directly proportional to the voltage applied to the circuit, and inversely proportional to its resistance.

$I = \frac{V}{R}$ I is current in amperes, V is potential difference in volts, and R is resistance in ohms.

Resistors are used in electric circuits to control the flow of current. Resistors are commercially available which have a specific amount of resistance and power dissipation ability. The amount of resistance is usually marked on the resistor using a color code. The power dissipation is determined by the physical size of the resistor.

An "ohmmeter" is used to measure resistance. Most ohmmeters are part of an instrument that is also capable of measuring other electrical quantities, such as voltage and current. These are typically called "multi-meters", and since they usually have a digital display, they are called "digital multi-meters" or "DMM's" for short. To use the DMM you need to know how to set it to make the desired measurement (function), and how to set it for best accuracy (range).

Objectives

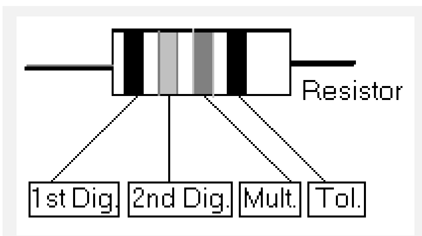
The purpose of this lab exercise is to learn how to measure resistance with the DMM. An error analysis will compare the measured resistor values to the labeled resistor values using a spreadsheet. In addition, you will measure the resistance of series and parallel combinations of resistors, and compare the results to theoretical calculations based on equations provided.

Series and parallel connections will be made using a solder-less breadboard. The object of this part of the exercise is to learn to use the breadboard. Theoretical knowledge of series and parallel resistor connections is not expected.

Resistance values are read using the color code given below.

Standard Resistor Color Code

Color	Value	Color	Value
Black	0	Blue	6
Brown	1	Violet	7
Red	2	Gray	8
Orange	3	White	9
Yellow	4	Gold	0.1 / 5%
Green	5	Silver	0.01 / 10%



This color code is for “standard” resistors with an accuracy rating, or “tolerance”, of ±5% or ±10%. That is, their value is guaranteed to be within ±5% or ±10% of their labeled value.

Their colors are read from left to right. The first two color bands represent the first two significant digits of the resistor value. The color of the third band represents a multiplier of 10^N, where N is the value represented by the color.

The fourth band is always gold or silver, which indicates a tolerance of ±5% or ±10%. The first band is never gold or silver. *So to read a resistor’s value correctly, the gold or silver band must be on the right.*

For example, a resistor whose first band is red, second band is yellow, third band is orange, and fourth band is gold, has a value of 24,000-ohms (24×10^3), and a tolerance of 5%.

Resistance Measurement

A digital multi-meter (DMM) will be used for measuring the resistance values. The instructor may explain the operation of the instrument before you use it for the first time. You may also check to see if an instruction manual is available for the instrument. The DMM will have buttons and/or switches to set its function and range.

Set the function to "OHMS". Some meters are capable of automatically setting the range to get the most accurate reading, which is related to the number of significant digits displayed. You should be able to get at least three significant digits of accuracy. Experiment with the range settings when making the measurements specified in the procedure below.

The power rating of each resistor is determined by its physical size. Smaller dimensions represent a smaller power handling capability. A sample of several different size resistors should be available in the lab. A very common power rating is ¼ watt. If a ¼ watt resistor dissipates more than ¼ watt it will get excessively hot and may burn out.

Procedure

Equipment and Parts

DMM and Breadboard.
Resistors: 1K, 4.7K, 10K, ¼ watt, 5% or 10% tolerance.

Part 1: Measurements and the Spreadsheet

Do not touch the metal tips of the DMM probes when making measurements.

1. Use the resistor color code to select the 1K, 4.7K, and 10K resistors. Determine their tolerance. Measure the values of the resistors with the DMM to at least three significant digits.

2. Enter the results into a spreadsheet. Calculate the deviation of each resistor's measured value compared to its labeled value. Calculate the percent deviation of each resistor's measured value compared its labeled value.

Use the spreadsheet layout shown below to do the calculations.

	A	B	C	D	E	F
1	Resistor	Labeled Val.	Measured Val.	Deviation	Labeled %	Measured %
2	1	1000				
3	2	4700				
4	3	10000				
5						
6						

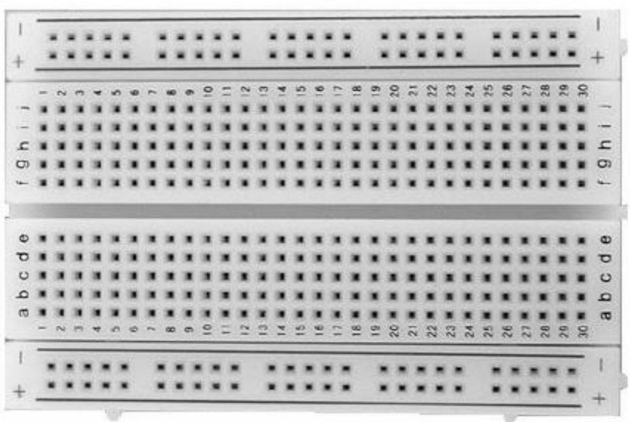
Deviation: =C2-B2   Percent deviation: =(D2/B2)*100

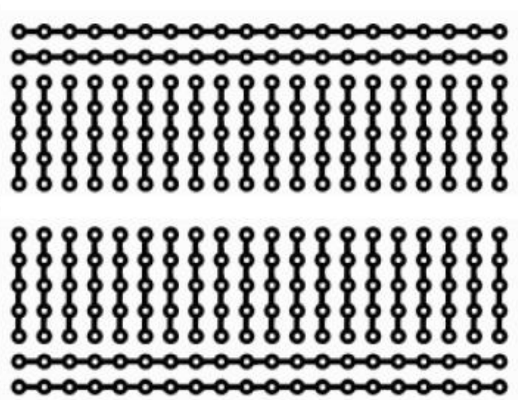
Enter the expression for deviation into cell D2 and percent deviation into cell E2. Use the "fill down" feature of the spreadsheet to calculate rows 3 and 4.

Part 2: Series and Parallel Connections

Before starting this exercise (and the other exercises in this manual) you need to have a way of connecting electronic parts together into a circuit. An easy and very common method to quickly connect parts together is to use the "solder-less breadboard", also called a "prototyping board" or "protoboard". The board has holes 0.1 inches apart into which component leads can be inserted.

Solder-less breadboards are available from a variety of manufacturers and sources, in a variety of sizes, but they all have the same arrangement of the holes and connections



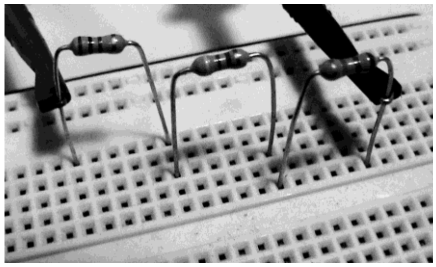


The picture above on the left shows a typical breadboard. Components such as resistors, capacitors, transistors, integrated circuits, and wires can be plugged into it. The picture above on the right shows how the holes are connected. You should memorize these connections.

Components such as resistors can be connected in series, parallel, and in a combination of series and parallel. The following exercises show how to connect resistors in series and parallel and measure the resistance of the series and parallel combinations.

The measurements will be compared to the theoretically expected values using the equations provided. If a measurement does not agree with the calculation, check the breadboard connections and the labeled values of the resistors.

1. Connect your 1K, 4.7K and 10K resistors in series. Measure and record the resistance, R_{TS}, of the series combination as shown in the circuit's schematic diagram below. A connection example is shown on the right.



Ohmmeter leads connect between points a and b.

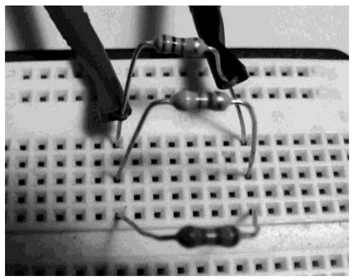
R_{TS} ____________________ (measured total series resistance)

2. Calculate the theoretical resistance of this series combination with the equation:

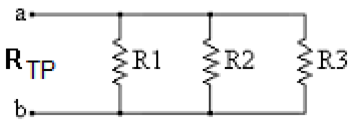
$$R_{TS} = R1 + R2 + R3.$$

Use the measured values of the resistors from part 1. Enter the equation into your spreadsheet and have the spreadsheet do the calculation. Also enter the measured value into the spreadsheet as shown in the example spreadsheet on the next page.

3. Connect the 1K, 4.7K and 10K resistors in parallel as shown in the diagram below and picture on the right. Measure and record the resistance, R_{TP}, of the parallel combination.



Ohmmeter leads are connected between points a and b.



R_{TP} ____________________ (measured total parallel resistance)

4. Calculate the resistance of this parallel combination using the equation below.

$$\frac{1}{R_{TP}} = \frac{1}{R1} + \frac{1}{R2} + \frac{1}{R3} \quad \text{so that} \quad R_{TP} = \frac{1}{\left(\frac{1}{R1} + \frac{1}{R2} + \frac{1}{R3}\right)}$$

Use the measured values of the resistors from part 1. Enter the equation into your spreadsheet and have the spreadsheet do the calculation. Also enter the measured value of the series resistance and the measured value of parallel resistance into the spreadsheet as shown in the example spreadsheet below.

.

	A	B	C	D	E	F
1	Resistor	Labeled Val.	Measured Val.	Deviation	Labeled %	Measured %
2	1	1000	996	-4	0.05	-0.4
3	2	4700	4760	60	0.05	1.276595745
4	3	10000	9720	-280	0.05	-2.8
5						
6						
7		Calculated Val.	Measured Val.	Deviation	% Deviation	
8	Series	15476	15390	-86	-0.555699147	
9	Parallel	759.3125376	762.2	2.8874624	0.380273246	
10						

Equation in B8: =C2+C3+C4 Equation in B9: =1/(1/C2+1/C3+1/C4)

LAB REPORT

1. Open a word processor document and save it as: “Experiment 1 Report”. Use the following format:

   Student name and lab partner name (if applicable).
   Course number: Lab experiment number and name.

   Example:

   George Jones and Sally Smith

   ENGR221: EXPERIMENT 1: Electrical Resistance and the Resistor

2. Copy your spreadsheet results and paste them into the document.

   The instructor will specify how to turn in the report. You may just need to show the resulting document on the lab computer, or the instructor may also ask for additional analysis, including a more comprehensive lab report.

Experiment 2: Voltage, Current, and Power

Introduction

Electrical devices which do work consume energy. Power is the rate that energy is consumed. Equations describing the relationship between power, energy, voltage, and current are given below. P is power in watts, W is work (energy) in joules, I is in amperes, and V is in volts:

$$P = \frac{dW}{dt} \qquad P = IV \qquad V = IR$$

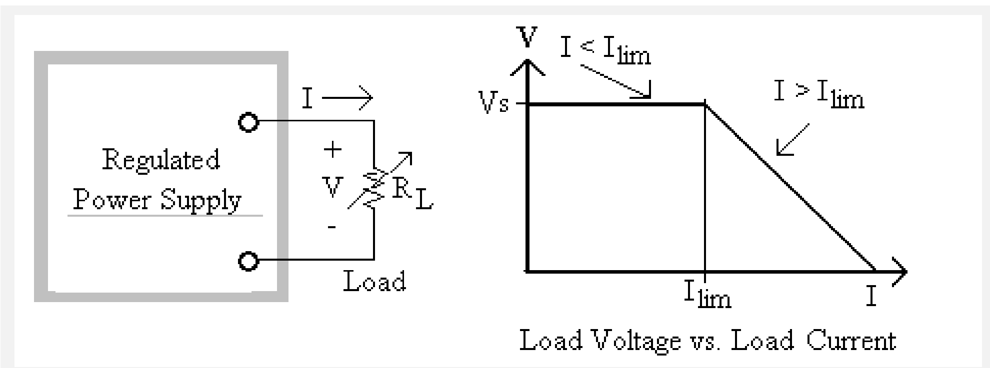
All electric power sources are basically sources of potential difference or voltage. Examples include batteries, generators, solar cells, and power supplies.

A typical lab power supply has variable (adjustable) voltages from 0 to about 30 volts at a maximum current of about 1 ampere. Maximum ratings are often printed on the front panel. Note what maximum voltages and currents are available from your power supply. You may refer to the instrument's manual, if available, for more details.

Voltage Sources

Voltage sources used in circuit analysis are usually "ideal" sources. By ideal we mean that the value of the voltage of the source does not change regardless of how much current it supplies. Practical sources are limited in the amount of current they can supply.

A voltage regulated power source will supply a constant output voltage, V_S, when the output current, I, is less than its maximum limit, I_{lim}. Some power supplies have an adjustable current limit, I_{lim}, and they will supply a constant output voltage for load currents less than I_{lim}. Refer to the graph below.



A power supply will be used to supply power to a light emitting diode (LED) in part 1 and to a light bulb in part 2. The voltage versus current characteristics and power dissipation of both will be measured and compared.

Voltage Measurement

The DMM will be used to measure voltage. *To measure voltage the meter leads are connected across (in parallel with) the device.* Voltage is measured between two nodes. Voltage is a potential difference. You are measuring the potential difference between two nodes.

Current Measurement

To measure current the meter must be connected in series with the circuit so that the current flowing through the ammeter is the same as the current flowing through the circuit. The circuit must be temporarily disconnected to insert the ammeter in series with it.

Current can also be determined by measuring the voltage drop across a known resistance and calculating the current flowing through it using Ohm's law. Current through an LED and an incandescent lamp will be determined from the voltage measured across a series connected resistor.

Objectives

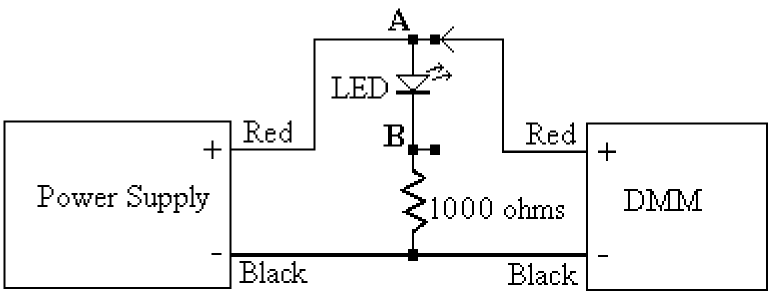
The voltage versus current graph of a resistor is a straight line. The resistance of a resistor is constant and does not change with the current through it. This experiment will show that this is not the case for LEDs and light bulbs. Objectives of this exercise include gaining experience with voltage measurement, spreadsheet calculations, and spreadsheet graphing.

Procedure

Equipment and Parts
Power supply, DMM, and Breadboard Resistors: 10 Ω and 1000 Ω (1/4 watt, 5%). Light bulb: Type 2182 (14 V, 80 mA). Light Emitting Diode (LED).

Part 1: I versus V for an LED

1. Connect the circuit below. The positive lead of the LED is the anode. It is the longer lead and it should be connected to point A on the diagram below. Turn on the power supply and set the voltage to zero. Set the DMM function to volts and set the range to 20-volts.



2. The red DMM lead will be moved between points A and B follows:

(a) Set the power supply voltage to exactly 0.0 V (as read on the DMM). The DMM's red lead should be connected to point A.

(b) Move the DMM's red lead to point B. Measure and record the voltage at B.

(c) Repeat steps (a) and (b) for voltages at point A of:
1.0, 2.0, 3.0, 4.0, 5.0, 6.0, 7.0, 8.0, 9.0, 10.0, 11.0, 12.0 V.

Create a spreadsheet table as shown below. You will have 13 rows of data.

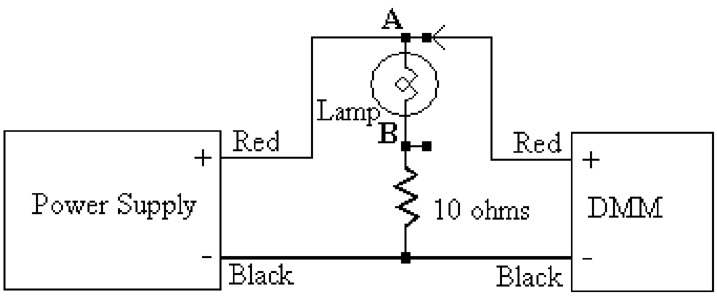
	A	B	C	D	E
1	Volts at A	Volts at B	Volts LED	A LED	Watts LED
3	0	0	0	0	0
4	1				
5	2				

C4 has: =A4–B4 D4 has: =B4/1000 E4 has: =C4*D4

You can automatically fill a column of data. For example, select cell C4 and then click the mouse on the bottom right corner of the cell, and drag the mouse down to fill all of the rows of that column.

Part 2: I versus V for a Lamp

1. Connect the circuit below. The lamp is not "polarized". Its leads can be connected in either direction. Make sure that the power supply output voltage is set to zero. Set the DMM function to volts and set the range to 20 V.



2. The Red DMM lead will be moved between points A and B as follows:

(a) Set the power supply voltage to exactly 0.0 V (as read on the DMM). The DMM's red lead should be connected to point A.

(b) Move the DMM's red lead to point B. Measure and record the voltage at B.

(c) Repeat steps (a) and (b) for voltages (as read by the DMM at point A) of:
1.0, 2.0, 3.0, 4.0, 5.0, 6.0, 7.0, 8.0, 9.0, 10.0, 11.0, 12.0 V.

Create a spreadsheet table as shown below. You will have 13 rows of data.

	A	B	C	D	E
1	Volts at A	Volts at B	Volts Lamp	A Lamp	Watts Lamp
3	0	0	0	0	0
4	1				
5	2				

C4 has: =A4–B4   D4 has: =B4/10   Cell E4 has: =C4*D4

Analysis

1. Use the spreadsheet to graph the current versus voltage characteristics of the LED. Current is on the vertical axis and voltage is on the horizontal axis. Label the graph.

2 Use the spreadsheet to graph the current versus voltage characteristics of the lamp Current is on the vertical axis and voltage is on the horizontal axis. Label the graph.

3. The resistance of the device at a point on the graph is equal to the value of the voltage divided by the value of the current at that point. Use the spreadsheet to calculate the resistance at each point in column F.

   Plot the resistance of the LED as a function of power dissipation using your spreadsheet data (resistance on vertical axis).

   Plot the resistance of the lamp as a function of power dissipation using your spreadsheet data (resistance on vertical axis).

4. Copy each properly labeled graph and paste it into a word document. Your document should have the form below:

   Your Name   Course Number   Date   Experiment Number and Name

   Circuit diagrams with labeled voltages and currents
   LED Current versus Voltage Graph
   Lamp Current versus Voltage Graph
   LED Resistance versus Power Graph
   Lamp Resistance versus Power Graph

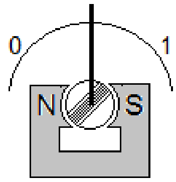
   Brief summary (what you learned from this exercise).

   The above is a short report. The instructor will specify how to turn in the report. The instructor may also ask for additional analysis, including a more comprehensive lab report.

<u>Experiment 3: Analog D'Arsonval Meter</u>

Introduction

An analog meter uses magnetic fields to measure electric current. A coil of wire wound on a bobbin creates a magnetic field whose strength is proportional to the wire's current. The bobbin is suspended in between the magnetic field of a permanent magnet as shown in the illustration on the right. The bobbin and attached needle rotate due to the magnetic force between the bobbin and the permanent magnet.



The wire winding has a resistance which determines one of the meter's characteristics. For example, if the winding resistance is 1000 Ω, and one milliamp causes "full scale" deflection, its "sensitivity" will be given as "1000 ohms per volt". By Ohm's law, 1 volt applied to 1000 ohms will create a current of one milliamp.

So a 1000 Ω, one-milliamp meter is also a voltmeter whose full scale voltage is 1 volt. The voltage range of the meter can be changed to 10 volts by adding a series resistance that will result in a one milliamp current when the applied voltage is 10 volts. The total resistance required to limit the current to 1 milliamp is 10,000 ohms. Connecting a 9000 Ω resistor in series with the meter makes the meter a voltmeter whose range is zero to ten volts.

Objectives

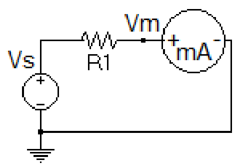
Measure the operating characteristics of an analog milliammeter. Use the measured meter characteristics to design a dual range analog voltmeter. Evaluate the performance of the voltmeter.

Procedure

Equipment and Parts
Power supply, DMM and Breadboard. Analog meter, 1 mA < 1000 Ω (or 100 μA < 10,000 Ω). R1: 1k, ¼ watt, 5% for 1 mA meter (or 10k, ¼ watt, 5% for 100 μA meter). Values of resistors R2 and R3 to be determined.

Part 1: Meter Characteristics

1. Measure and record the value of R1. R1 is the 1k resistor if using a 1 mA meter (or 10k if using a 100 μA meter). Set the power supply voltage to zero. Connect the circuit on the right. Vs is a variable power supply (0 to 10 V). Connect the DMM to read the voltage, Vm, across the meter.



R1 ________________

2. Slowly increase the voltage, Vs, until the analog meter reads full scale. Your eye should be directly above the meter's needle to reduce "parallax error". Measure and record the voltage, Vm, across the meter. Connect the DMM to Vs. Measure and record Vs.

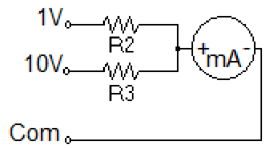
Vm ________________ Vs __________________

3. Calculate the actual meter current, I_{ma}, and resistance, R_{ma}, using the voltages, Vm and Vs, and the measured value of R1.

$$I_{ma} = \frac{Vs - Vm}{R1} = \_\_\_\_\_\_\_\_\_\_\_\_ \qquad R_{ma} = \frac{Vm}{I_{ma}} = \_\_\_\_\_\_\_\_\_\_\_\_$$

Part 2: Meter Design

1. Use the calculated values of I_{ma} and R_{ma} from part 1, step 3, to design a dual range voltmeter circuit, as shown on the right, to measure 1 V full scale and 10 V full scale. Calculate the required values R2 and R3.



R2 _________________ R3 __________________

Use a series combination of resistors for R2 and R3 so that the resistors used in the circuit have measured values within 1% of the calculated values. Use a high value and low value resistor in series.

For example, if R2 = 930 Ω, you could connect the standard values of 910 Ω and 22 Ω in series to obtain 932 Ω. The percent error for this combination is:

$$\%\,Error = \frac{932 - 930}{930} \times 100\% = 0.215\%$$

Measure the resistance of your series combination. You may need to try another combination if your measured resistance is not within 1% of the required resistance. Note that 1% of 1000 Ω is equal to 10 Ω.

2. Test the voltmeter circuit by comparing the analog meter readings to the DMM readings as follows:

a) Connect the DMM to the variable power supply. Connect the 1 V input range of the analog meter (R2) to the variable power supply. The meters will be in parallel.

b) Carefully set the power supply voltage so that the analog meter reads 0.20 V. Observe the analog meter reading from directly above the needle to minimize parallax error.

c) Read the voltage on the analog meter. Record the result in the table below.

d) Repeat steps (b) and (c) for voltages of 0.40, 0.60, 0.80, and 1.00 V.

e) Repeat the procedure for the analog meter's 10 V range and record results in the table below (for 2.0, 4.0, 6.0, 8.0, and 10.0 V).

1 VOLT RANGE				10 VOLT RANGE		
Analog Meter	DMM	% Error		Analog Meter	DMM	% Error
0.2				2.0		
0.4				4.0		
0.6				6.0		
0.8				8.0		
1.0				10.0		

Analysis

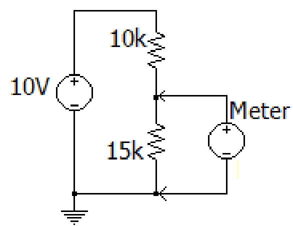
1. Calculate the sensitivity of the analog voltmeter in ohms per volt on the 1 V and 10 V range.

2. Calculate the input resistance of the analog voltmeter on the 1 V and 10 V range.

3. Calculate the percent error for all of the readings and record the results in the table. You could use a spreadsheet to do the calculations.

4. Calculate the average percent error of the analog meter on the 1 V and on the 10 V range.

5. Assuming that the analog meter is as accurate as the digital meter, calculate the voltage that each meter would read across the 15k resistor given that the digital meter has resistance of 1 megohm and the analog meter has a resistance of 10k ohms. Calculate the percent error for each meter.



6. Use the values of I_{ma} and R_{ma} of the meter to design an analog ammeter with a full scale deflection current of 100 mA. Calculate the required value of the shunt resistance (resistance in parallel with the meter). What is the net resistance of the 100 mA meter (shunt resistance in parallel with the meter resistance, R_{ma}).

Experiment 4: DC Measurements and Meter Loading

Introduction

Meters used to measure voltage or current have an internal resistance. Since the meter must be connected to the circuit to make a measurement, the circuit is changed by the resistance of the meter. This is referred to as the meter's "loading effect".

Most meters have amplifiers built in so that only a small amount of power is needed from the circuit, and the loading effects are minimized. But this is not always the case.

A voltmeter is connected in parallel with the circuit across which the voltage is being measured. The loading effect of the meter will be minimal if the meter resistance is much larger than the circuit resistance. Ideally, the voltmeter resistance should be infinite. Practically, most electronic voltmeters (DMM's) have a resistance of 10 megohms.

The effect of the voltmeter resistance, Rm, across a circuit element, Rc, can be calculated using the parallel resistor equation:

$$Rt = \frac{Rc \cdot Rm}{Rc + Rm}$$

An ammeter is connected in series with the circuit in which the current is being measured. The loading effect of the meter will be minimal if the meter resistance is much smaller than the circuit resistance. Ideally, the ammeter resistance should be zero. Practically, the resistance of most electronic ammeters (DMM's) varies with the range setting, with the highest ranges having the least resistance. This is one case where setting the range for the most significant digits may not always result in the most accurate reading.

The effect of the ammeter resistance, Rm, in series with circuit element, Rc, can be calculated using the series resistor equation:

$$Rt = Rc + Rm$$

Even if the meter loading effect is insignificant there will be an uncertainty in the measured value due to measurement errors caused by the accuracy of the meter.

Objectives

A series-parallel circuit will be connected on a solder-less breadboard. Voltage and current measurements will be made on the circuit and the results will be compared to theoretical expectations. The effect of the resistance of the meter will be determined.

Procedure

Equipment and Parts
DMM, Power Supply, and Breadboard. Resistors: 100, 220, 1 Meg, 2.2 Meg, ¼ watt, 5%.

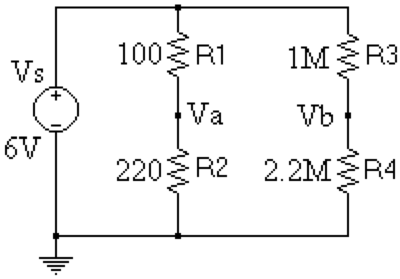
Part 1: Voltage Measurement

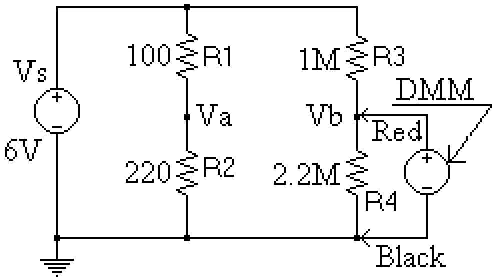
1. Measure and record the values of the resistors in the circuit. You will use these measured resistor values for your calculations and for *PSpice* input in experiment 5 and LTspice input in experiment 6.

   R1 _______________ R2 ________________

   R3 _______________ R4 ________________

2. Connect the circuit below on the left and set the power supply voltage to 6.0 volts.

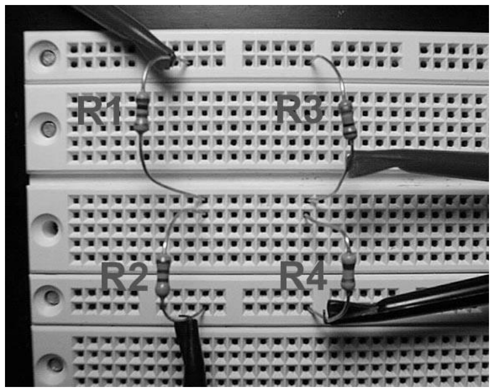




. Lay out the circuit so that it looks similar to the schematic diagram. Use the minimum number of wires to connect the circuit.

The positive power supply lead is connected to the top of R1 and the negative power supply lead is connected to the bottom of R2.

The voltmeter is connected across R4.



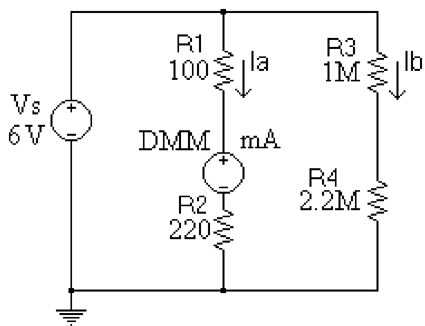
3. Measure and record Va, Vb and Vab (Vab = Va – Vb)

   Va _______________ Vb ______________ Vab __________________

4. Obtain and record the internal resistance, Ri, of your voltmeter. Ri ________________

Part 2: Current Measurement

1. The circuit on the right shows the DMM connected in series with the 100 Ω and 220 Ω resistors. You need to break the connection between R1 and R2 and insert the meter as shown. The meter will read the current through the 100 Ω and 220 Ω resistors.



2. The internal resistance of a DMM on the current ranges varies with the range. Check the manual on your DMM to see what its internal resistances are on the current ranges. If you don't have a manual, your instructor should provide the information. Record the meter resistances below.

Rm(0.2 mA) ______________ Rm(2 mA) ______________

Rm(20 mA) _____________ Rm(200 mA) _____________

3. Measure and record the current through the 100 Ω and 220 Ω resistors with the meter on the 20 mA range.

Ia_{20} ___________________

4. Measure and record the current through the 100 Ω and 220 Ω resistors with the meter on the 200 mA range.

Ia_{200} ___________________

Analysis, Part 1

1. Calculate the theoretical values of the voltages: Va, Vb, and Vab, without taking the meter resistance into account. Calculate the percent error between the theoretical and measured results.

2. Calculate the voltages: Va, Vb, and Vab taking the meter resistance into account. Calculate the percent error between the measured and calculated results.

Analysis, Part 2

1. Calculate the theoretical current, Ia, (without taking the meter resistance into account). Calculate the percent error between the theoretical and measured values (both current ranges).

2. Calculate the current, Ia, taking the meter resistance into account. Calculate the percent error between the calculated and measured results.

3. Briefly explain the significance of the error analysis in steps 1 and 2 above.

Experiment 5: PSpice Circuit Simulation

Introduction

This exercise will go through the steps of simulating the circuit of experiment 4. The software used is in two parts. “OrCAD Capture” is used to enter the schematic diagram. The circuit is then simulated using “OrCAD PSpice”. The schematic diagrams and simulation results may be copied and pasted into other applications.

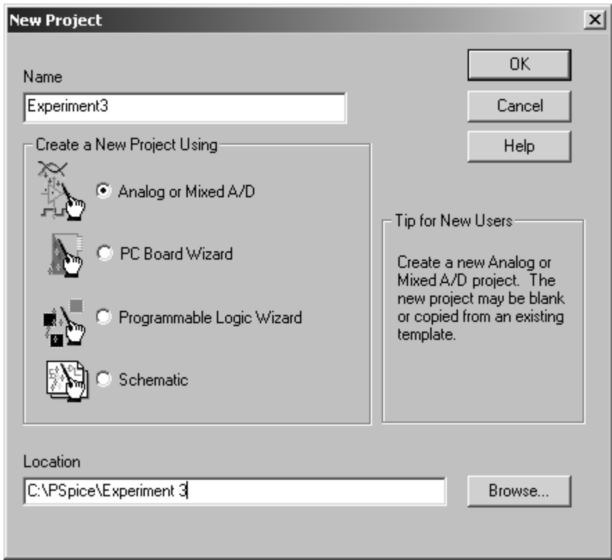
There may be some variation in the software interface depending on which version of PSpice is being used. “Help” is available within the program and online.

Drawing the Circuit

1. Start the “Capture” program. In the main menu bar, click on File. Select *New* and then *Project.* The [New Project] dialog box shown on the right will open.

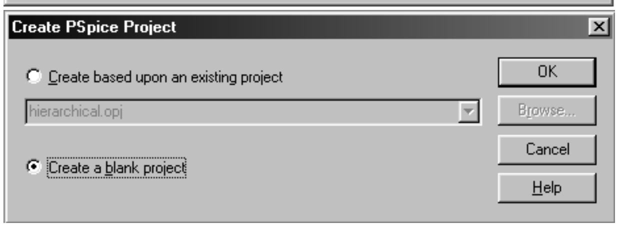
2. Create the new project using: *Analog or Mixed A/D*.

3. Name the project and specify a saving location, as shown. Save project to a folder with the name of the project. You can copy that folder to another disk if you wish to run the simulation on another computer. Click on *OK*.



4. Select *Create a blank project* in the next dialog box.

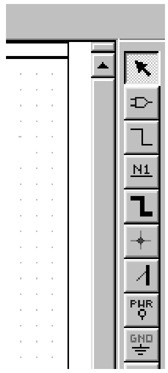
5. After clicking *OK*, you can start drawing your schematic diagram.



6. Note that some versions of “OrCAD Capture” startup slightly differently. You may get a dialog box asking you to select the libraries to use in your project. Be sure to select only the PSpice libraries.

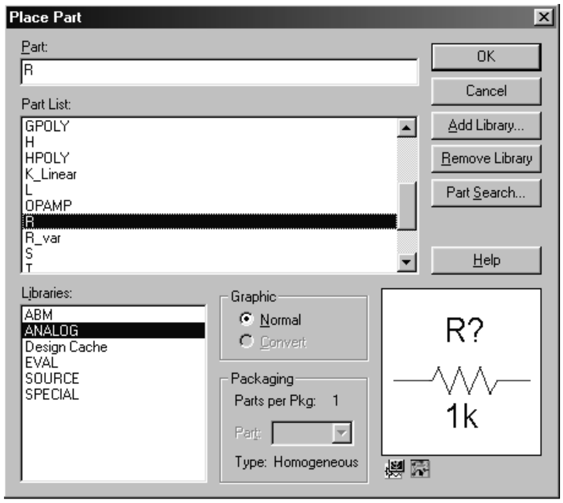
   After selecting the libraries you can start drawing your circuit. Click on the blank workspace to start.

7. You should see a column of buttons on the right side of the screen. The ones you will use most often are shown here on the right. The first button, which is shown selected, allows you to select parts in your drawing by clicking on them. The parts change color when selected.



8. The second button is used to get parts for your schematic diagram. The third button is used for drawing wires. The fourth button, N1, is used to label nodes in your circuit. The last button, GND, is used to place ground references in your diagram. These four buttons are the ones most often used in drawing a circuit schematic diagram.

9. Click on the parts button. In the [Place Part] dialog box, click on *ANALOG* and select "R" from the parts list, as shown on the right.



10. When you click on *OK* the dialog box will disappear, and you will be able to place the resistor into your diagram by using drag and drop. You can place multiple resistors into your diagram. The resistors are automatically numbered, starting with R1. The default value is 1k ohms.

11. Get four resistors, R1, R2, R3 and R4.

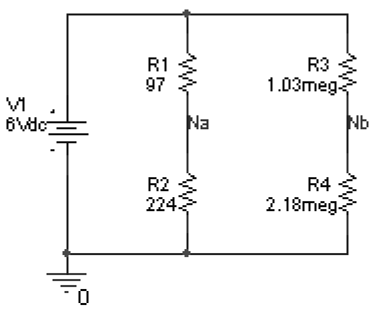
Place them into your diagram oriented vertically. The resistors and other parts may be rotated by clicking the right mouse button while the resistor is selected and clicking on *Rotate*.

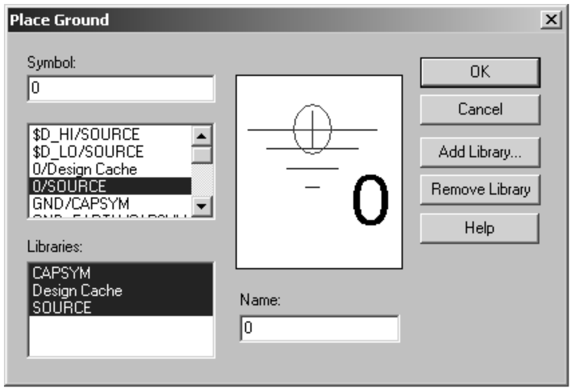
The part references (R1, R2, R3, and R4) may be changed by double clicking on the part references. The part values may be changed by double clicking on the part values. Any selected item may be moved by dragging and dropping.

12. Get the part "VDC" from the source library and change its value to 6Vdc. Change the values of the resistors to the values measured in experiment 3. In PSpice, $M = 10^{-3}$ *and* $Meg = 10^{6}$. Click on the "wiring" button on the right side to connect the parts with wires. You should experiment with moving the parts and connecting the wires.

Try to make the circuit look like the one below. The ground symbol is obtained by clicking on the GND button, selecting *SOURCE* and selecting "0" as shown on the next page. The circuit will not simulate without the ground.

Schematic Diagram

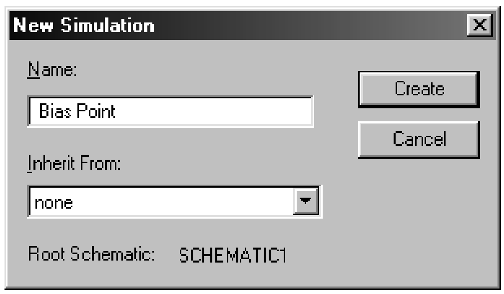




Simulating the Circuit

1. To simulate the circuit, click on PSpice and then on <u>New Simulation Profile</u>. You can also use the shortcut buttons. Note the menu bar below and note the button: <u>New Simulation Profile</u>. The next button is to <u>Edit Simulation Profile</u>. The third button (right arrow) is used to start the simulation.

2. The simulation was named "Bias Point" as shown on the right. After naming the simulation and clicking on *<u>Create</u>* you will get a [Simulation Settings] dialog box. "Analysis type" is set to *<u>Bias Point</u>*. Bias point analysis outputs the voltages at all of the nodes as well as the currents supplied by the power supplies.

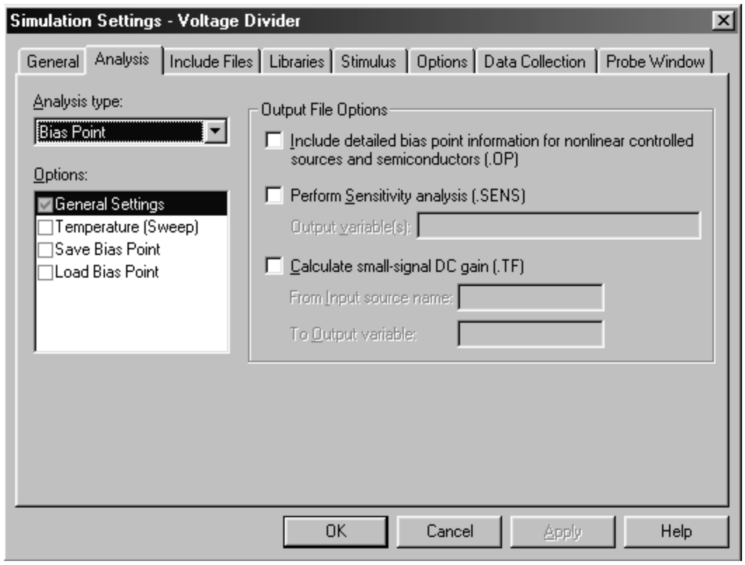


Results are available in an "Output File" which is generated for all simulations.

3. Refer to the [Simulation Settings] dialog box on the right.

4. Simulate the circuit by clicking the simulation button (arrow in main menu bar).

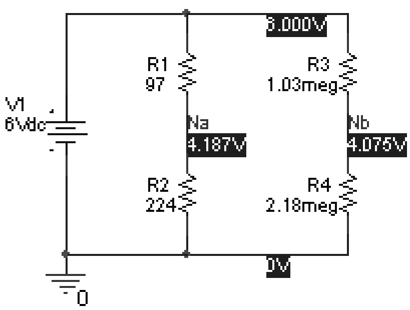
   Click on View and then on *Output File* in the window that opens. Scroll down until you get the voltages at the nodes, as shown below.

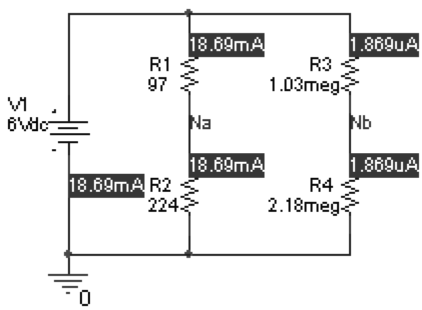


NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE
(NA) 4.1869 (NB) 4.0748 (N00149) 6.0000

VOLTAGE SOURCE CURRENTS
NAME CURRENT
V_V1 -1.869E-02

5. Voltages can be displayed on the schematic diagram by clicking on the V button in the main menu. Currents can displayed on the schematic diagram by clicking on the I button in the main menu. Note the results below.





6. Record simulation results below. Vab = Va – Vb.

Va _______________ Vb _______________ Vab _______________

Ia ________________ Ib ________________

7. Connect a 10 meg resistor in parallel with R4 to simulate the effect of the voltmeter. Run PSpice and record the result below.

   Vb ________________

8. Connect a resistor whose value equals the resistance of the milli-ammeter on the 20 mA range between R1 and R2 to simulate the effect of the milli-ammeter. Run PSpice and record the result below.

   Ia _________________

Analysis

Present your measured results from experiment 4 and your simulated results from this exercise in a spreadsheet table. Have the spreadsheet calculate the percent differences between the measured and simulated results.

Cadence OrCAD PSpice is a very powerful program with lots of features. One way to learn more about how to use it is to use the "Help" in the main menu. There are also many reference books published on Spice and OrCAD PSpice.

Experiment 6: LTspice Circuit Simulation

Introduction

LTspice IV is similar to OrCAD PSpice and is easy to use. It is free to download from Linear Technology: http://www.linear.com/designtools/software/

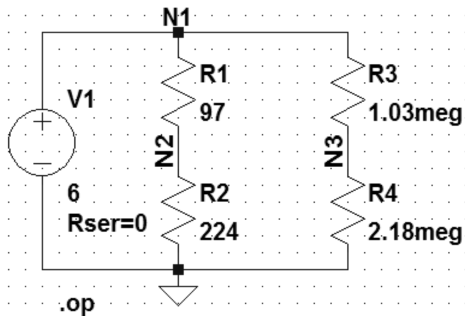
You can also download a manual and a "getting started guide". One advantage over the evaluation version of OrCAD PSpice is that there are no limits on schematic size or number of components used. It is also used at many schools and universities.

Drawing the Circuit

1. Start the *LTspice* program. In the main menu bar, click on File and select New Schematic.

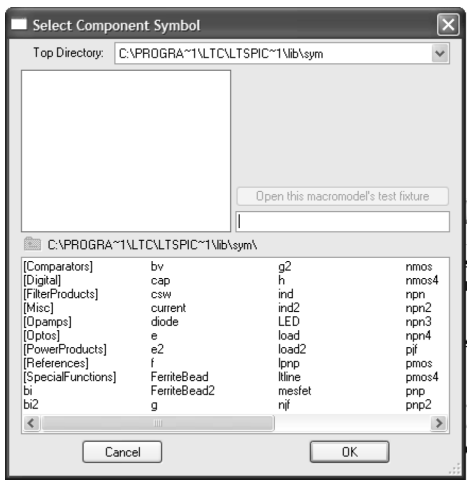
2. Create the new schematic. Left click on the "resistor" symbol and drag and place the resistor, R1. Get three more resistors and place them as shown on the right.

   Right click on the "*R*" of each resistor. Change the values to the measured values from Experiment 4.



3. Left click on the gate symbol between the diode and the hand in the main menu to get the dialog box on the right. Select the part "*voltage*" and place it in the schematic.

   Click on the ground symbol and place it under R2.



4. Left click on the pencil in the main menu to connect the components.

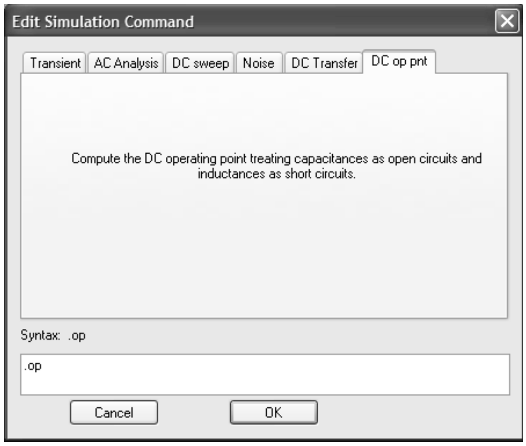
5. Left click on the *A* between the ground symbol and the resistor symbol to label the nodes *N1, N2,* and *N3* as shown on the schematic.

Click Help *in the menu for more information on creating the schematic.*

6. Click on Simulate and then select the *Edit Simulation Cmd* in the menu. Select *DC op pnt*. Click on *OK*. A "*.op*" command will appear which can be placed anywhere on the schematic.

7. Click on *Simulate* and select *Run*. If there are no errors, you will see an "*Operating Point*" file.

   The Spice netlist shows the connections of the parts, part models, and types of analysis to be performed.




```
       --- Operating Point ---
V(n1):          6              voltage
V(n2):          4.18692        voltage
V(n3):          4.07477        voltage
I(R4):          1.86916e-006   device_current
I(R3):          1.86916e-006   device_current
I(R2):          0.0186916      device_current
I(R1):          0.0186916      device_current
```

Netlist

```
R1 N1 N2 97
R2 N2 0 224
R3 N1 N3 1.03meg
R4 N3 0 2.18meg
V1 N1 0 6 Rser=0
.op
.backanno
.end
```

Exercise and Analysis

1. Simulate the loading effect of the voltmeter at nodes 2 and 3 using a resistor whose resistance is equal to the meter's internal resistance. Compare simulated results to your measured results in Experiment 4.

2. Simulate the loading effect of the ammeter using a resistor whose resistance is equal to the meter's internal resistance. Compare simulated results to your measured results in Experiment 4.

Experiment 11: Thevenin's Theorem and the Bridge Circuit

Introduction

Thevenin's theorem states that a circuit between two nodes, a and b, can be replaced by a single voltage source (Thevenin voltage, V_{TH}) in series with a single resistance (Thevenin resistance, R_{TH}). The Thevenin voltage is equal to the "open circuit" voltage between nodes a and b. The Thevenin resistance is equal to the resistance between nodes a and b.

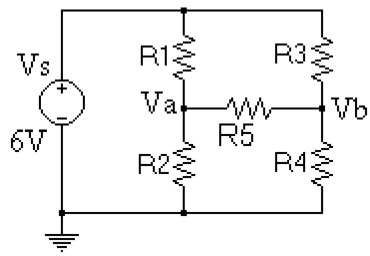
Norton's theorem states that a circuit between two nodes, a and b, can be replaced by a single current source (Norton current, I_N) in parallel with a single resistance (Norton resistance, R_N). The Norton current is equal to the "short circuit" current between nodes a and b. The Norton resistance is equal to the resistance between nodes a and b.

The application of Thevenin's or Norton's theorem may also require the application of other analysis methods. Determining an "open circuit" voltage or a "short circuit" current may require the use of node voltage or mesh current analysis. Having a Thevenin or Norton equivalent circuit as a model of a more complex circuit makes it much easier to determine how the circuit will perform when loads or sources are connected to it.

It is possible to find the Thevenin or Norton equivalent of a circuit by measurement, without knowing, necessarily, the components and connections of the original circuit. If we can measure the original circuit's open circuit voltage and short circuit current, we can then determine the circuit's Thevenin or Norton resistance: $R_{Th} = R_N = \frac{V_{Th}}{I_N}$.

Wheatstone Bridge

Wheatstone bridge circuits are used in many applications in electronics. A common problem is to find the voltage across the center element (R5 on the right) and the current through it. The node voltage method can be used to find the voltage. However, if the value of the center element is varied, it is easier to use the Thevenin equivalent circuit.



The Thevenin voltage of the circuit above "external" to R5 is found by determining the open circuit voltage, Va – Vb, with R5 removed.

The Thevenin resistance is found by determining the resistance across the terminals where R5 was removed, with the resistance of Vs equal to zero.

When R5 is removed, Va and Vb can be calculated with the voltage divider equation.

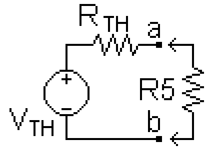
$$Va = \frac{R2}{R1 + R2} Vs \quad \text{and} \quad Vb = \frac{R4}{R3 + R4} Vs$$

The Thevenin voltage can be calculated as V_{Th} = Va – Vb.

It can be shown that the Thevenin resistance is:

$$R_{Th} = (R1 \parallel R2) + (R3 \parallel R4) = \frac{R1 \cdot R2}{R1 + R2} + \frac{R3 \cdot R4}{R3 + R4}$$

According to Thevenin's Theorem, the Thevenin equivalent circuit on the right will produce the same voltage across R5, for any value of R5, as the original bridge circuit.



Objectives

A comparison will be made between the solutions for the voltage across the resistor, R5, using the node voltage method and the Thevenin equivalent circuit method. The Thevenin equivalent circuit will be determined by measurement as well as by calculation. The value of the center element, R5, will be varied in both circuits to show that they are equivalent. The Thevenin equivalent circuit will also demonstrate the Maximum Power Transfer Theorem.

Procedure

Equipment and Parts
Power Supply, DMM, and Breadboard. R1 = 1500 R2 = 2200 R3 = 1000 R4 = 470 R5 values: 390, 680, 1200, 2200, 3900.

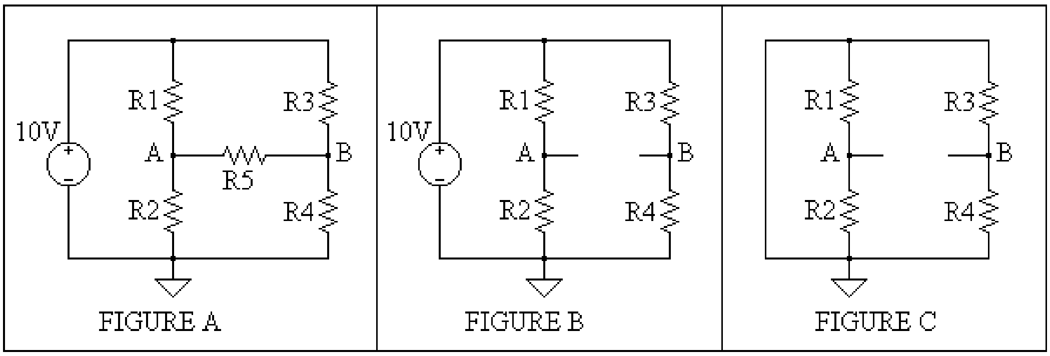
1. Measure and record the values of all the resistors you will be using.

   R1 ______________ R2 _______________ R3 ________________

   R4 ______________ $R5_{390}$ ______________ $R5_{680}$ ______________

   $R5_{1200}$ ____________ $R5_{2200}$ _____________ $R5_{3900}$ ______________

2. Set the DMM to read DC volts. Use the meter range which gives you at least three significant digits. Connect the circuit in figure A below. Use R5 = 390-ohms initially.



3. Set the power supply voltage to 10.0 V.

4. Measure and record the voltage V_{AB} in the table on the right for the given values of R5.

R5 (labeled)	Vab
390	
680	
1200	
2200	
3900	
∞ (open)	

5. Turn off the power supply and disconnect it from the circuit. Then connect a wire from the top of R1 to the bottom of R2, as shown in Figure C.

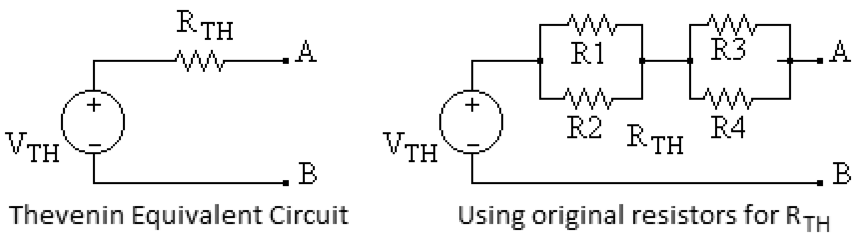
   Use the ohmmeter to measure the resistance of the circuit between nodes A and B. This resistance is the measured Thevenin resistance.

6. Record the Thevenin resistance, R_{TH}, from step 5 above. Record the voltage Vab with R5 = ∞ from step 4 as the Thevenin voltage, V_{TH}.

7. R_{TH} _______________ V_{TH} _________________

8. Use the values in step 7 above to connect a Thevenin equivalent of the bridge circuit for nodes A and B. Use a combination of resistors for R_{TH} to match the resistance recorded in step 7 to within one percent.

   The exact value may be obtained by connecting your bridge circuit resistors as shown below.



9. Set the power supply voltage to V_{TH}. Measure and record the voltage V_{AB} in the Thevenin equivalent circuit for the given values of R5.

R5	Vab
390	
680	
1200	
2200	
3900	

Analysis

1. Enter the data for the voltage V_{AB} into a spreadsheet as shown below:

	A	B	C	D	E
1	R5 (measured)	Vab (Step 4)	Vab (Step 9)	% Error	Power (Step 9)
2	390				
3	680				
4	1200				
5	2200				
6	3900				
7	∞ Open				

2. Use the measured values of R5 in column A. Use the equation for percent error in cell D2, and use "fill down" to have the spreadsheet calculate the error for cells D2 to D7.

   Similarly, enter the equation for power in cell E2 and use "fill down".
   % Error = ((2 - B2) / B2) *100   Power = C2*C2 / A2

3. Solve the circuit in Figure A for the voltage V_{AB}, for R5 equal to 1200-ohms, using the node voltage method.

4. Simulate the circuit in Figure A. Generate a list of values of Vab for all 5 of your measured values for R5. Use the spreadsheet to calculate the percent error between the simulated and measured results (measured values from step 5).

PSpice Example: Simulating a Bridge Circuit and Variable Resistor

In the circuit simulation below, the value of R5 was varied. Note that its value is indicated as "{RX}". Two methods of analysis will be used.

First the circuit will be simulated for 5 specific values of R5. Simulation output will be a list of voltages across R5, and a list of currents through R5, for each specific value of R5.

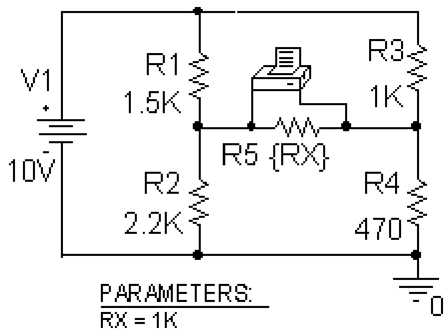
Second the circuit will be simulated by increasing the value of R5 from 100 Ω to 5000 Ω in 10 Ω steps. The output will be a graph generated by the "*Probe*" feature of *PSpice*.

Get the Parts

Get and place the resistors. Get the voltage source, *Vdc.* Set the voltage to 10 V. Get the printer, *VPRINT2*, and connect it across R5 as shown.

Get the part, *PARAM* . Place it in any convenient place. Get and place the analog ground (0/SOURCE).

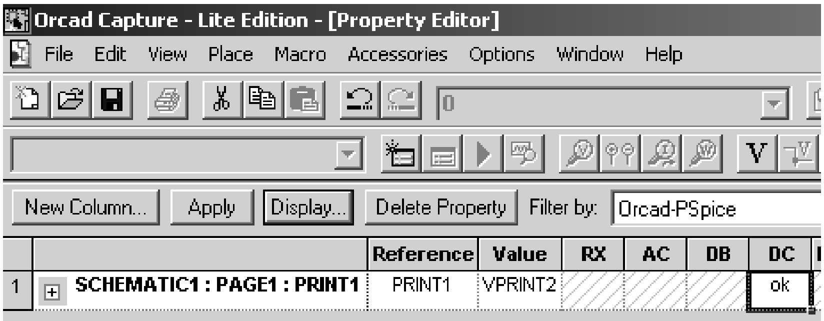
Note: *VPRINT2*, and *PARAM*, are in the "*Special*" library.



Setup the Analysis

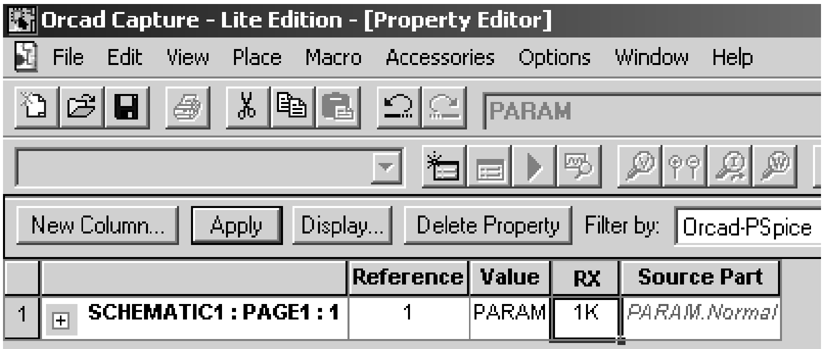
Double click on the printer, VPRINT2, select “DC” and type “ok” for value.

Save and exit. Note that “*Filter by: OrCAD-PSpice*” is selected.



Double click on “*PARAMETERS”* on the diagram. Set RX = 1K.

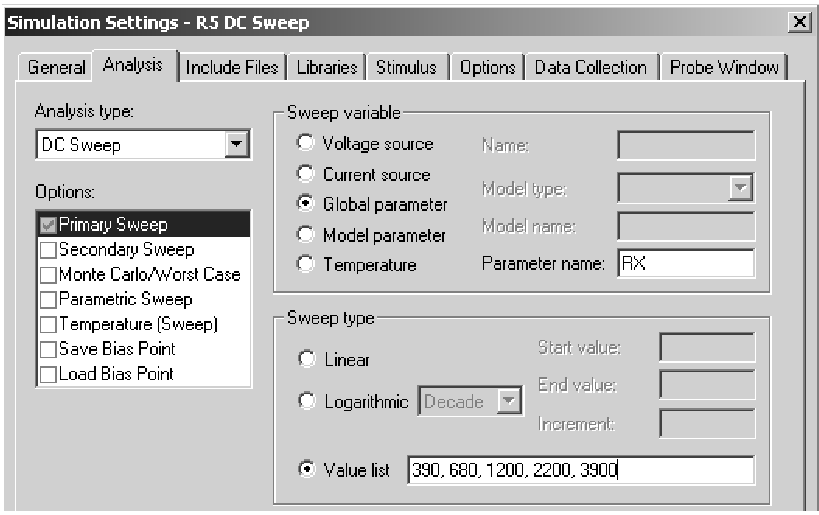
If the column “RX” does not appear, click on *New Column* and add it.



Click on *PSpice* in the menu bar. Select *New Simulation Profile*.

Name the profile in the dialog box that follows, such as: “R5 DC Sweep”.

Set up the simulation as shown in the dialog box on the right.



Simulate the Circuit

After the simulation runs, a “*Probe*” window opens without traces on it. Click on *View* in the menu and select *Output File***.**

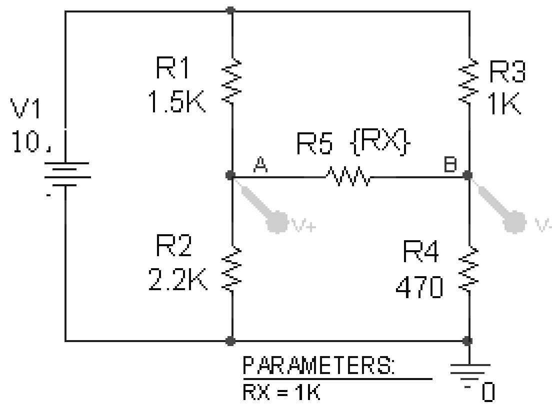
Scroll down to the “DC TRANSFER CURVES” and note the results. You can copy the results into another file, such as a word processor.

In addition to the simulation results, the output file contains the circuit's "netlist". The netlist specifies the parts, their values, and the nodes to which they are connected. It includes analysis information and part models.

To save paper, print only the necessary simulation results. These can be copied to a word processor document and edited. Simulation results output file on the right:

RX	V(N00297,N00239)
3.900E+02	6.693E-01
6.800E+02	9.881E-01
1.200E+03	1.368E+00
2.200E+03	1.772E+00
3.900E+03	2.097E+00

Graphical output with "Probe"



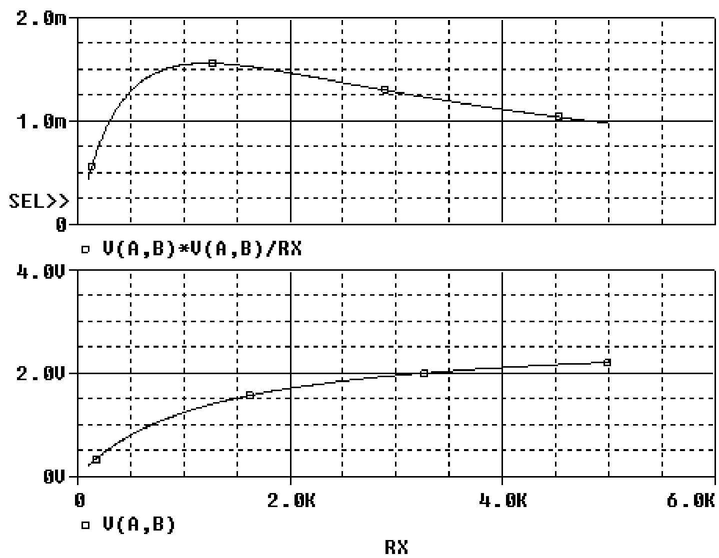
The printer was deleted in the circuit on the right. Nodes "A" and "B" were added using the "N!" button in the vertical menu bar.

"Markers" were added by clicking on PSpice in the main menu and selecting *Markers* followed by *Voltage Differential*. Place the first marker at node A and the second at node B. Short cut commands are also available.

The value of the resistor, R5, is varied from 100 Ω to 5000 Ω in steps of 10 Ω. Set up new simulation settings as follows:

Set "*Analysis type*" to DC SWEEP, and set the following:

Sweep Variable = Global Parameter, *Sweep Type* = Linear
Name = RX, *Start Value* = 100, *End Value* = 5000, *Increment* = 10



Simulate the Circuit: The probe window will open with a graph of V(A,B), the voltage at node A with respect to node B, as a function of the value of R5.

The top graph on the right is a plot of the power dissipation of R5. In the "*Probe Window*" menu, click on Plot and select *Add Plot to Window*. Then click on *Add Trace* and in the "*Trace Expression*" box, type: V(A,B)*V(A,B)/RX.

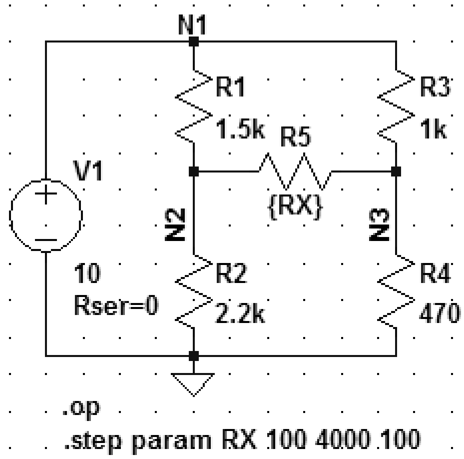
A cursor can be used to display the X and Y values on the graph. Click on Trace in the probe window menu bar, then on *Cursor*, then on *Display*. Clicking a mouse button on the symbol of a trace at the bottom of the graph will cause that mouse button to move the cursor over that trace and display the X and Y values in the "Cursor" window.

LTspice Simulation of a Bridge Circuit and Variable Resistor

In the circuit simulation below, the value of R5 was varied from 100 Ω to 4000 Ω in 100 steps. R5's value is indicated as "{RX}" on the schematic.

Get the Parts

Get and place the resistors. Get and place the voltage source and ground. Connect the circuit. Label nodes N1, N2, and N3.



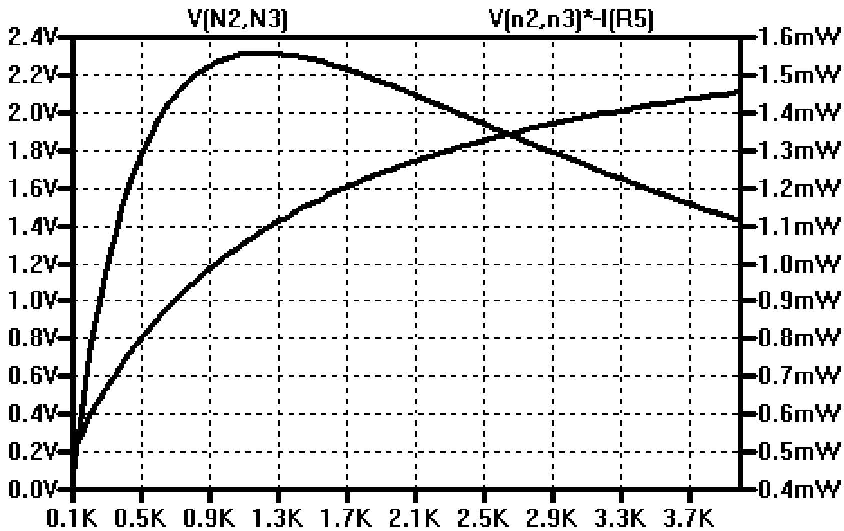
Setup the Analysis

Spice directives can be entered directly into the schematic by pressing the letter "s". Enter the directives "*.op*" and "*.step param RX 100 4000 100*" and place them in a convenient place on the schematic. Value of R5 is varied from 100 Ω to 4000 Ω in 100 Ω steps.

Simulate the Circuit

Move the mouse to N2, left click and drag the probe to N3. This will display the voltage across R5. To plot of the power dissipation of R5, click in the plot window, click on *Plot Settings* in the main menu, select *Add trace*, type V(n2,n3)*-I(R5), and click *ok*.

The plot on the right shows that maximum power occurs for a value of R5 of about 1200 ohms.

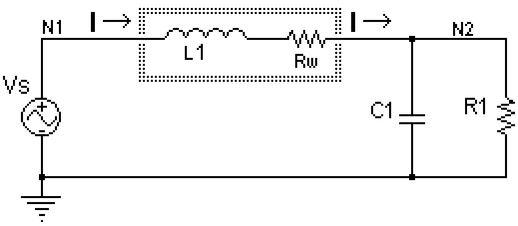


Experiment 20: Series-Parallel AC Circuit Measurements

Introduction

Refer to the circuit on the right. The impedance of this circuit is given by:

$\boldsymbol{Z}$ = (Rw + jωL1) + $\boldsymbol{Z}_P$, where: $\boldsymbol{Z}_P = \dfrac{-\boldsymbol{j}R1\left(1/\omega C1\right)}{R1-\boldsymbol{j}\left(1/\omega C1\right)}$.



Rw is the resistance of the inductor. The voltage at node N2 can be calculated using Ohm's Law. First calculate the circuit current, **I**: **I** = Vs/$\boldsymbol{Z}$.

Then calculate the voltage developed by that current across the impedance of the parallel combination of C1 and R1. $\mathbf{V}_{N2} = \mathbf{I}\cdot\boldsymbol{Z}_P$.

Objectives

A steady state sinusoidal voltage will be applied to a series-parallel connected RLC circuit. The amplitudes and phase angles of voltages and currents in the circuit will be measured. The measurements will be compared to theoretical calculations.

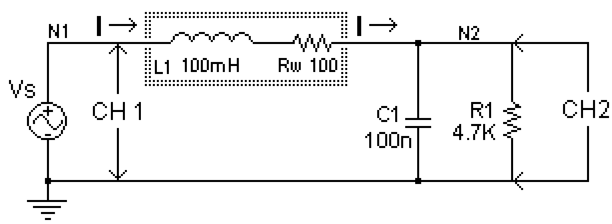
Procedure

Equipment and Parts
Function Generator, Oscilloscope with 10X probes, and Breadboard. Resistor: 4.7K, ¼ watt, 5%, Inductor: 100 mH, 5%, Capacitor: 0.1 µF, 5%.

1. Measure and record the values of the components, including the resistance of the inductor, Rw.

   R1 __________ Rw _________ L ___________ C ___________

2. Connect the circuit on the right. Connect oscilloscope channel 1 to N1 and channel 2 to N2. Set the trigger to channel 1.



3.. Set the generator to produce a 3.0 V peak-to-peak, 800 Hz, sine wave at node N1.

4. Measure and record the peak-to-peak magnitude and phase angle of the voltage at node N2 (ch2). Record your results into the data table on the next page.

Data Table

Freq. Hz	$\mathbf{V}_{N1}$ p-p	VP_{N1} deg.	$\mathbf{V}_{N2}$ p-p	VP_{N2} deg.	$\mathbf{I}_{N2}$ p-p	IP_{N2} deg.
800.00	3.00	0.00				
1600.00	3.00	0.00				
3200.00	3.00	0.00				

5. Set the function generator to produce a 3.0 V peak-to-peak, 1600 Hz, sine wave, as measured by channel 1 of the oscilloscope. Repeat step 4.

6. Set the function generator to produce a 3.0 V peak-to-peak, 3200 Hz, sine wave, as measured by channel 1 of the oscilloscope. Repeat step 4.

Analysis

1. Use the voltage measurements at node N2 to calculate the circuit current, **I**, at each frequency. Enter the results into the table above.

2. Use your scientific calculator to calculate inductive and capacitive reactance at each frequency. Then calculate the impedance of the circuit, **Z**, at each frequency, using the measured values of your components.

   TI-89 example at 1600 Hz:

   X_{L1}: 2*π*1600*.1 = 1005 ohms. X_{C1}: = 1/(2*π*1600*.1E-6) = 995 ohms.

   Z: 100+1005**i**+(-4700*995**i**)/(4700-995**i**) = (306.2$\angle$ 9.91).

3. Use Ohm's law to calculate the circuit current, **I**, at each frequency.

   I: 3/(306.2$\angle$ 9.91) = (9.80$\angle$-9.91).

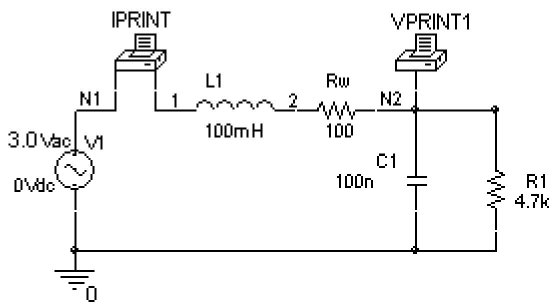
4. Simulate the circuit. Compare the simulated results to your calculations. They should agree exactly. Express the percent difference between your calculated and measured current and between your calculated and measured node 2 voltage.

PSpice Example

Connect the circuit as shown on the right.

The printers, "IPRINT" and "VPRINT1", are in the "SPECIAL" library.

Enable the printers by double clicking on each printer to open the property editor. Type "ok" under AC, MAG, and PHASE.



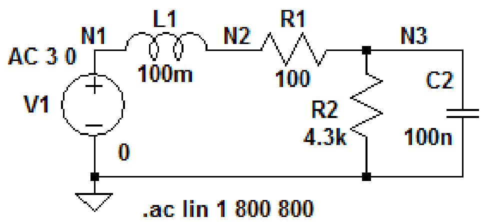
Simulation Settings: Analysis type: AC SWEEP/Noise, Sweep Type: Log/Octave, Start Frequency: 800, End frequency = 3200, Points/Octave: 1.

Output File Results:

FREQ	IM(V_PRINT2)	IP(V_PRINT2)
8.000E+02	2.088E-03	5.550E+01
1.600E+03	9.800E-03	-1.001E+01
3.200E+03	1.965E-03	-8.428E+01

FREQ	VM(N2)	VP(N2)
8.000E+02	3.824E+00	-1.156E+01
1.600E+03	9.536E+00	-8.806E+01
3.200E+03	9.722E-01	-1.682E+02

LTspice Example



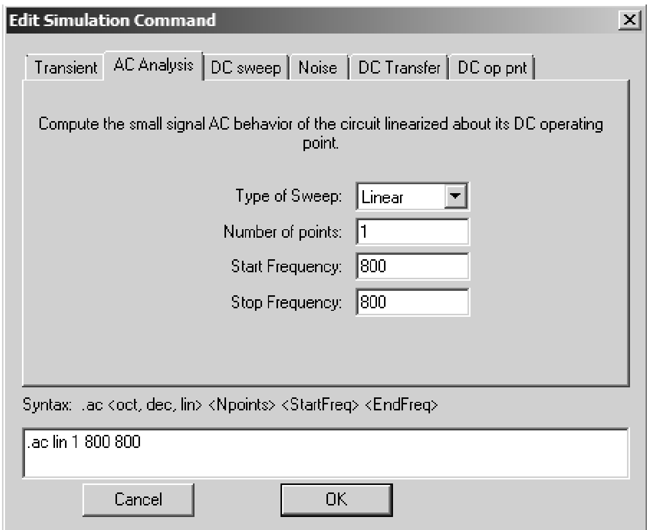
Connect the circuit on the right. Use your measured part values.

Right click on V1and set "AC amplitude" to 6 under "Small signal AC analysis" in the dialog box that opens.

Labeling the nodes (N1, N2, and N0) makes it easier to interpret the results.

Click on Simulate in the main menu.

Select *Edit Simulation Cmd* to open the dialog box shown on the right.



Select *AC Analysis* and enter values as shown. Note that this is for one frequency. If you specify more than one frequency the output of the simulation will be a graph instead of a list.

You will need to run the simulation three times, once for each frequency. Run the simulation. A window will open listing the simulation results.

```
       --- AC Analysis ---
frequency:     800            Hz
V(n3):         mag:    3.80461 phase:   -12.2394°        voltage
V(n1):         mag:          3 phase:          0°        voltage
I(V1):         mag: 0.00210717 phase:   -127.067°        device_current
```

The results for the voltages should agree with your expectations. The currents need to be interpreted according to LTspice conventions.

Experiment 21: RLC Steady State Sinusoidal Response

Introduction

When applied to an electric circuit connected to a sinusoidal source, the term "steady state" means that the sinusoidal source has been applied to the circuit for a long enough time so that the magnitudes and phase angles of the voltages and currents in the circuit have stopped changing with time.

Steady state conditions are usually reached within the first few cycles after the source is applied. When making measurements, the circuit voltages and currents will usually have reached steady state conditions way before the measurements can be made.

Objectives

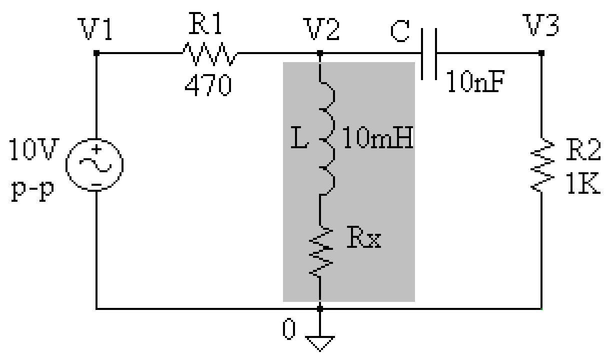
This exercise will demonstrate how to make AC voltage magnitude and phase angle measurements, and how to perform a simulation of the circuit. Circuit measurements will be made at 3 frequencies: 10 KHz, 20 KHz, and 30 KHz. The circuit's AC steady state response will be simulated at these frequencies. The measured and simulated results will be compared.

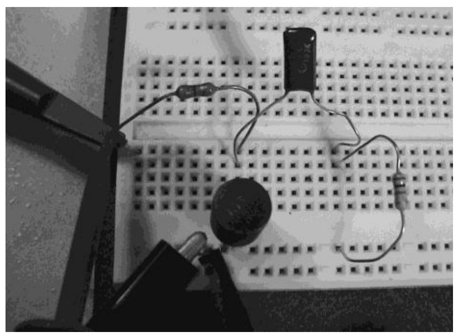
Procedure

Equipment and Parts

Oscilloscope, Function Generator, Breadboard.
Resistors: 470 Ω, 5%, 1 KΩ, all ¼ watt, 5%
Capacitor: 10 nF, 5%. Inductor: 10 mH, 5% (air core preferred).

1. Connect the circuit below (Rx is the resistance of the inductor). Layout the circuit on the breadboard like the diagram, as in the picture below.



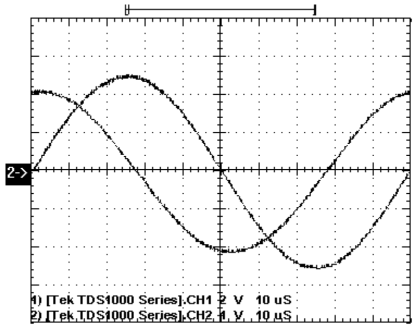


2. Connect channel 1 of the oscilloscope to measure V1. Set the function generator to produce a 10 volt peak-to-peak sine wave with no offset at exactly 10.00 KHz. Use the oscilloscope to set the amplitude.

The oscilloscope should be set as follows:

VERTICAL: Channel 1 Volts/DIV = 2 Coupling = AC TRIGGER: Channel 1 Mode = Auto Coupling = AC HORIZONTAL: Time/DIV. = 10 µs/div.

3. Connect channel 2 of the oscilloscope to measure V3. Adjust the trigger and centering controls to obtain a display similar the one shown on the right.



Adjust the trigger controls so that the sinusoid displayed on channel 1 starts on the left side of the screen at zero amplitude.

Both wave forms are centered.

Set the vertical coupling to GND on both channels to center the traces. Observe the phase difference between channels.

4. One cycle of the sinusoid on channel 1 should begin at 0 volts and have a positive slope on the left side of the screen, and also 0 volts and a positive slope at the right side of the screen. Since one cycle is 360^0, and there are 10 divisions across the screen, each division represents 36^0

The display above shows the channel 2 sinusoid crossing the zero volt axis with a negative slope 2.2 horizontal divisions before the channel 1 reference sinusoid. Therefore channel 2 leads channel 1 by about 2.2 times 36^0, or about 79^0.

Another method could be used which can be done with more accuracy. Measure the period of the reference sinusoid. Then measure the time difference between the zero axis crossings of both sinusoids, t_Z (be sure that they both have the same slope at the zero crossings). The phase angle can then be expressed as:

$$\theta = \frac{t_Z}{T} 360^0$$

The TIME/DIVISION controls can be used to measure t_Z more accurately. If the frequency is known accurately, the period T can be calculated: $T = 1/f$.

5. Measure and record the magnitudes and phase angles of the voltages, V2 and V3, at the frequencies of 10 KHz, 20 KHz, and 30 KHz.

Frequency	V2 Magnitude	V2 Angle	V3 Magnitude	V3 Angle
10k				
20k				
30k				

Use the VOLTS/DIVISION control on channel 2 to get the largest vertical displacement of the waveform. This increases the accuracy of both the magnitude and phase measurements. Also, make sure that both waveforms are centered vertically.

6. You should have 3 sets of voltage and phase measurements for V2 and V3, one set for each frequency. Enter your data into a spreadsheet, formatted as shown below. Fill in the rest of the spreadsheet after you do the simulation.

Frequency	Measured V2 p-p		Simulated V2 p-p		Percent Error, V2	
KiloHertz	Magnitude	Phase	Magnitude	Phase	Magnitude	Phase
10						
20						
30						
Frequency	Measured V3 p-p		Simulated V3 p-p		Percent Error, V3	
KiloHertz	Magnitude	Phase	Magnitude	Phase	Magnitude	Phase
10						
20						
30						

Analysis

1. Use *PSpice* to simulate your circuit at the frequencies of 10 KHz, 20 KHz, and 30 KHz. Refer to the example simulation on the next page.

2. Use the spreadsheet to compare the measured data to the *PSpice* simulation.

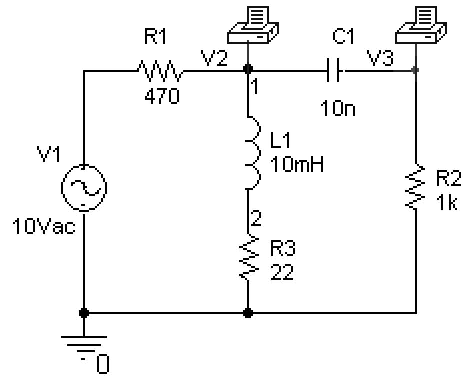
3. Discuss the accuracy of your results. It is usually harder to measure the actual values of the inductor and capacitor you used in this experiment. There are instruments that can measure capacitance accurately, but the problem with inductance measurements is that the inductance can vary with frequency and inductor current. To reduce the uncertainty in inductance, use an air core inductor with a 5% or better accuracy.

   Given all the variables, your measured results should be within about 10% of your simulated results.

4. Optional: Use the “node voltage” method and *Maple* to calculate the values of V2 and V3 at the frequency of 10 KHz. Your calculated results should agree with the results of the simulation.

PSpice Example: RLC Circuit Phasor Analysis

The circuit below is simulated at three frequencies by the “AC SWEEP” feature of *PSpice*. The magnitudes and phase angles of the voltages at the circuit nodes, V2 and V3, are measured by the printers for each frequency.



Printers in the schematic read the voltages with respect to ground, and list their values in the “Output File”. Phase angles are with respect to the source, V1.

The part name for the printers is: “VPRINT1”. Double click on each printer and set the following: AC = ok, MAG = ok, and PHASE = ok.

V1 is a source whose part name is: “Vac”. Set V1 to: ACMAG = 10V, ACPHASE = 0, DC = 0.

Set the Simulation Settings: Analysis type = AC SWEEP/ NOISE. AC Sweep Type = Linear, Total Points = 3, Start Frequency = 10k, End Frequency = 30k.

Run the simulation. When done, click on “View” in the *Probe* menu bar and select “Output File”.

Analysis Results:

**** AC ANALYSIS TEMPERATURE = 27.000 DEG C

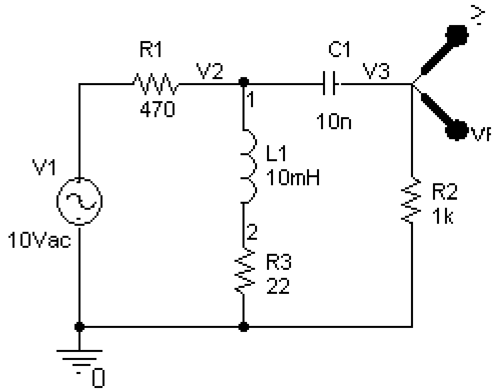
FREQ	VM(V2)	VP(V2)
1.000E+04	7.832E+00	2.479E+01
2.000E+04	7.678E+00	6.388E+00
3.000E+04	7.295E+00	2.288E+00

FREQ	VM(V3)	VP(V3)
1.000E+04	4.167E+00	8.265E+01
2.000E+04	6.008E+00	4.490E+01
3.000E+04	6.444E+00	3.023E+01

Using Probe to Plot the Frequency Response

A plot of the voltage and phase response at any node may be obtained by increasing the number of points in the frequency range. "Markers" at the nodes cause Probe to plot the response. The "Voltage Level" marker plots the magnitude of the voltage and the "Phase of Voltage" marker plots the phase angle.

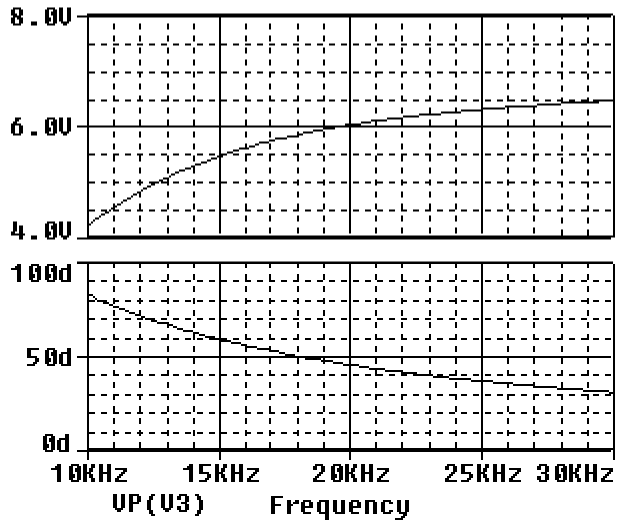
Note the "V" and "VP" markers on node V3 in the diagram on the right.



Set the Simulation Settings:
Analysis type = AC SWEEP/ NOISE.
AC Sweep Type = Linear.
Total Points = 100.
Start Frequency = 10k.
End Frequency = 30k.

Analysis Results:

The Probe output is shown on the right.



Voltage magnitude of V3 starts at about 4.2 volts at 10 KHz and increases to about 6.5 volts at 30 KHz.

Phase angle of V3 starts at about 82 degrees at 10 Khz and decreases to about 31 degrees at 30 Khz.

Cursors could be used to read the voltage magnitude and phase angle more accurately.

Experiment 22: Capacitive Voltage Dividers

Introduction

Voltage dividers may incorporate inductors and capacitors as well as resistors. Since inductive and capacitive reactance are frequency dependent, the voltage division may also be frequency dependent. In addition, there may be a frequency dependent phase shift. Capacitive voltage dividers are widely used in communications circuits.

We will investigate the characteristics of a capacitive voltage divider. The circuit in Figure A below consists of a resistive divider, R1 and R2, and a capacitive divider, C1 and C2. The output voltages of these dividers may be written as:

$$V_{N1} = \frac{R2}{R1+R2} 6V\ p-p.$$

$$V_{N2} = \frac{-j\frac{1}{\omega C2}}{-j\frac{1}{\omega C1} - j\frac{1}{\omega C2}} 6V\ p-p = \frac{C1}{C1+C2} 6V\ p-p.$$

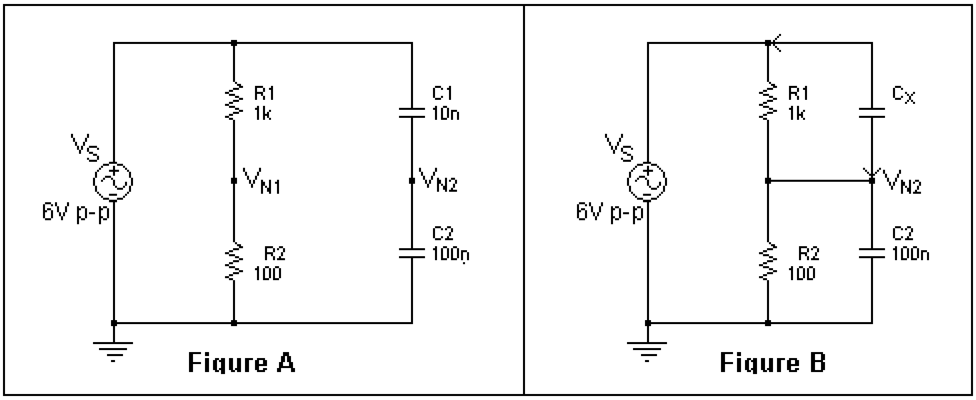


Figure A   Figure B

The voltage division is frequency independent. If both dividers divide by the same ratio, that is, $V_{N1} = V_{N2}$, then node N1 may be connected to node N2 without changing the voltage division ratio. This principle is used to “frequency compensate” oscilloscope probes.

An oscilloscope’s input can be approximated as a resistance in parallel with a capacitance. The input impedance for a Tektronix TDS1002 is: 1 MΩ, ±2% in parallel with 20 pF, ±3pF.

In Figure B above, R2 and C2 could represent the oscilloscope input, and R1 and C_X could represent the probe.

Objectives

The characteristics of capacitive voltage dividers, and capacitive voltage dividers connected in parallel to resistive voltage dividers will be measured. The results will be compared to theoretical calculations.

Procedure

Equipment and Parts
Oscilloscope, Function Generator, and Breadboard. Resistors: 100 Ω, 1 KΩ, all ¼ watt, 5%. Capacitors: 10 nF, 20 nF (or 22 nF), 100 nF, all 5%.

1. Measure and record the values of your resistors and (if possible) capacitors.

   R1 ________________ R2 ________________ C2 ______________

   C_X(10nF)___________ C_X(20nF)___________

2. Connect the circuit in Figure A on the previous page. Connect oscilloscope channel 1 to measure V_S, and channel 2 to node N1 to measure V_{N1}.

3. Set the function generator to produce a 6.00 V, peak-to-peak, sine wave with no offset, at a frequency of exactly 8.00 KHz. Use the oscilloscope to set the amplitude. This is the setting of the function generator for steps 4 through 7.

4. Measure and record the amplitude and phase angle of the voltage at node N1 with respect to V_S. Move the oscilloscope channel 2 to node N2. Measure and record the amplitude and phase angle of the voltage at node N2 with respect to V_S.

   V_{N1} ______________ volts p-p θ_{N1} _________________ degrees

   V_{N2} ______________ volts p-p θ_{N2} _________________ degrees

5. Connect the circuit in Figure B on the previous page without C_X. Connect the oscilloscope channel 1 to measure V_S (6.00 V peak-to-peak). Connect the oscilloscope channel 2 to node N2 to measure V_{N2}.

   V_{N2} _______________ volts p-p θ_{N2} _________________ degrees

6. Connect the circuit in Figure B with C_X = 10 nF. Connect the oscilloscope channel 2 to node N2 to measure V_{N2}.

   $V_{N2(10nF)}$ _____________volts p-p $\theta_{N2(10nF)}$ _______________ degrees

7. Connect the circuit in Figure B with C_X = 20 nF. Connect the oscilloscope channel 2 to node N2 to measure V_{N2}.

   $V_{N2(20nF)}$ ______________ volts p-p $\theta_{N2(20nF)}$ _________________ degrees

8. Connect the circuit in Figure B without C_X. Connect the oscilloscope channel 1 to observe V_S. Set the function generator to produce a 6.00 V, peak-to-peak, square wave with no offset, at a frequency of exactly 8.00 KHz. Use the oscilloscope to set the amplitude. This is the setting of the function generator for steps 9 through 11.

9. Connect channel 2 to node N2 to observe V_{N2}. Sketch or capture one cycle of the waveform on channel 2.

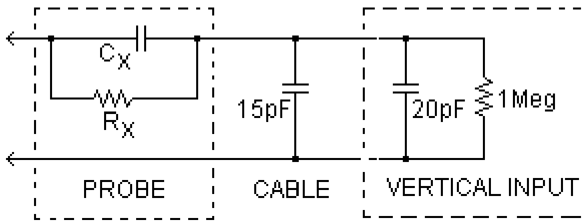
10. Connect the circuit in Figure B with C_X = 10 nF. Connect the oscilloscope channel 2 to node N2 to observe V_{N2}. Sketch or capture one cycle of the waveform on channel 2.

11. Connect the circuit in Figure B with C_X = 20 nF. Connect the oscilloscope channel 2 to node N2 to observe V_{N2}. Sketch or capture one cycle of the waveform on channel 2.

Analysis

1. Calculate the theoretical results for procedure steps 5 through 7 and compare the calculations to your measured results.

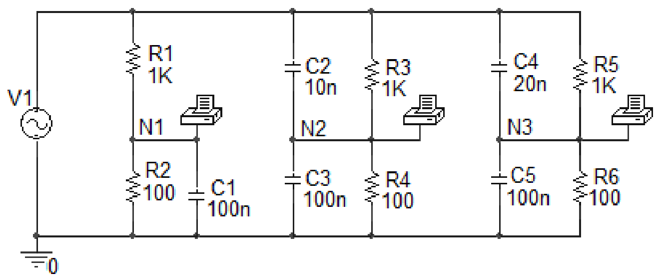
2. Use the specifications for the vertical input impedance of a Tektronix TDS1002 to design a compensated 10X probe (÷10). The cable connecting the probe to the vertical input adds 15 pF to the parallel capacitance. Calculate C_X and R_X in the model on right.



3. Optional: Compare your results in procedure steps 4 through 7 to a PSpice simulation. Use printers to obtain the magnitudes and phase angles of the voltages. See the example below.

All three situations are simulated simultaneously.

N1 is uncompensated.
N2 is compensated.
N3 is over compensated.



RESULTS AC ANALYSIS (From output file)	FREQ	VM(N1)	VP(N1)
	8.000E+03	4.961E-01	-2.456E+01
	FREQ	VM(N2)	VP(N2)
	8.000E+03	5.455E-01	-4.696E-15
	FREQ	VM(N3)	VP(N3)
	8.000E+03	6.782E-01	1.641E+01

4. Compare your results in procedure steps 9 through 11 to a PSpice simulation. Use Probe to obtain the response for each case.

[Replace the printers in step 3 above with voltage probes. Replace the AC source with a square wave source (Vpulse). Use transient analysis in the simulation settings.]

Experiment 30: Three Phase Power / Delta Connection

Introduction

This lab exercise uses the "Phase Tripler" circuit described in appendix 2 in this manual. It is a 3-phase wye source with an ACB phase sequence. The neutral connection of this source shares the same ground as the lab power supply, function generator, and oscilloscope. Therefore care must be taken to not ground a phase output, such as would occur if the ground lead of the oscilloscope were connected to one of the phase outputs.

Objectives

This exercise involves the measurement of the magnitude and phase of voltages and currents in a 3-wire, 3-phase, delta connected power system. The effect of balanced and unbalanced loads is measured.

Procedure

Equipment and Parts

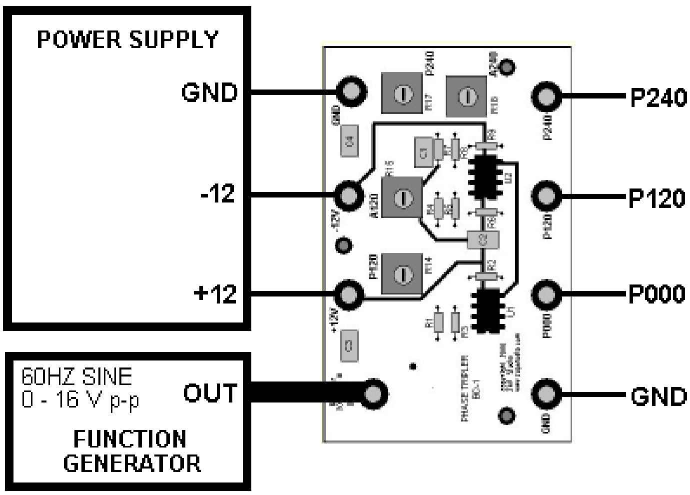
Function Generator, Power Supply, Phase Tripler Board, Oscilloscope, and Breadboard.
Resistors: 220-ohm, Three 390-ohm, ¼ watt, 5%.
Inductors: Three 100mH, 5% (iron core, 50mA minimum).

Don"t measure the values of the components. Use components with a 5% or better accuracy.

Before starting this exercise, read the "Phase Tripler Circuit Information" in appendix 2 of this manual. If you don't have the phase tripler board, one could be built on a breadboard using the schematic diagram and calibration procedure provided. The Phase Tripler printed circuit board is available at: www.zapstudio.com.

1. Connect the phase tripler board to the power supplies and function generator as shown on the right. Set the power supply to +12VDC and –12VDC. Set the generator to produce a 60 Hz, 12 V peak-to-peak, sine wave.

2. Connect channel 1 of the oscilloscope to P000 and set trigger to channel 1. Set the function generator amplitude to get exactly 12 V peak-to-peak on channel 1.



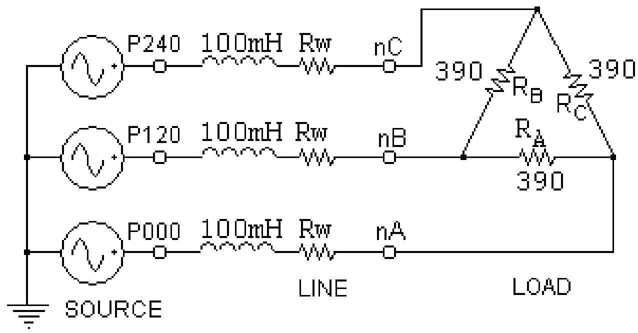
3. Connect channel 2 to P120. Check that the amplitude is exactly 12 V peak-to-peak and that the phase is leading channel 1 by exactly 120^0.

4. Connect channel 2 to P240. Check that the amplitude is exactly 12 V peak-to-peak and that the phase is lagging channel 1 by exactly 120^0.

   [Refer to the calibration procedure in Appendix 2 if steps 3 and 4 don't check.]

5. Connect the phase tripler source to the load as shown on the right.

6. Connect the oscilloscope to P000. The ground must be connected to the phase tripler ground and never anywhere else for this entire lab. Channel 1 will be the zero degree phase reference for measurements made with channel 2.



7. Measure the magnitude and phase angle of the voltages at nodes nA, nB, and nC with the oscilloscope channel 2 and record results in the table below (balanced case).

Step 7 node	nA	nB	nC
Magnitude p-p			
Phase, Deg.			

8. Replace the 390 Ω resistor R_C with a 220 Ω resistor.

9. Measure the magnitude and phase angle of the voltages at nodes nA, nB, and nC with the oscilloscope channel 2 and record results in the table below (unbalanced case).

Step 9 node	nA	nB	nC
Magnitude p-p			
Phase, Deg.			

Analysis

1. Use the node voltage method to calculate the voltages at nodes nA, nB, and nC for the balanced circuit. Save the equation on your calculator for the next step.

2. Use the node voltage method to calculate the voltages at nodes nA, nB, and nC for the unbalanced load circuit (edit your equation from step 1). Refer to the TI-89 example on the next page.

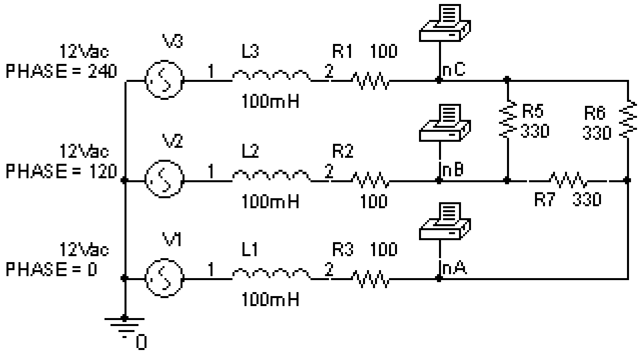
3. Calculate the percent difference between the measured and calculated values of the voltages at nodes nA, nB, and nC for the balanced and unbalanced loads.

4. Calculate the voltage across each load resistor using the measured results for the voltages at nA, nB, and nC. [($V_{nA} - V_{nB}$), ($V_{nB} - V_{nC}$), and ($V_{nC} - V_{nA}$)]

PSpice Example

Results for the circuit on the right.

FREQ	VM(NC)	VP(NC)
60	6.187	-130.2
FREQ	VM(NB)	VP(NB)
60	6.187	109.8
FREQ	VM(NA)	VP(NA)
60	6.187	-10.18



TI-89 Example

Let **V**(nA) = x, **V**(nB) = y, **V**(nC) = z.

Equations:

$$\frac{x-12\angle 0}{100+37.7i}+\frac{x-y}{330}+\frac{x-z}{330}=0 \text{ and } \frac{y-12\angle 120}{100+37.7i}+\frac{y-x}{330}+\frac{y-z}{330}=0 \text{ and } \frac{z-12\angle 240}{100+37.7i}+\frac{z-x}{330}+\frac{z-y}{330}=0$$

TI-89 input:

Balanced case:

csolve((x-12)/(100+37.7i)+(x-y)/330+(x-z)/330=0 and (y-(12∠120))/(100+37.7i)+(y-x)/330+(y-z)/330=0 and (z-(12∠240))/(100+37.7i)+(z-x)/330+(z-y)/330=0,{x,y,z})

x = (6.187∠-10.8) y = (6.187∠109.8) z = (6.187∠-130.2)

Unbalanced case:

csolve((x-12)/(100+37.7i)+(x-y)/330+(x-z)/220=0 and (y-(12∠120))/(100+37.7i)+(y-x)/330+(y-z)/330=0 and (z-(12∠240))/(100+37.7i)+(z-x)/220+(z-y)/330=0,{x,y,z})

x = (5.611∠-15.12) y = (6.187∠109.82) z = (5.477∠-127.29)

Experiment 31: Series RLC Circuit Step Response

Introduction

A step response is generated when the input to a circuit changes instantaneously from one value to another value. A function generator will be used as a step source by setting it to produce a 5 V peak-to-peak square wave with an offset of 2.5 V. The period of the square wave is set so that the circuit returns to steady state before the next step occurs.

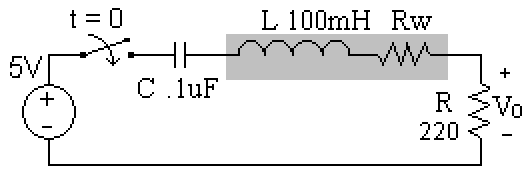
The circuit will exhibit a “forced” step response when the generator voltage transitions from 0 to 5 V. During this time energy is being forced into the circuit. The circuit will reach steady state when the capacitor charges to 5 V and the circuit current goes to zero. When the generator voltage transitions from 5 V to 0 V the circuit will exhibit a “natural” response. During this time the circuit’s stored energy is dissipated.

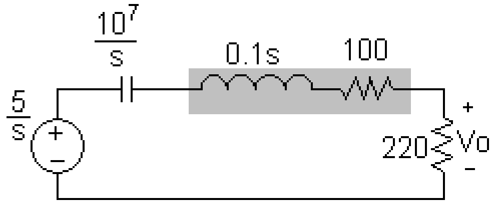
Forced Step Response

The step response of a circuit is observed when the voltage applied to it changes instantly from one value to another.

A 5 V DC source is applied to the circuit on the right. When the switch closes at t = 0 the forced step response is observed as Vo across the 220 Ω resistor.

An s-domain version is shown on the right.





$$\frac{Vo}{Vs}=\frac{R}{R+Rw+Ls+\frac{1}{Cs}}=\frac{\left(\frac{R}{L}\right)s}{s^2+\left(\frac{R+Rw}{L}\right)s+\frac{1}{LC}}=\frac{2200\,s}{s^2+\left(3200\right)s+1\times10^8}$$

$$Vo=\frac{2200\,s}{s^2+\left(3200\right)s+1\times10^8}\left(\frac{5}{s}\right)=\frac{11000}{(s+1600-\boldsymbol{j}9871)(s+1600+\boldsymbol{j}9871)}$$

.

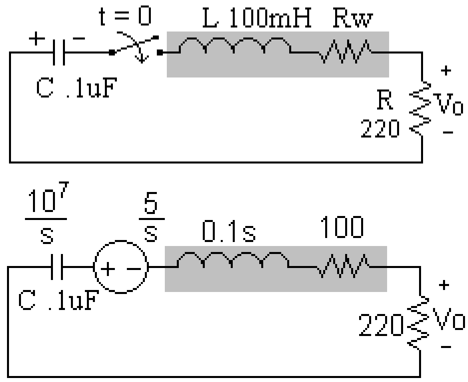
The time domain result is: $v_0(t)=1.114e^{-1600t}\sin(9871t)$

This circuit exhibits an under-damped forced step response. Its damped frequency is 9871 radians per second and decay rate is 1600 nepers per second.

Natural Response

The natural response of a circuit is observed when a circuit dissipates its stored energy. For example, a capacitor charged to 5 V is discharged at t = 0. The resulting response is observed as the voltage Vo across the 220 Ω resistor.

Time domain and s-domain versions of the circuit are shown on the right. Solving for Vo in the s-domain results in:



$$Vo = \frac{2200s}{s^2 + (3200)s + 1\times10^8}\left(\frac{-5}{s}\right) = \frac{-11000}{(s+1600-\boldsymbol{j}9871)(s+1600+\boldsymbol{j}9871)}$$

.

The time domain result is: $v_0(t) = -1.114e^{-1600t}\sin(9871t)$

This circuit exhibits an under-damped natural step response. Its damped frequency is 9871 radians per second and decay rate is 1600 nepers per second. The response is identical to the forced step response except for the negative sign.

This is the result obtained if a 5 V square wave with a 2.5 V offset were applied to the circuit with a long enough period so that the circuit returns to steady state during each half cycle.

The forced and natural responses are identical except that the natural response is 180 degrees out of phase with the forced response (negative sign).

Objectives

This lab exercise will demonstrate the production, measurement, and evaluation of the step response of a series RLC circuit.

Procedure

Equipment and Parts
Function Generator, Oscilloscope, Breadboard. L = 100 mH, C = .1 uF capacitor, R = 220 Ω, R = 2200 Ω.

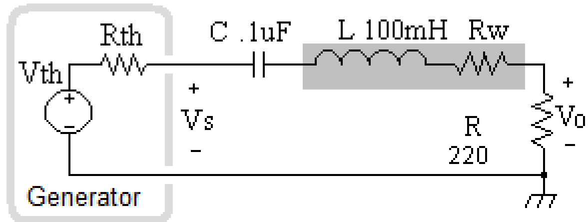
1. Measure and record the resistance of the resistor, R, and inductor, Rw. If possible, measure the inductance of the inductor and capacitance of the capacitor.

   R_{220} ___________ R_{2200} ____________ Rw _____________

   L ______________ C ______________

Connect C, L, and R directly together, with no wires in between.

Connect the function generator and oscilloscope leads directly to the parts.



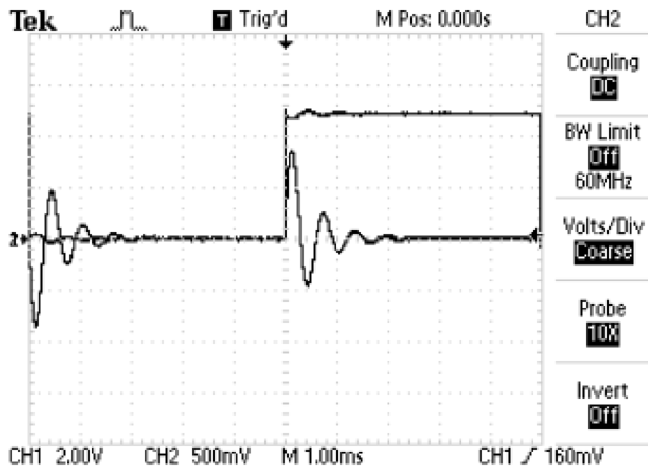
2. Set the function generator to produce a 100 Hz, 5 V peak-to-peak square wave, with a 2.5 V offset, when it is <u>not connected</u> to the circuit.

3. Use R equal to 220 Ω. Connect the function generator to the circuit. Channel 1 of the scope monitors the circuit's input, Vs, and channel 2 monitors output, Vo.

4. Set the time base to 1 mS per division. Trigger on channel 1, positive slope. The display should be similar to that on the right.

5. Set channel 1 to 2 V/DIV and channel 2 to 500 mV/DIV.

   Observe the relationship of the square wave to the damped sinusoidal waveform. Set the oscilloscope to DC input coupling and center both channels. Also note the oscillation on the channel 1 waveform.



When the generator voltage changes from 0 to 5 V, a forced step response is observed. Note that the oscillation begins on the positive half cycle and decays exponentially. When the generator voltage changes from 5 V to 0 V, a natural response is observed.

Note the similarities and differences between the natural and the forced response (compare the amplitude and phase of the forced response and the natural response).

6. Change the oscilloscope's time base to 200 or 250 µS/DIV. With the trigger on channel 1, negative slope, display only channel 2. The entire natural response should be observed. You can further expand the waveform to measure the period more accurately

7. Determine and record the period of the damped oscillation, T_d, by measuring the time between zero crossings of the same slope.

   T_d ___________________

Record the magnitude and time of occurrence of the first two positive peaks. Solve the exponential equations simultaneously (equations for v1 and v2 are given below) for the peak value, Vp, and decay rate, α. Organize your data into a table such as the one below:

Positive Peaks	220 ohm resistor	
	Time	Voltage
1st peak		
2nd peak		

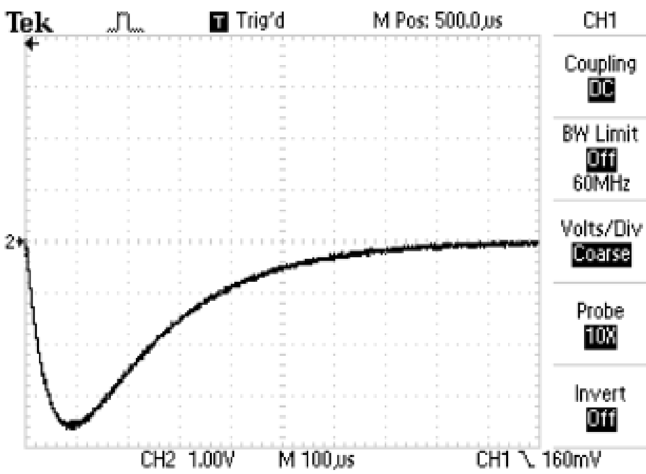
$$v1 = (Vp)\,e^{-\alpha(t1)}, \qquad v2 = (Vp)\,e^{-\alpha(t2)}$$

8. Repeat the procedure using a 2200 Ω resistor for R. Set up the oscilloscope to get a display of both the forced and natural response (response should be over-damped).

9. Adjust the oscilloscope to observe the natural response and expand the display to measure and record the following as accurately as possible:

a) Amplitude and time of occurrence of the negative peak, T_P.

b) Time when the amplitude decays to 20% of its the maximum value $T_{20\%}$.



V_P ________________________ T_P ____________________

$V_{20\%}$ ______________________ $T_{20\%}$ ___________________

Analysis

1. Calculate the theoretical response of the circuit using your measured part values for R equal to 220 Ω and for R equal to 2200 Ω.

   Explain why the internal resistance of the function generator must be included in the calculations.

2. Compare the natural response results of the experiment with the 220 Ω resistor to theoretical calculations.

   Indicate response type. Compare the decay rate, α, and damped oscillation frequency, ω_d, by calculating the percent difference between the measurements and the theoretical calculations.

3. Compare the natural response results of the experiment with the 2200 Ω resistor to theoretical calculations.

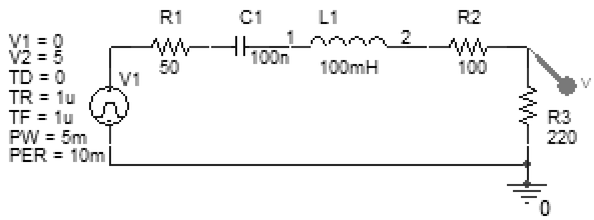
   Compare the peak voltage V_P and time of occurrence, T_P by calculating the percent difference between the measurements and the theoretical calculations.

   Compare the theoretical time of occurrence $T_{20\%}$, of the 20% amplitude, $V_{20\%}$, by calculating the percent difference between the measurements and the theoretical calculations.

4. Optional: Simulate the circuit and compare simulated results to measured results. See example simulations below.

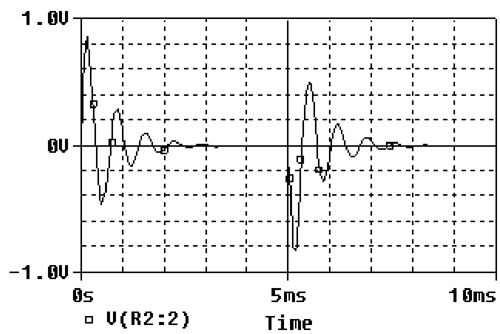
PSpice Example: RLC Step Response

"Vpulse" was used as the source for a 100Hz square wave with the settings shown in the circuit diagram on the right.

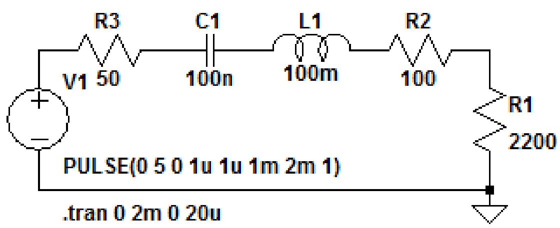


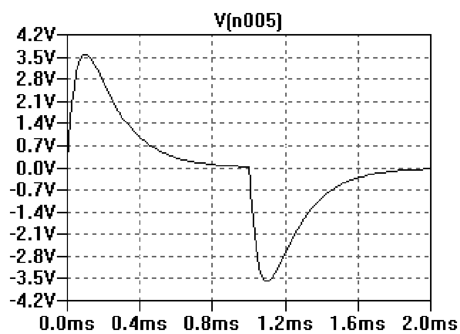
A transient simulation was run for 10mS to generate the graph on the right.

Cursors can be used to measure the peak voltages and their time of occurrence.



LTspice Example: RLC Step Response



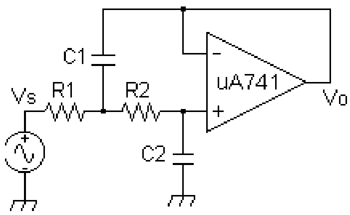


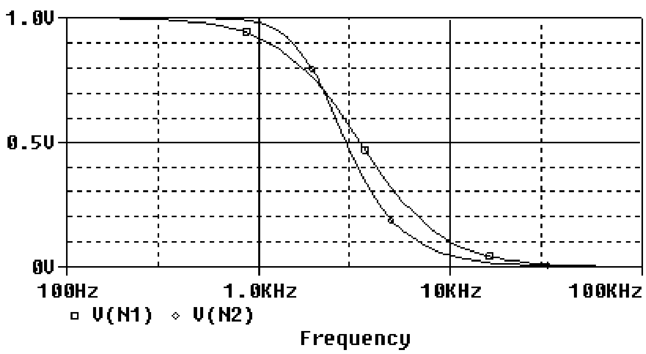
Experiment 41: Active Two-Pole Low-Pass Butterworth Filter

Introduction

A Butterworth unity gain low-pass filter frequency response is given by the equation:

$|H(j\omega)| = \dfrac{1}{\sqrt{1+(\omega/\omega_c)^{2n}}}$, where ω_c is the cutoff frequency and n is the order of the filter.





A second order low-pass Butterworth filter circuit is shown above. Its frequency response, V(N2), is compared to the frequency response, V(N1) of the cascaded second order filter in the previous lab experiment (40). The Butterworth filter has a much flatter response in the pass band and more attenuation beyond the cutoff frequency.

If R1 is set equal to R2, the transfer function of the second order Butterworth filter is:

$$H(s) = \frac{\left(1/R^2\, C1C2\right)}{s^2 + \left(2/RC1\right)s + \left(1/R^2\, C1C2\right)}. \quad If\ R1 = R2 = R\ \ and\ \ C1 = 2C2,\ \ \omega_c = \frac{1}{\sqrt{2}RC2}.$$

Higher order Butterworth filters can be designed by cascading filter sections using the appropriate “Butterworth polynomials” to calculate the part values required for each section.

Objectives

The frequency response of a two pole active low-pass Butterworth filter will be measured. The results will be compared to theoretical calculations and to simulation.

Procedure

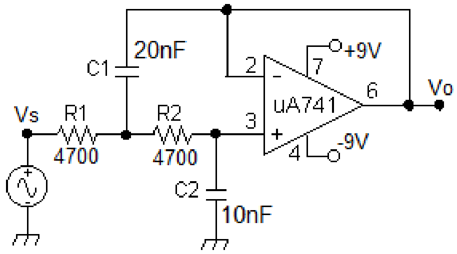
Equipment and Parts

Power supply, Oscilloscope, Function Generator, DMM, Breadboard.
Resistors: Two 4.7 KΩ, ¼ watt, 5%. Capacitors: 10 nF, 20 nF, 5% (or 11 nF and 22 nF)
Operational Amplifier: LM741 or equivalent.

1. Measure and record the values of your parts.

R1 __________ R2 __________ C1 __________ C2 ___________

2. Connect the circuit on the right. Connect the generator and oscilloscope channel 1 to Vs. Connect oscilloscope channel 2 to Vo.



3. Set channels 1 and 2 of the oscilloscope to 2 V/DIV. Set the time base to 20 mS and set the trigger to channel 1, positive slope.

4. Set the function generator to produce a 10 V peak-to-peak, 10 Hz, sine wave. Two cycles should be observed on both channels of the oscilloscope. Channel 2 amplitude should be about the same as channel 1 (10 V peak-to-peak).

5. Generate a table of filter output voltages at specified frequencies. Amplitude of the generator must remain constant at all frequencies. Measure the output amplitude as accurately as possible by using the largest oscilloscope wave form display.

   Use the channel switch on the oscilloscope to switch between channels 1 and 2 when making measurements. Measure the output voltage at the following frequencies: 10, 100, 500, and 1000 Hz.

   Measure the frequencies where the output is 7.07 V, 5.00 V, 1.00V, and 0.100V.

	A	B
1	Frequency	Output
2	Hertz	Volts
4	10	
5	100	
6	500	
7	1000	
8		7.07
9		5.00
10		1.00
11		0.10

6. Record your measurements into spreadsheet columns A and B. Suggested layout is given on the right.

7. Optional: Design a third order Butterworth filter whose cutoff frequency is the same as this lab's two pole filter. Refer to your textbook.

Analysis

1. Use the spreadsheet to plot the frequency response of the filter. Set the x-axis and the y-axis on the spreadsheet plot to "log".

2. Use the slope of the plot for frequencies one decade beyond the cutoff frequency to determine the filter's attenuation in units of dB per decade.

3. Calculate the filter's theoretical cutoff frequency. Calculate the percent difference between the theoretical and measured cutoff frequencies.

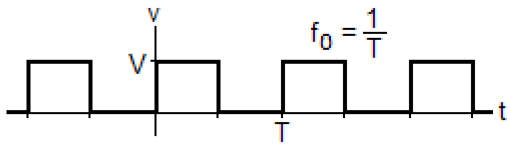
4. Simulate the circuits using your measured component values. Compare the simulated and calculated cutoff frequencies. They should be the same. Compare the measured and simulated attenuation in dB per decade.

Experiment 45: Band-Pass Filter / FFT / Square Wave

Introduction

According to Fourier analysis, a square wave may be considered to be a superposition of an infinite number of odd harmonic frequencies whose amplitudes decrease inversely with frequency. The fundamental is the lowest frequency of the square wave. The Fourier series of a square wave voltage, whose peak-to-peak amplitude is V, and average value is V_{AVE}, may be expressed as:

$v(t) = V_{AVE} + \frac{2V}{\pi}\sum_{n}^{\infty}\frac{1}{n}\sin(2\pi n f_0 t)$, where n is an odd integer and the fundamental frequency is f_0.



If a square wave is applied to a band-pass filter, the wave's frequency components will be affected by the filter's frequency response. The filter's output frequency spectrum will be different than the input frequency spectrum.

In Fourier analysis, the concepts of "time-domain" and "frequency-domain" are important to understand. The square wave input to the filter and output from the filter are considered to be expressed in the time-domain when amplitude is expressed as a function of time.

An oscilloscope normally plots waveforms in the time-domain. When the amplitude of the waveform is expressed as a function of frequency, it is expressed in the frequency-domain.

Using Fourier analysis, it is possible to convert time-domain expressions into the frequency-domain, and frequency-domain expressions into the time domain. This may be done by a computer algorithm called the "Fast Fourier Transform", or "FFT", which is built into many analysis software packages, including *OrCAD PSpice*.

Although the Fourier series of any time-domain waveform consists of an infinite number of harmonic frequencies, an adequate approximation may be obtained using only the first several terms of the series. This exercise will involve the average value of the series plus the first 3 harmonics. These are:

$$v(t) = V_{AVE} + \frac{2V}{\pi}\sin(2\pi f_0 t) + \frac{2V}{3\pi}\sin(6\pi f_0 t) + \frac{2V}{5\pi}\sin(10\pi f_0 t)$$

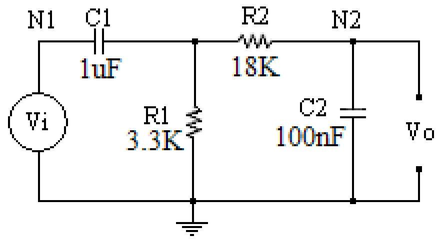
Objectives

The FFT feature of the Tektronix TDS oscilloscope will be used to display and measure the response of a band-pass filter to a square wave input. The results will be compared to theoretical calculations.

One objective of this experiment is to investigate the relationship between time-domain analysis and frequency-domain analysis. Acquired time-domain data will be compared to simulation data using *OrCAD PSpice.*

Procedure

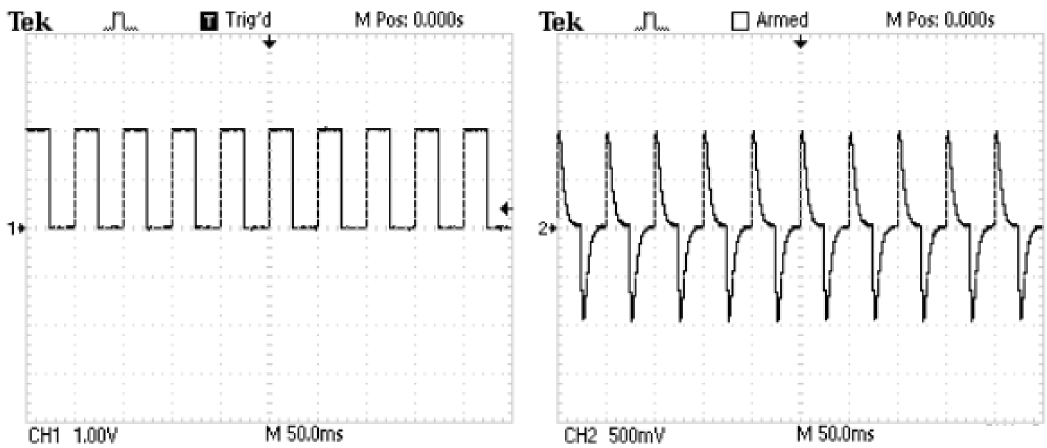
Equipment and Parts
Function Generator, Digital Oscilloscope with FFT, and Breadboard. Resistors: 3.3 KΩ, 18 KΩ, ¼ watt, 5%. Capacitors: 1 µF (non-polarized), 100 nF, 5%.



1. Connect the circuit on the right. Connect the oscilloscope channel 1 to node N1 and channel 2 to node N2.

2. Vi is the function generator set to produce a 20 Hz, 2 V peak-to-peak, square wave with a 1 V offset (goes between 0 and 2 V).

3. Set the oscilloscope to display 10 cycles of the input and output waveforms as shown below.



It may be helpful to read the section on displaying the Fourier spectrum of your instrument's user manual.

The TDS1002 uses a "Math FFT" algorithm to generate the Fourier spectrum from the acquired time-domain data. The time-domain waveform needs to be set up carefully to get an accurate display of the spectrum. Please note the following:

a. The FFT spectrum is calculated from the center 2048 points of the time-domain waveform. Since there are 2500 points in the 10 divisions of the entire display, the FFT is calculated from about the center 8 divisions.

b. There is a tradeoff between frequency resolution and the bandwidth of the displayed spectrum (due to aliasing).

c. The highest frequency (and bandwidth) that can be measured accurately by a digitizing oscilloscope is one half the sample rate (the Nyquist frequency).

d. The TDS1002 transforms 2048 time-domain points to 1024 frequency-domain points resulting in a spectrum whose bandwidth is equal to the Nyquist frequency.

e. The amplitude is displayed in dB where 0 dB is equal to 1 V rms.

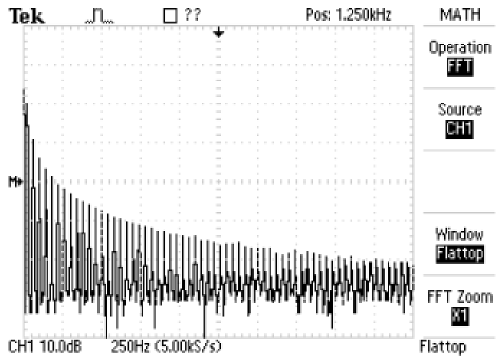
$$dB_{VRMS} = 20\,\mathbf{log}\frac{V_{RMS}}{1Vrms} \; .$$

f. Use the cursors to determine the amplitudes of the harmonics.

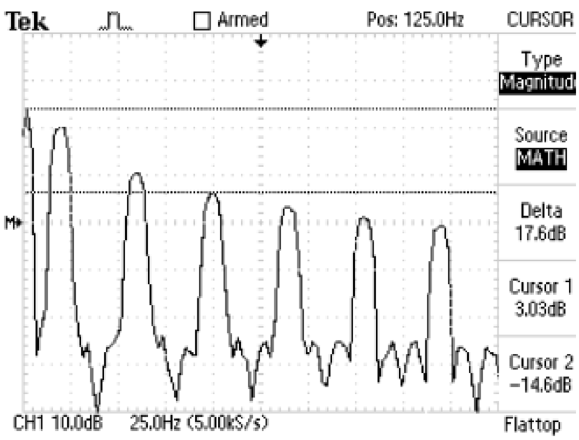
g. Use data capture such as the TekXL toolbar for greater accuracy.

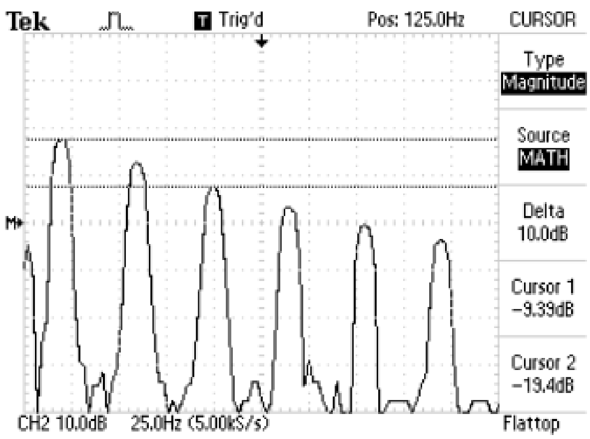
4. To obtain the FFT spectrum of channel 1: Push the "MATH MENU" button, select "FFT", set the MATH FFT source to channel 1.

Note the resulting display on the right.
The frequency scale is 250Hz/Div.
The amplitude scale is in dB, where
0 dB = 1 V rms.



5. The frequency scale needs to be expanded by a factor of 10 in order to accurately measure the frequency components. This could be done with the time base (SEC/DIV) control. However, this would reduce the bandwidth from 2500 Hz to 250 Hz. The TDS1002 has an FFT zoom control that provides a zoom up to 10. A zoom factor of 10 was used to obtain the input (CH1) and output (CH2) frequency spectrums shown below.





Cursors should be used to measure the amplitudes. The input frequency spectrum (channel 1) above shows an rms amplitude value of 3 dB at 0 Hz. This corresponds to 1.414 V rms, which is equal to a 1 V average value (DC offset). The fifth harmonic at 100 Hz has an amplitude of –14.6 dB. This corresponds to a peak voltage of 0.254 V. The output frequency spectrum shows the fifth harmonic amplitude to be –19.4 dB.

6. Display the first six harmonics of the input and output spectra of your band pass filter. Record the amplitude of the DC component and the first three harmonics of the input and output spectra (DC, 20 Hz, 60 Hz, 100 Hz).

Vi_{DC} ________dBV Vi_{20} ________dBV Vi_{60} _________dBV Vi_{100} _________dBV

Vo_{DC} _______dBV Vo_{20} ________dBV Vo_{60} ________dBV Vo_{100} _________dBV

Analysis

Note: Use a spreadsheet as applicable to do the analysis below.

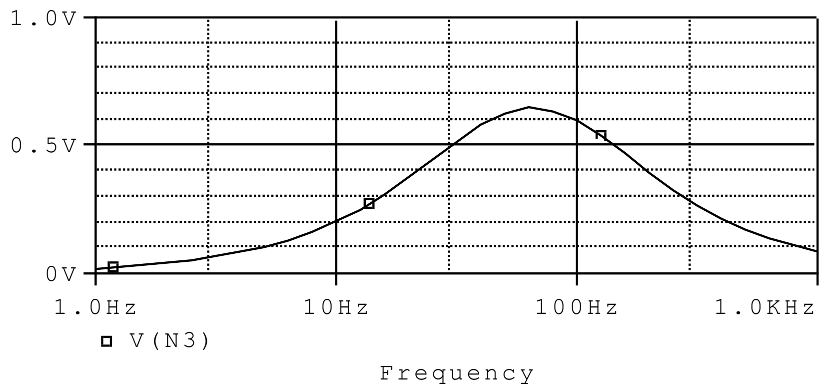
1. The Fourier series of the input waveform is that of a 2 V peak-to-peak amplitude square wave with a 1.0 V DC component (average value). Determine the theoretical magnitude of the waveform's DC component, fundamental, third harmonic and fifth harmonic. Convert your calculated results to dBV using the following equation:

$$\mathrm{dBV} = 20\log\frac{\mathrm{Vrms}}{1\mathrm{Vrms}}.$$

Compare your measured values to the theoretical values by calculating the percent difference between them.

2. Simulate the filter circuit with *OrCAD PSpice* and use the FFT feature of *Probe* to obtain the frequency spectrum of the output. Compare the simulated results to your calculated results from step 1 above. See PSpice example on the next page.

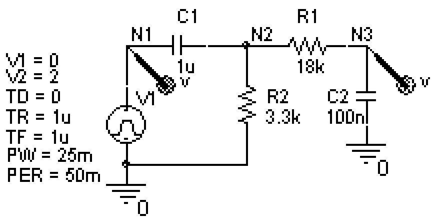
3. Simulate the circuit with *OrCAD PSpice* and use the frequency sweep analysis to obtain a Bode Plot of the filter's response (similar to plot shown below). Compare the results to analysis steps 2 and 3 above.

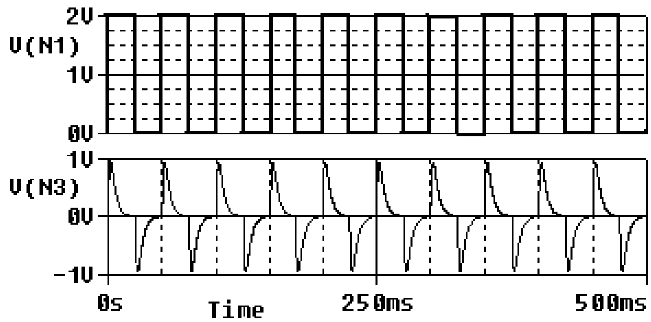


PSpice Example: Fourier Analysis / Square Wave

It is easy to obtain the frequency spectrum of a time-domain waveform in *OrCAD PSpice.* There is an FFT button in the main menu of Probe that converts the time-domain display to frequency-domain. Display about 5 to 10 cycles of the waveform to get good resolution in the frequency-domain.

The waveforms and spectrum shown below are for the simulation of a band-pass filter circuit. The square wave is generated by the part “VPULSE”.



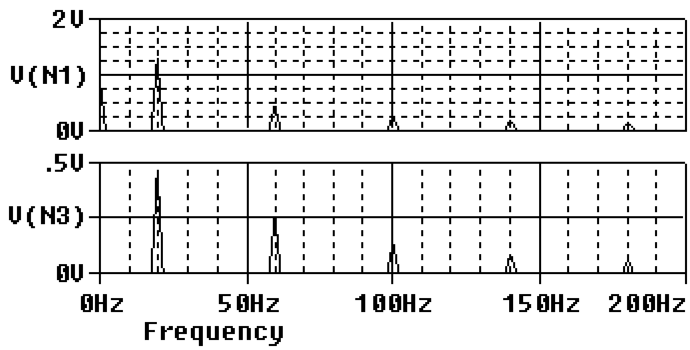


Filter Circuit

The spectrum displayed in the Probe window is a plot of the peak voltage at each harmonic frequency.

To convert these voltages to dBV apply the following equation:

$$dBV = 20\log(\frac{V_{PEAK}}{\sqrt{2}})$$



For example, the output spectrum shows 0.47 volts at 20Hz and 0.26 volts at 100Hz.

$dBV_{20HZ} = 20\log(\frac{.47}{\sqrt{2}}) = -9.57$, and $dBV_{100HZ} = 20\log(\frac{.15}{\sqrt{2}}) = -19.5$.