Electrical Engineering 233

Introductory Electrical Engineering Laboratory

Experiments

and

Laboratory Manual

by Robert C. Maher, Assistant Professor
with Duane T. Hickenbottom, Graduate Assistant

University of
Nebraska
Lincoln
Department of Electrical Engineering

Contents

This manual and the experiments were compiled, written and/or edited by Robert C. Maher and Duane T. Hickenbottom during Fall Semester 1991, with revisions during Spring Semester 1992.

Laboratory Procedures and Reports

The purposes of this laboratory course are to *learn* the basic techniques of electrical measurements, to *practice* essential laboratory notebook and report preparation skills, and to *reinforce* the concepts and circuit analysis techniques taught in EEngr 213.

Lab Format

Each of the lab manual entries consists of several sections: Abstract, Introduction and Theory, References, Pre-lab Preparation, Experiment, and Results.

The *Abstract* is a brief summary describing the experiment.

The *Introduction* and *References* sections provide some of the background information necessary for the experiment. This material is intended only to be supplementary to the classroom lectures and exercises in EEngr 213, and the material covered in EEngr 121 and 122.

The *Equipment* section lists the components and measurement instruments needed to perform the experiment.

The *Pre-lab Preparation* section contains several tasks that must be performed *BEFORE* arriving at the lab. The pre-lab typically requires calculation of specific component values, prediction of the behavior to be measured in the lab, and preparation using computer simulation.

The *Experiment* section contains a description of the circuits, components, and the actual lab measurements to be recorded in the lab notebook. This section is the *minimum* required effort: you are encouraged to try additional ideas once the required measurements are made. Ask the lab teaching assistant (TA) for guidance with this.

Finally, the *Results* section lists the minimum required items to be presented in the final lab report.

Lab Evaluation and Grading

Your grade in this laboratory course is based upon several components.

Pre-lab exercises	20%
TA evaluation of your lab skills and knowledge	20%
Lab practical final exam	20%
Lab results and reports	40%

The *pre-lab exercises* are to be turned in at the beginning of the lab period. If you need any of the pre-lab results in order to perform the experiment you will have to make a copy of your pre-lab solutions, preferably in your lab notebook. Your TA should grade the pre-labs during the lab period and return them to you before you leave.

The *TA evaluation of lab skills* will be based on how well you respond to the questions he or she will ask you during the experiments throughout the semester. In other words, the TA will occasionally ask questions to ascertain whether you (i) understand the theory and rationale for the experiment, (ii) are able to operate the lab instruments properly, and (iii) can express your knowledge in a meaningful, concise manner. *Hint:* if you do not understand the lab concepts it is probably advisable to ask your TA for help *before* he or she puts you on the spot!

A *lab practical exam* will be given in order for you to demonstrate your analytical ability and proficiency with the equipment, circuits, and concepts considered in this course. The content, format and schedule of the practical exam will be determined by your TA.

Finally, the largest individual component of your lab grade will be based on your *lab results and reports*. Some suggestions for your lab notebook and lab reports are given next.

Analysis of Data and Errors

All lab measurements are subject to some degree of uncertainty. Although these uncertainties are casually referred to as "errors", it is better to realize that they represent a fundamental constraint on any physical measurement. Specifically, it is vital to understand *and specify* the uncertainties in all measurements: it is unreasonable to obtain a result using a calculator to 10 significant digits if the measurements used in the calculation have only 3 significant digits.

Each electrical component and measurement instrument has a limitation of *tolerance*, *accuracy*, and *precision*. Tolerance refers to the discrepancy between the marked or "nominal" value of a component and its actual value. Tolerance is often expressed as a percentage of the nominal value, such as a voltage source specified by the manufacturer to be "10 volts ± 1%": meaning that the actual voltage is somewhere between 9.9 volts and 10.1 volts. Accuracy refers to the discrepancy between the actual value of a quantity and the reading given by a particular measurement instrument. Accuracy is related to the concept of *calibration*, where an extremely accurate instrument or measurement technique is used to adjust the accuracy of another instrument. Precision is different from accuracy. Precision refers to the repeatability and stability of a particular instrument, i.e., the deviation of the reading from measurement to measurement.

Lab Notebook

Good laboratory practice begins with pre-lab preparation. It is essential to read the lab manual, perform the pre-lab assignments, and carefully think through all the steps to be performed and the measurements to be made. This process centers around the need for good documentation: a lab notebook.

The lab notebook is a complete record of ALL work pertaining to the experiment. It is not necessary to include lengthy explanations and procedures in the notebook, but *the entries must be sufficient for another person to understand your methods and replicate the experiment*. The purpose of the notebook is to follow the required practices of industrial or academic research and development laboratories, where complete and accurate records of laboratory work are vital. The lab notebook is a legally recognized paper that is essential in documenting inventions, discoveries, and patent disclosures. Some companies require lab notebooks to be officially notarized and filed so that any legal questions later on can refer directly to the original, unaltered notebook entries.

The pages of the notebook must be bound (not loose leaf or spiral) and should be numbered consecutively. The notebook entries must be in ink, and no pages should be left blank between entries. Begin the entries for each experiment on a new page, giving the title of the experiment, your name, the name of your lab partner, and the date. In case some of the data or calculations written in the notebook turn out to be in error, *do not* tear out the page or completely obliterate the entries: a *single line* through the error is preferred. This way there is no question regarding the legitimacy and completeness of the notebook material. ***Furthermore, you will not be***

penalized in this course for having lined-out errors and corrections in your notebook.

Pre-lab preparation exercises should be written in the notebook, along with an outline of the experiment to be performed, circuit sketches, and anticipated results. The expectations and predictions from the pre-lab work are extremely important in guiding the measurements actually made in the lab. For example, if we expect a linear relationship (y = m*x) between two circuit parameters and measure something completely different, we are in a position to double-check the circuit and the measurement techniques to discover whether the discrepancy is due to an incorrect circuit, limitations of the measurements, or faulty assumptions and predictions. Similarly, the ideal number and spacing of data points on a graph can guide the number and spacing of lab measurements. Always determine the level of accuracy of each measurement and include these limitations with the results.

The process of comparing expectations and results is best accomplished *during* the lab period: this way any questionable data can be verified and further measurements can be made. Along with tables of the measured data it is useful to sketch graphs of the measurements. This helps to spot trends or errors in the data. For the same reason it is also desirable at least to "work through" the required solutions to the Results section of the experiment *before* leaving the lab.

Lab Report

All jobs in electrical engineering require proficiency in technical writing. The written lab report is just one example. The report should be written specifically to meet the needs of the reader, meaning that the writing must be brief, interesting, and complete. It is good engineering practice when writing to always begin with a *summary* of each important conclusion, followed by the *results and reasoning* that led to that conclusion, and finally a *review* of what was stated. Keep in mind that the specific format and content requirements of the lab report may vary depending upon the preferences of the reader, in this case the lab TA.

The report should be assembled in some reasonable manner, such as:

- Cover Page showing the experiment title and number, date experiment was performed, date report was finished, and the name of the author and partner.

- Abstract giving a summary of the *complete* report. The abstract is normally written last, and should be no more than 3 or 4 sentences.

- Introduction giving a 1 or 2 paragraph explanation of what the reader must know to understand the report. Basically, the introduction indicates whether you understood what the lab was all about!

- Procedure describing the steps used in the lab. This need only be sufficient to recreate the experiment in conjunction with the lab notebook, *not* a lengthy minute-by-minute account.

- Results and Discussion of the experiment, including the requested information from the lab manual, comparisons with pre-lab predictions, and *reasonable* explanations of any difficulties or surprising results. Tables of "raw" data should be left in the notebook, not in the report, except where necessary to support the discussion. Do include graphs of the results where appropriate. Include a discussion of the methods and circuits used in the experiment and indicate any extra measurements or investigations you made in addition to the steps in the lab manual.

- Conclusion giving the main items learned in the experiment.

Before finalizing the report to turn in, look over the entire report with a critical eye. Is the report complete and concise? Is the substance of the report good enough *that you would show it to a potential employer* as an example of the quality of work you do? Does it indicate that you know what you are doing? Are the sections labeled? Are the graphs labeled and *interpreted* (slopes, breakpoints, etc. identified)? Are the circuit diagrams accurate and labeled? Do you tend to use imprecise, meaningless phrases like "very large", "negligible", "this experiment demonstrates to the student...", "the results validate the theory", etc.?

Lab Safety

Safety in the laboratory includes not only preventing physical harm to your body, but also avoiding damage to equipment and lab components. Although the experiments in this course are believed to be safe, you must share responsibility for your own well-being.

It may seem reasonable that 25,000 volts is more deadly than 50 volts, *but this is not necessarily true*. A static electricity spark may involve a potential difference of thousands of volts, but is more of an annoyance than a cause of death in most cases (except, of course, if the static discharge is in the form of lightning!). Instead, the danger of electric shock is related to the *current* that flows through the body; a small spark of static electricity has insufficient current to cause serious injury. The human body is actually most susceptible to fatal injury for currents in the range of 100 to 300 *milli-amperes*. Currents below this level can be painful and cause damage to tissue but are rarely fatal. Currents above this range can cause severe burns and other injuries but may not always cause death. Why? It is because current in the 100 to 300 mA range is sufficient to disrupt the electrical activity of the heart, causing fatal ventricular fibrillation, while at higher currents the muscular contractions can be so severe that the heart is essentially "clamped" during the shock and may resume beating properly if the shock is removed quickly and resuscitation is started.

A few simple electrical laboratory safety guidelines apply:

1) There must always be at least three people in the lab at one time: one able to assist the injured person and one to go for help.

2) All electrical apparatus that connects to the AC power line must have a protective ground through a three-wire power cable.

3) Always double check circuit wiring before applying power. Always have a single switch or button that will immediately remove power from the circuit in case of trouble.

4) Always switch the circuit power off before changing components or connections. It is tempting to become lazy and change connections in low-power circuits with the supply on, but this is asking for trouble in the form of unintentional short circuits and blown components.

5) Always ask for directions or help if you are unsure of the correct measurement procedure or circuit connection. *Be honest with yourself*: if you don't understand what you are doing, seek assistance from the lab TA.

Some Final Words...

This lab course has been put together to benefit a specific customer: *you*. If you are having difficulty understanding the experiments and concepts because of the way in which the material is presented, let your lab TA know! You will notice that part of the results section for each experiment is a question asking you to give specific examples of how you would change the experiment to make it more understandable, more interesting, and more useful to you.

By the time you have completed this course you will have gained some knowledge and confidence in making laboratory measurements using the basic tools of the electrical engineering trade. You will probably notice that the experiments in this lab often have some *cookbook* directions: "assemble the circuit, connect wire A to point B, write down the voltage at point C, repeat until done". It's important that you do not get the idea that engineering lab work consists of blindly following a bunch of tedious directions. As you progress through your undergraduate studies, get your diploma, and head out into the real world, you will be expected to do more and more of your own experimental design work. You can use your practical lab experiences now as a basis for your future efforts in engineering.

A lot of the real fun in electrical engineering comes as you begin to design, build, and test your own circuits. Although much of the experience in this course will be *analytical*, that is, making measurements of existing circuits, you should always be thinking about what it means to create a new design from scratch. I have always found that the creative aspects of *engineering design* are the most satisfying and rewarding elements of electrical engineering. I hope you will, too.

Have fun!

Revised 4/92

Resistors
(5% tolerance, carbon or carbon film, 1/4 watt)

Qty	Nominal Value
5	1Ω
6	10Ω
10	100Ω
4	220Ω
4	330Ω
4	470Ω
6	1kΩ
2	2.2kΩ
2	10kΩ
2	100kΩ
2	1MΩ

Resistors
(5% tolerance, carbon or carbon film, 1 watt)

Qty	Nominal Value
2	100Ω

Capacitors

Qty	Nominal Value and Type
2	0.1µF (polyester or plastic film, ±10% tolerance or better)
2	0.01µF (metal film bypass capacitors)

Semiconductors

Qty	Component
2	UA741 Op Amp
2	74LS00 Quad NAND
2	74LS02 Quad NOR
2	74LS04 Hex INVERT
2	74LS08 Quad AND
2	74LS32 Quad OR
2	74LS74 Dual D FF
2	74LS86 Quad XOR
2	74LS107 Dual JK FF

Miscellaneous (optional)

Qty	Component
1	Breadboard
5	Alligator Clips
2	Clip Leads

Lab # 1
TITLE: Introduction I: Basic Lab Equipment and Measurements

ABSTRACT

This lab exercise introduces the basic measurement instruments that will be used throughout this course. These instruments include *oscilloscopes*, *multimeters*, *AC signal generators*, and *DC power supplies*. Although the particular measurement devices used in this lab are not highly sophisticated, the basic operation and measurement concepts presented here are extremely important: this basic knowledge is assumed of electrical engineers regardless of specialization. *The introduction is continued in Lab # 2.*

INTRODUCTION AND THEORY

Electrical laboratory work depends upon various devices to supply power to a circuit, to generate controlled input signals, and for circuit measurements. The basic operation of these instruments may seem somewhat complicated at first, but you will gain confidence as your experience grows. Eventually the operation of an oscilloscope or multimeter should seem as natural to you as punching an expression into your scientific calculator.

Electrical measurements, like all physical measurements, are subject to uncertainty. The sources of uncertainty include so-called "human" errors, like misreading a dial setting, systematic errors due to incorrectly calibrated instruments, and random errors due to electrical noise and interference, environmental changes, instrument resolution, or uncertainties in the measurement process itself. An effective engineer needs to keep measurement uncertainty in mind: *nothing is gained by performing mathematical operations to 8 significant digits if the original lab data contains, say, only 3 significant digits with +/- 5% error*. Thus, it is very important to consider measurement errors when performing laboratory work.

Frequently you may find that several possible approaches are available to make a particular measurement. By considering the shortcomings of each measurement technique you may discover that one of the possible methods is better (less prone to error) than the others. Moreover, you may be able to *verify* a questionable result by choosing alternate measurement methods. For example, a resistance can be measured by supplying a known current and measuring the voltage, or by supplying a known voltage and measuring the current, or by using a voltage divider or bridge containing a known resistance.

Basic Equipment: The multimeter.

Each lab station is equipped with a digital multimeter. It is called a *multi*meter because the same instrument can be used to measure voltage, current, and resistance: voltmeter, ammeter, and ohmmeter. The voltmeter measures the voltage difference *between* two nodes in a circuit, i.e., the voltmeter is connected *in parallel* with a circuit branch. The ammeter must be connected *in series* with a circuit branch, which means that the circuit must be connected so that the current in the branch of interest passes through the meter. The ohmmeter is used to test *isolated* resistances. An internal battery in the ohmmeter is used to determine the resistance of a circuit element via Ohm's Law. Note that it is generally impossible to measure a resistance *in situ*: the element should be removed from the circuit to avoid unwanted "stray" current paths.

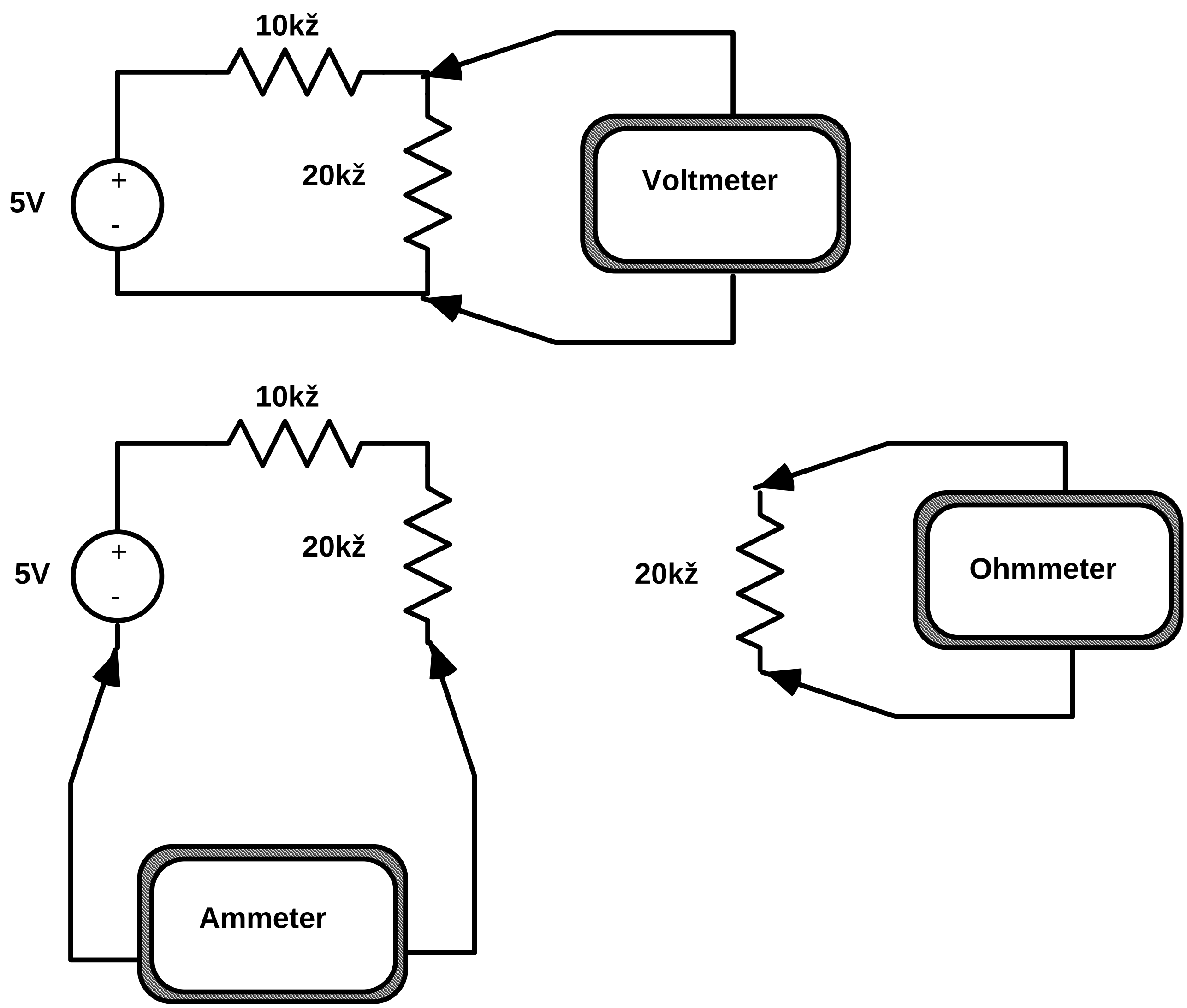


Figure 1: Basic Meter Connections

Although an "ideal" voltmeter would act like an open circuit (no current through the meter) and an ideal ammeter would act like a short circuit (no voltage across the meter), the instruments actually available for these measurements cannot be perfect. This means that the multimeter may actually *load* the circuit being measured by changing the resistance between the test points. This problem is particularly important when measuring circuits with resistances comparable to the meter resistance.

In order to obtain the best resolution from a multimeter several different sensitivity ranges are usually provided. This allows the limited number of digits available in the display to be used most effectively. For example, a voltmeter might have a smallest range setting to measure voltages from 0 to 500 mV with 100 μV resolution, a largest range setting to measure voltages from 0 to 500 V with 100 mV resolution and several range settings in between. Thus, we would obtain the most accurate measurement of a voltage by choosing the *smallest* range setting that is still larger than the input voltage.

The particular multimeter used in the lab (Fluke model 37) is *autoranging*, which means that it will automatically choose the proper display range for the measurement being made once you select an appropriate setting such as "V", or "mV". You may encounter other meters that require you to manually set the proper range using a panel switch. To measure an *unknown* input signal with such a meter you should always start out with the highest range setting, determine the approximate value of the signal, then reduce the range setting to the optimum level. An autoranging meter performs most of this process automatically.

Examining the Fluke 37 multimeter you will notice the digital readout in the upper right corner, the volt-amp-ohm function selection dial in the middle, and the signal input jacks on the left. The input jacks include "V Ω" for measuring volts and ohms, "mA/μA" for measuring current up to 320mA, "A" for measuring current up to 10A, and "COM" (*common*) which is used in conjunction with the other three jacks. The various jacks are required because of the multipurpose nature of the meter. Recall that a good voltmeter behaves like an open circuit, drawing negligible current from the circuit under test, while a good ammeter behaves like short circuit. There is a very high resistance between the "V Ω" jack and the "COM" jack (acts like an open circuit), while the "mA/μA" and "A" inputs have a small resistance to the common jack (acts like a short circuit).

It is very important to connect the inputs properly. One common error is to connect the test leads to the <u>current</u> jacks, then attempt to measure a <u>voltage</u> source: the low resistance of the current inputs essentially short circuits the circuit being measured. It is likely to blow the meter's internal fuse, damage the meter, or even damage the circuit being tested by this mistake! Similarly, the "mA/μA" input must not be used to measure currents greater than 320mA. Be familiar with the maximum and minimum limitations of this or any other meter *before* making any measurements.

Basic Equipment: The DC power supply.

Active electronic circuits require a power supply providing electrical energy to operate the circuit. The power supply can be a battery, a DC supply operating from the AC power line, or some other source such as a solar cell, fuel cell, generator, or thermoelectric element. Several experiments in this lab course will make use of a line-powered DC power supply.

DC lab bench power supplies usually have so-called "floating" output. This means that the supply acts like a battery, producing *X* volts between its output terminals with neither terminal internally connected to chassis (earth) ground. Thus, you can choose to use the DC supply as a positive or negative voltage with respect to a circuit reference node. It should be noted that although DC lab bench supplies often contain an internal meter indicating the supply's DC output voltage and/or current, the internal meter is typically *not* extremely accurate: the output voltage should be adjusted using an accurate multimeter if the circuit requires a specific voltage.

A *dual* lab bench supply contains two independent DC supplies in a single chassis. A dual supply is often used to produce a *bipolar* or *split* supply providing both positive and negative voltages with respect to a reference node. The Heathkit "trainers" used in some of the experiments in this course contain both a fixed +12 volt and -12 volt supply.

DC supplies often are equipped with a *current limiter* to prevent excessive current from flowing if the supply is incorrectly connected or if sensitive circuit components are used. The current limit setting is usually adjusted to allow the supply to provide the necessary *normal* circuit current at the chosen voltage, but to limit the current if the normal power level is exceeded sufficiently to possibly damage the circuitry.

Basic Equipment: The oscilloscope.

The oscilloscope is arguably the most fundamental measurement device in electrical engineering. The usefulness of the oscilloscope–or 'scope' for short–is due primarily to its ability to display electrical signal information directly in visual form.

A basic oscilloscope can be described in terms of the block diagram shown in Figure 2.

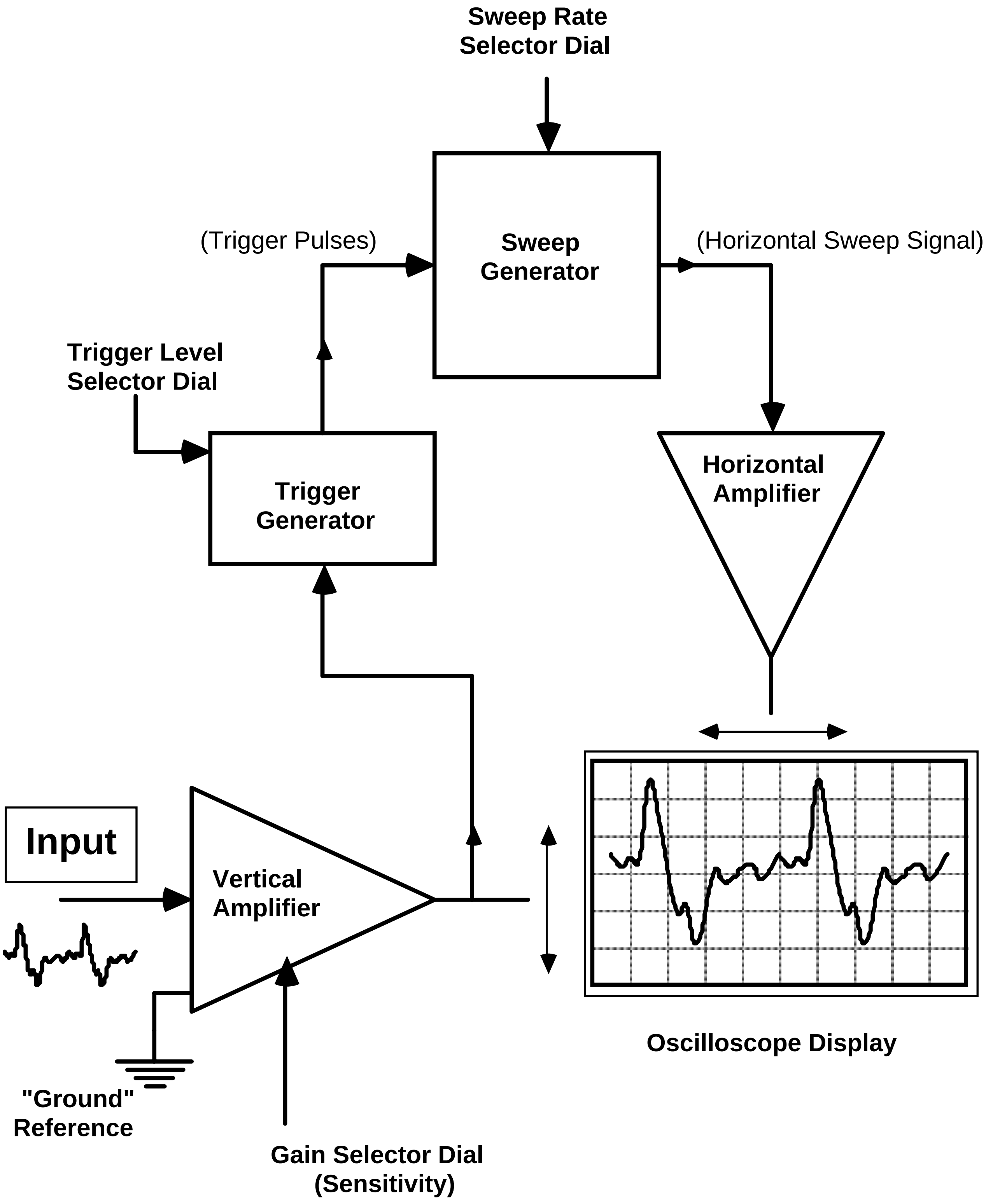


Figure 2: Block Diagram of Basic Oscilloscope

Although it may appear complicated, the basic oscilloscope is quite simple in concept:

- The vertical position, or *vertical deflection*, of the electron beam of a cathode ray tube (CRT) is linearly proportional to the input *voltage*. The input voltage is measured between a particular circuit node and the circuit *ground* reference. A calibrated variable amplifier is used to boost low-level signals in order to produce a visible beam deflection. The input voltage can thus be determined by measuring the amount of beam deflection for a particular input amplifier gain. Vertical deflection calibration is usually indicated as *volts per division*, where "division" refers to the grid lines etched on the face of the CRT.

- The *horizontal deflection* is controlled either by another input signal or more commonly by a calibrated *sweep generator*. The sweep generator moves the electron beam at a selectable, constant rate from left to right across the CRT screen. The calibrated sweep rate is usually indicated as *time per division*, where "division" again refers to the CRT display grid lines.

The most important use of oscilloscopes is in the observation of periodic signals: repetitive waveforms (oscillations) can be viewed by synchronizing the sweep generator with the repetition rate of the input signal. This is accomplished using a *trigger generator* which starts the horizontal sweep when the input signal exceeds an adjustable voltage threshold. The trigger can be selected to occur for either a positive or negative slope at the threshold voltage.

Thus, by adjusting the vertical gain and the sweep speed, time varying input signals can be viewed directly as a voltage-vs.-time display.

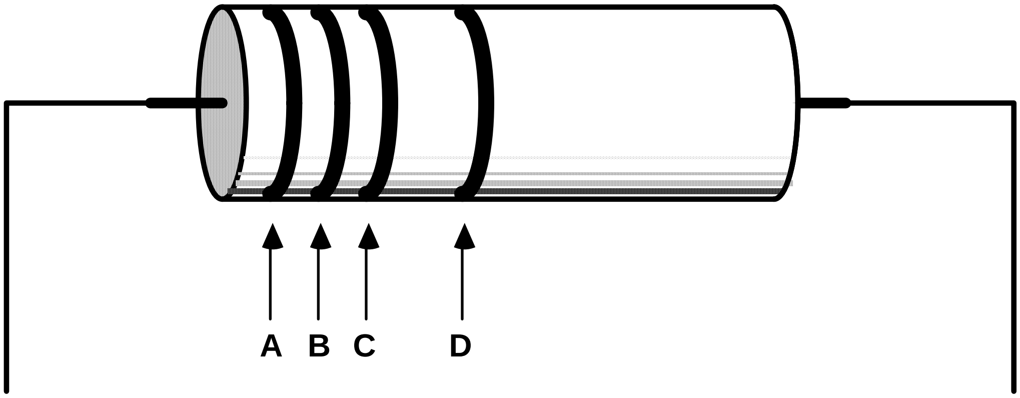
Oscilloscopes have many other features for producing displays of particular types of signals. For example, most 'scopes have two or more independent input amplifiers for displaying two or more input signals simultaneously. Some special features will depend upon the instrument manufacturer and the sophistication of the 'scope itself. Some of these "advanced" features will be considered later in this course.

Basic Equipment: The signal generator.

A source of known waveforms–a *signal generator*–is another standard piece of laboratory gear. A basic signal generator produces a repetitive (periodic) output waveform, typically a sinusoidal signal or square wave, with adjustable frequency and amplitude. A *function generator* is a signal generator able to produce several different waveforms, typically sinusoidal, square, triangle, sawtooth, and pulse. Unlike the DC power supplies, the signal generator is usually used to provide circuit excitation only and not operating power.

Basic Equipment: Resistors.

Real resistors are labeled with an approximate, or *nominal*, resistance value and a *tolerance* specification. For example, a resistor might be labeled as 1kΩ +/- 5%, meaning that the manufacturer guarantees that the actual resistance will be between 950Ω (1kΩ - 5%) and 1,050Ω (1kΩ + 5%). The nominal resistance value and its tolerance for 5, 10, and 20% resistors are either printed numerically on the resistor body (if the resistor is large enough), or indicated by four colored bands (ABCD) on the resistor body.



$$\text{Resistance} = AB \times 10^{C} \pm \text{tolerance (D)}$$

The numerical values corresponding to the colors are:

0 ∅	Black	5 ∅	Green	Tolerance:	
1 ∅	Brown	6 ∅	Blue	20% ∅	No band
2 ∅	Red	7 ∅	Violet	10% ∅	Silver
3 ∅	Orange	8 ∅	Gray	5% ∅	Gold
4 ∅	Yellow	9 ∅	White		
		-1∅	Gold		

For example, a 1kΩ ± 5% resistor [10 x 10^{2}] is labeled BROWN:BLACK:RED:GOLD, a 220Ω ± 10 % resistor is labeled RED:RED:BROWN:SILVER, and a 1Ω ± 5% resistor must be represented as [10 x 10^{-1}], or BROWN:BLACK:GOLD:GOLD.

Resistors are also available in 1% tolerance values. The 1% resistors have five colored bands (JKLMN) instead of four and indicate the resistance value as JKL x 10^M Ω, with the tolerance band (N) colored BROWN.

If we test many resistors with the same nominal (labeled) value we will find that the *actual* resistance varies from resistor to resistor within the tolerance range. On the other hand, the resistance value of a *particular* resistor is usually quite constant with time, staying close to its original measured value.

Resistors are also rated according to their maximum power dissipation. Most of the resistors used in this course are 0.25 watt devices, meaning that they can safely handle up to one-quarter watt of power. Resistors with lower and much higher power ratings are available, with the resistor size increasing with the power rating.

It is important to realize that only a limited number of nominal resistances are available. Manufacturers produce *standard* values that are spaced by approximately twice the tolerance specification (a logarithmic spacing) so that a wide range of resistances are covered by a minimum number of nominal values. The standard 5% resistors, for instance, follow the pattern:

1.0
1.1
1.2
1.3
1.5
1.6
1.8
2.0
2.2
2.4
2.7
3.0
3.3
3.6
3.9
4.3
4.7
5.1
5.6
6.2
6.8
7.5
8.2
9.1
10.0

which then repeats for each greater power of 10.

REFERENCES

For specific questions regarding the use of the lab instruments, consult the operating manuals for each device. The manuals should be available in the lab, or consult your TA.

For general lab instrumentation questions see the references available in the Engineering Library. For example:

> *Electronic Components and Measurements*, by Bruce Wedlock and James Roberge, Prentice-Hall, 1969 (an oldie but a goodie!).

EQUIPMENT

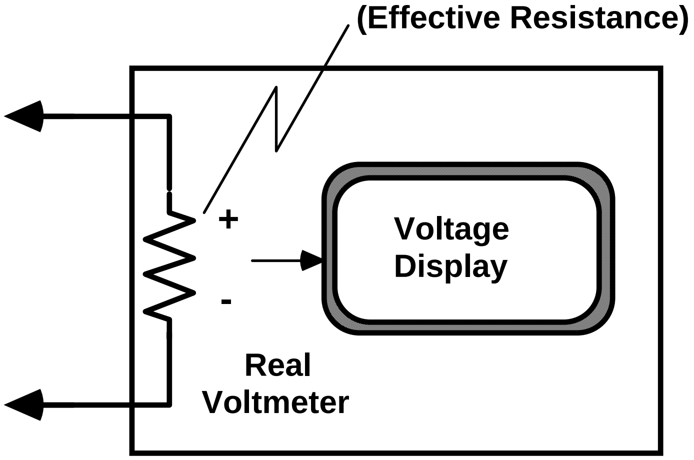
Resistor kit	DC Power Supply
Multimeter (Volt, Ohm, Current)	Oscilloscope

PRE-LAB PREPARATION

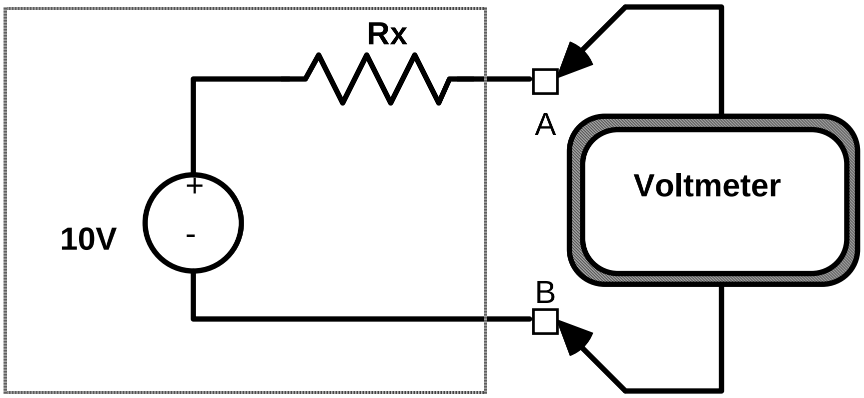
(I) Quickly skim over this entire lab manual. Try to identify the layout of the experiments and the types of measurements to be made.

(II) Carefully read through the experimental procedure for Lab #1. If several steps are required to make a measurement try to decide the best order in which to do them.

(III) An *ideal* voltmeter has an infinite effective resistance, meaning that the circuit being measured is not disturbed by connecting the voltmeter. A real voltmeter, however, has a large but finite effective resistance.



If an *ideal* voltmeter is used to measure the voltage difference between terminals A and B in the circuit below, what would the voltage reading be? Remember Ohm's law (V=IR): if there is no current in a resistor there is no voltage across it. If a *real* voltmeter with an *effective resistance of 1MΩ* is connected to the circuit, what is the voltage reading if R_X is 100Ω ? If R_X is 1MΩ ?



EXPERIMENT

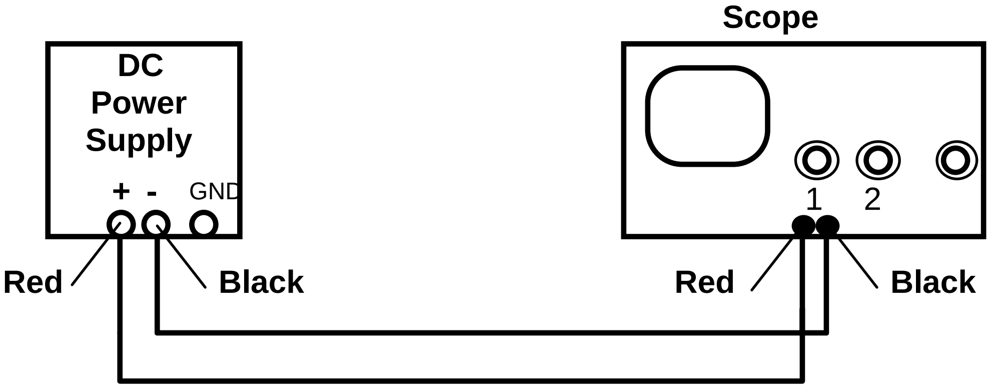
(1) Identify two 10kΩ (nominal) resistors and six 1kΩ (nominal) resistors (0.25 watt) from your lab kit. Measure each resistor using the "ohm" function of the multimeter and record the values in your notebook. Be sure to keep track of which resistor is which: you will need to use the resistors later in the experiment!

(2) Turn on the DC bench power supply. First adjust the voltage and current knobs slightly above the minimum position, then increase the voltage knob until the power supply gauge indicates approximately 10 volts DC. Now use the "DC volt" function of the multimeter to adjust the power supply output to be as close to 10 volts as you can (the accuracy of the multimeter is better than the internal indicator of the power supply).

(3) Now use the oscilloscope to observe the DC power supply voltage. First turn on the oscilloscope and adjust the vertical sensitivity of channel 1 to 2V per division (turn the CH1 knob until 2 volts is next to the '1X' mark). Set the 'scope trigger mode to P-P AUTO and the trigger source to CH1. Adjust the horizontal sweep rate to 1 millisecond per division. Set the vertical display mode switches to 'CH1' only, NORM, and ALT, respectively. The input coupling switch for channel 1 (below the sensitivity knob) should be set to DC. Adjust the intensity control so that the trace is visible on the screen but not extremely bright: HIGH INTENSITY SETTINGS CAN ACTUALLY DAMAGE THE CRT SCREEN!!!

Set the channel 1 coupling switch to the GND (ground) position to temporarily disable the channel 1 input. Adjust the vertical position and channel 1 horizontal position knobs so that the horizontal trace lines up with the lowest grid line on the CRT and extends across the entire display from left to right. When the trace is aligned release the GND switch back to the DC position.

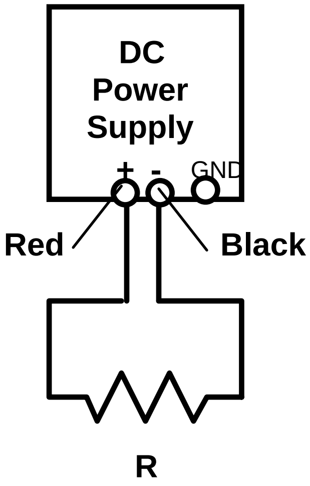
With the DC bench power supply OFF, connect the positive terminal of the supply to the positive (RED) input of 'scope channel 1, and connect the negative terminal of the supply to the "ground" (BLACK) terminal on the 'scope. NOTE that the "floating" output of the DC supply is referenced to the "ground" point at the oscilloscope. Turn on the DC supply and determine the DC voltage by counting the number of divisions of the deflection and the 2V per division vertical sensitivity.



Experiment with different vertical sensitivity ranges. Also compare the measured voltage for different power supply output voltages and the measurements made with the multimeter.

After trying several combinations, re-adjust the power supply using the multimeter for 10 volts output, and turn the supply off without changing the voltage knob setting.

(4) With the supply off, connect any one of the nominal 1kΩ resistors in the circuit below. Then turn on the power supply and *slowly* increase the current limit knob until just above where the current indicator stops changing. *Note* that the deflection of the current indicator may be too small to see: in this case simply set the current limit near the maximum level. Verify that the power supply voltage is still 10 volts. Now turn the supply off (without changing the voltage or current limit knobs) and connect the multimeter in series with the circuit using the "DC current" function. Turn the supply on and record the current in the loop. Repeat the measurements with each of the 1kΩ resistors and each of the 10kΩ resistors. Record all voltage, current, and resistance values.



RESULTS

(a) Prepare a table showing the measured values for the resistors from step 1, the percent difference between the measured value and the nominal (color band) value, and the range of resistances covered by the ±5% resistor tolerance. Did all of your resistors fall within the tolerance range?

(b) It is important to know the precision and accuracy of any lab instrument. When measuring a 10V DC source the Fluke 37 digital multimeter has an accuracy specification of ±(0.1% + 1 digit) and a resolution of 10mV per digit. This means that the actual voltage lies within the range { (reading)x(99.9%) - 10mV } to { (reading)x(100.1%) + 10 mV }. If the meter reading was "10.00" in step 2, what could be the minimum and maximum *actual* voltage due to the meter uncertainty? The range between minimum and maximum represents what percentage of the 10V DC indicated voltage? Repeat these calculations for a meter reading of "3.50" volts DC assuming the same accuracy and resolution specs. Discuss the results.

(c) Discuss the resolution of voltage measurements made with the oscilloscope, i.e., how many *significant digits* can you obtain from the 'scope display?

(d) Using the current measurements made in step 4 and the known resistor values, determine the experimental agreement between the measured values of voltage, current, and resistance the theoretical relationship of Ohm's law: V=IR.
Explain your results. Comment on the possible effects of internal meter resistance on current measurements.

(e) How could this lab assignment be improved?

Revised 7/93

Lab # 2
TITLE: Simple Circuit Measurements and Ohm's Law

ABSTRACT

In this experiment simple electrical circuits containing only resistors and voltage sources are investigated. One of the most basic relationships of electrical engineering, V=IR (Ohm's Law), is examined using measurements of voltage, current and resistance. Standard methods for presenting experimental measurements in both tabular and graphical form are also considered. *This experiment is a continuation of the material from Lab #1.*

INTRODUCTION AND THEORY

Ohm's law is among the most fundamental relationships in electrical engineering. It relates the current, voltage, and resistance for a circuit element so that if we know two of the three quantities we can determine the third. Thus, if we measure the current flowing in a resistor of known value, we can deduce the voltage across the resistance according to V = IR. Similarly, if we measure the voltage across a resistor and the current through it, we calculate the resistance of the element to be R = V/I. Not only does this reduce the number of measurements that must be made, it also provides a way to check the results of several different measurement methods.

REFERENCES

See sections 2.1 through 2.3 of the text by J. David Irwin, *Basic Engineering Circuit Analysis*, 4th ed., Macmillan Publishing Co., 1993 (pp. 21-44).

For least-squares descriptions, see, for example, Erwin Kreyszig, *Advanced Engineering Mathematics*, 6th ed., Wiley, 1988.

EQUIPMENT

Resistor kit
DC Power Supply
Oscilloscope
Signal Generator
Multimeter

PRE-LAB PREPARATION

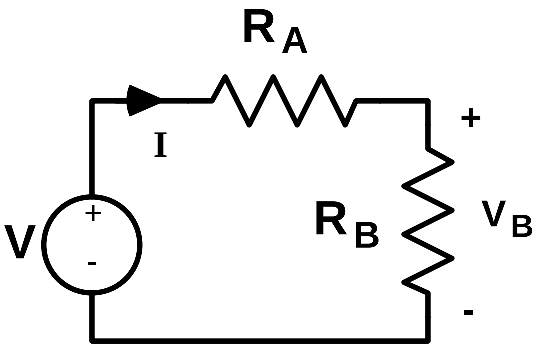


Figure 2.1

(I) Consider the circuit shown in Figure 2.1. Determine a mathematical expression for the current **I** in the loop and for the voltage $\mathbf{V_B}$ across the resistor R_B in terms of the applied voltage *V* and the resistors. What are **I** and $\mathbf{V_B}$ if $\mathbf{R_A}$ = 10kΩ, $\mathbf{R_B}$ = 1kΩ, and *V* = 10 volts?

(II) Now assume that the nominal resistances used in this circuit are actually ±5% tolerance resistors: *$\mathbf{R_A}$ and $\mathbf{R_B}$ may not be exactly 10kΩ and 1kΩ*. If we randomly select a resistor labeled 10kΩ ± 5% and one labeled 1kΩ ± 5% and place them in the circuit, determine the possible *range* of current **I** and voltage $\mathbf{V_B}$ that we might observe due to the ±5% resistor tolerance.

EXPERIMENT

(1) Identify three 1kΩ (nominal) resistors (0.25 watt) from your lab kit. Measure each resistor using the "ohm" function of the multimeter and record the values in your notebook. Keep track of which resistor goes with which measurement!

(2) Turn on the DC bench power supply. First adjust the voltage and current knobs to slightly above the minimum position, then increase the voltage knob until the power supply indicates approximately 10 volts DC. Now use the "DC volt" function of the multimeter to adjust the power supply output to be as close to 10 volts as you can. After setting the power supply for 10 volts output, turn the supply off without changing the voltage knob setting.

With the supply off, connect any one of the nominal 1kΩ resistors across the power supply terminals. Then turn on the power supply and *slowly* increase the current limit knob just until the current indicator stops changing.

(3) Now assemble the simple circuit in Figure 2.2 using the three 1kΩ resistors measured in Step 1. Measure the voltage across each of the resistors using the "DC voltage" function of the multimeter.

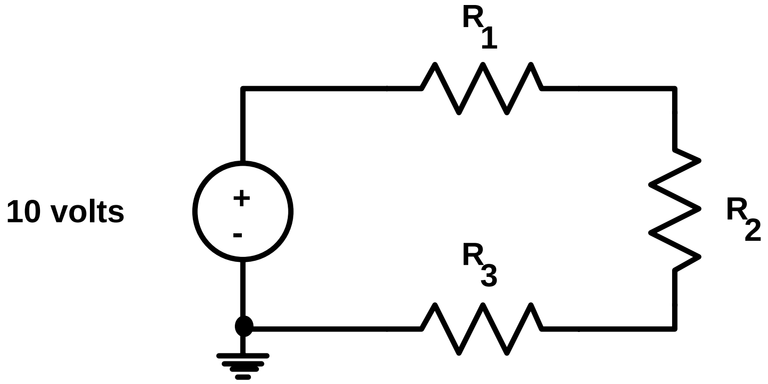


Figure 2.2: $R_1 = R_2 = R_3 = 1k\Omega$ (nominal)

Measure the current in the circuit loop using the "DC current" function of the multimeter. *NOTE* that you must "break" the loop to connect the multimeter current inputs in series with the circuit. Again, it is mandatory practice to turn the power supply OFF while changing circuit connections. Record the resistance, voltage and current measurements.

(4) Repeat this process for applied voltages decreasing to zero in 0.5 volt steps (9.5, 9.0, 8.5,...). Record all the resistance, voltage and current measurements at each step.

(5) Now practice using the oscilloscope and signal generator. Begin by connecting the output of the signal generator (both signal [RED] and ground [BLACK]) to the input of 'scope channel 1. Adjust the signal generator frequency dial to approximately 1kHz output frequency, then use the oscilloscope to adjust the signal amplitude to ±5V. Adjust the trigger level control, sweep rate, and vertical amplifier gain and observe the effect on the waveform display. Record the vertical and horizontal settings that provide approximately two complete signal periods filling the entire screen. Try out all the knobs!

RESULTS

(a) Present the measured and theoretical values for the simple circuit of part 3. Were the resistors measured to be within the ±5% tolerance range? Did the measurements of current correspond to the expected result using Ohm's law (I=V/R)?

(b) Present the results of part 4. Again, how do your measurements correspond to your expectations using Ohm's law? As you did in EEngr 121, determine the *least-squares* value of resistance R_1 *using your measurements*. In other words, use a spreadsheet or other method to calculate the experimental relationship between voltage and current in R_1, assuming the form: $V = RI$. How does the resistance determined from the voltage and current measurements compare to the ohmmeter readings? *Note* that if you don't know what the least-squares curve-fitting method is you should look it up in an engineering mathematics book such as the one listed in the references section above.

(c) Discuss your experience with the oscilloscope. What vertical and horizontal settings did you use to view two cycles of the ±5V ~1kHz signal? Were you able to determine the amplitude and waveform period from the 'scope display?

(d) How could this experiment be improved?

Revised 7/94

Lab # 3
TITLE: Introduction to Digital Circuits Using TTL

ABSTRACT

Digital circuits are used to implement logic operations based on Boolean algebra. The logic operations may be *combinational* (output depends upon present inputs only), *sequential* (output depends upon present and/or past inputs and outputs), or both. This lab experiment examines a particularly popular logic "family" called Transistor-Transistor Logic, or TTL for short. The experiment considers the typical input/output characteristics of combinational digital logic elements in general and the I/O performance of TTL in particular. *This introduction is continued in Lab # 4.*

INTRODUCTION AND THEORY

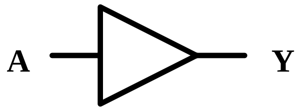
An ideal binary logic circuit element operates with only two signals: logic '0' and logic '1'. These signals are typically represented as two distinct voltage levels, often a low voltage indicating '0' and a high voltage indicating '1' (*positive* logic convention), or a low voltage meaning '1' and a high voltage meaning '0' (*negative* logic convention). The positive logic convention will be assumed here. In either case the most fundamental characteristic of digital logic is that the two voltage levels must be detected properly by the circuit inputs and generated properly by the circuit outputs.

Basic Combinational Logic Functions

Elementary logic functions comprise eight basic Boolean operations, or *gates*, as depicted here. For each gate the output (Y) is given in the form of an equation, a truth table, and a standard logic symbol.

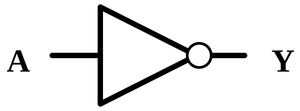
- BUFFER (Y=A):

A	Y
0	0
1	1



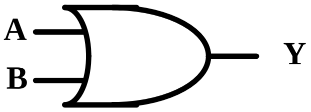
- INVERT ($Y=\overline{A}$):

A	Y
0	1
1	0



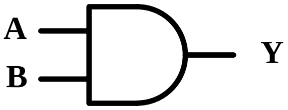
- OR ($Y=A+B$):

A	B	Y
0	0	0
0	1	1
1	0	1
1	1	1



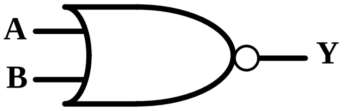
- AND ($Y=A \cdot B$):

A	B	Y
0	0	0
0	1	0
1	0	0
1	1	1



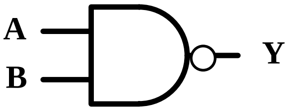
- NOR ($Y=\overline{A+B}$):

A	B	Y
0	0	1
0	1	0
1	0	0
1	1	0



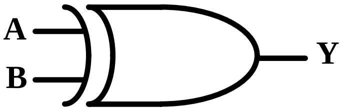
- NAND ($Y=\overline{A \cdot B}$):

A	B	Y
0	0	1
0	1	1
1	0	1
1	1	0



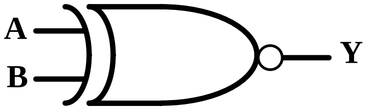
- XOR ($Y=A \oplus B$):

A	B	Y
0	0	0
0	1	1
1	0	1
1	1	0



- XNOR ($Y=\overline{A \oplus B}$):

A	B	Y
0	0	1
0	1	0
1	0	0
1	1	1



These basic logic operations can be grouped into sets that implement every possible multiple-input truth table, or *switching function*. In other words, we can find subsets of the basic gates which can be combined in various ways to implement all the other functions. Such a set of logic operations is referred to as *functionally complete*. Some examples of functionally complete sets of logic operators are:

{AND, OR, NOT}
{AND, NOT}
{OR, NOT}
{OR, XOR}
{NOR}
{NAND}
...etc...

Notice that the NOR and NAND operators are functionally complete. Functionally complete sets can also allow any otherwise unused gates (in an integrated circuit containing more than one gate) to be combined in useful ways. This fact is also used in practice to reduce the number of different kinds of gates needed in a specific design, thus reducing inventory and parts cost. Incidentally, the use of NAND and NOR logic is particularly important in modern integrated circuit design because of the simple and compact way in which these functions can be implemented in actual integrated circuit layouts.

Digital Logic and TTL

As mentioned above, it is necessary to define the proper operating signals representing logic '0' and logic '1'. Moreover, it is necessary to develop a set, or *family*, of *standard and compatible* logic circuits to implement the required Boolean functions. One such family is TTL, or T^2L, (transistor-transistor logic), which is implemented in integrated circuit (IC) form. The internal circuitry of TTL will not be considered until subsequent courses but the principles of operation can be described here in elementary terms.

The basic combinational logic functions such as AND and OR are sometimes referred to as *small-scale integration* (SSI) since only a dozen or so internal circuit elements are necessary to implement the required operation. Medium-scale (MSI), Large-scale (LSI), Very-large-scale (VLSI), and Ultra-large-scale (ULSI) integration loosely correspond to integrated circuits containing 10's, 100's, 1,000's and 10'000's of gates, respectively.* Although most modern digital systems (microcomputers, calculators, communications

---

* Perhaps JDRI-LSI (Just Down Right Incredibly Large Scale Integration) will be next?!

circuits, etc.) involve highly integrated components such as microprocessors and gate arrays, design using basic logic gates is still needed in many instances to interconnect, or 'glue', the VLSI and ULSI ICs together. For this reason, the use of simple logic gates is often referred to as *glue logic*.

Logic integrated circuits such as TTL are described by several parameters.

- Nominal output high and low voltages and currents: V_{OH}, V_{OL}, I_{OH}, I_{OL}
  These are the normal output voltages of the logic circuit in the high and low logic states. Negative currents indicate flow *out* of the output, while positive currents indicate flow *into* the output.

- Nominal input high and low currents: I_{IH}, I_{IL}
  These are the normal input currents with the input voltages of the logic circuit in the high and low logic states. A negative value indicates current flowing *out* of the input, and a positive value indicates current flowing *into* the input.

- Nominal input transition voltages: V_{IH} and V_{IL}
  The transition voltages specify the range of voltages recognizable by the gate inputs, i.e., an input voltage above V_{IH} is interpreted as 'high', while an input voltage below V_{IL} is interpreted as 'low'. Any voltage in between V_{IH} and V_{IL} is undefined.

- Noise margins: NM_H and NM_L
  The so-called 'noise margins' refer to the difference between the normal output high and low voltages and the nominal input transition voltages ($NM_H = V_{OH} - V_{IH}$; $NM_L = V_{IL} - V_{OL}$). The larger the noise margin the greater the circuit immunity from errors in determining the correct signal voltage level.

- Logic swing
  This is the difference between the normal output high and output low voltages ($V_{OH} - V_{OL}$). A large logic swing makes level determination easier, but requires more delay when changing the output from one level to the other.

- Fanout
  The fanout specifies how many gate inputs can be connected to ('driven' by) a single gate output. This is almost always given as a worst-case (minimum) value.

- Power dissipation

The power required by each gate under *standby* conditions. That is, the amount of power required to hold a constant output level. Additional power is required when the gate output changes from high to low or low to high.

- Propagation delay
  The amount of time between a level transition at the input of a gate and the corresponding output level change. Propagation delay is normally the *average* of the high-to-low and low-to-high propagation delays, which may be different.

Many other specifications are given for TTL logic circuits in manufacturer's data books and other references.

The signal levels for *standard* TTL circuits are summarized in the following table.

Parameters	Typical Values	Worst-case Values
V_{OH} , V_{OL}	3.5V, 0.2V	2.4V, 0.4V
V_{IH}, V_{IL}	1.5V, 0.5V	2.0V, 0.8V
NM_H, NM_L	2.0V, 0.3V	0.4V, 0.4V
Logic Swing	3.3V	2.0V
I_{OH}, I_{OL}	--	-0.4mA, 16mA
I_{IH}, I_{IL}	--	40µA, -1.6mA
Fanout	--	10
Power Dissipation (per gate)	--	10mW
Propagation Delay	--	10ns

Note that the TTL inputs *take in* (sink) current in the input-high state, and *put out* (source) current in the input-low state, while the outputs source current in the high state and sink current in the low state. The direction of the current at the inputs and outputs is indicated by the following sign convention: negative current means a current coming *out* of the specified terminal and positive current means current going *into* the specified terminal.

TTL ICs operate with a single +5V power supply. Most TTL circuits are available in standard *dual in-line packages*, or DIPs. The internal circuit connections ('pin-outs') of several standard TTL circuits are included with this experiment.

TTL devices are identified by standard part numbers beginning with 74- or 54-. For example, the 7400 is a *Quad 2-Input NAND* integrated circuit, meaning that the 7400 DIP contains 4 (quad) separate NAND gates, each with 2 inputs. The 74-series are devices guaranteed to operate over the *commercial temperature range* (0° to +70° C), while the 54-series are premium devices used in applications requiring the *military temperature range* (-55° to +125° C). The 74-series are, of course, less expensive and more readily available than the military-grade 54- devices. *Note that the 74- and 54- devices often have different pin-outs so they are not directly interchangable in a given circuit*. Typical TTL chips cost perhaps 50¢ and up.

In addition to the standard TTL ICs first introduced in the mid 1960's, new versions of the TTL family with improved performance have appeared over the years. The new versions are indicated by one or more letters placed between the '74' or '54' family number and the part identifier, e.g., 74LS00 or 74HC00. *A few* of the many TTL-compatible versions and their characteristics are given here.

S - *Schottky TTL*: Faster than normal TTL (propagation delay ~ 3ns), but higher power consumption (~ 20mW per gate).

LS - *Low-power Schottky*: Same speed as standard TTL but lower power dissipation (~ 2mW). LS-TTL has been the most popular version of TTL for many years.

AS - *Advanced Schottky*: Improved speed over S-TTL (~ 1.5ns, ~ 20mW).

ALS - *Advanced Low-power Schottky*: Improved speed and power over LS-TTL (~ 4ns, ~ 1mW)

HC - *CMOS*: Devices implemented using complementary metal-oxide-semiconductor field-effect transistors (CMOS FETs). HC- circuits perform similarly to LS-TTL but with power dissipation of only 2.5µW per gate.

Other types include L, F, C, HCT, AC, ACT, etc. Mixing circuits from different TTL versions is usually possible but care must be taken to ensure that the proper voltage levels and drive currents are available. For example, an LS-TTL circuit output has sufficient current capability to drive up to 20 LS-TTL inputs (fanout=20), but LS-TTL can drive only 5 standard TTL inputs.

It is common to include *power supply bypass capacitors* in TTL circuit designs. Capacitors are charge storage elements which will be discussed in more detail later in this course. *Bypass* refers to the good design practice of placing capacitors across the power supply connections to help stabilize the DC power supply voltages and "bypass" any noise or interference on the supply lines to ground. Although bypass 'caps' are not always necessary in small, simple circuits on a logic breadboard, they are important in stand-alone systems involving multiple ICs and circuit boards.

TTL bypass capacitors (typically 0.01µF ceramic disc capacitors) are connected between the positive supply voltage and the circuit ground. The bypass capacitors must have short leads and be placed as close to the ICs as possible. General rules of thumb indicate that one bypass capacitor should be used for every four SSI gate packages or every two MSI packages. Additional bypass capacitors should be used if the ICs are separated by more than two or three inches. A 10µF tantalum electrolytic capacitor is normally used between the power supply line and ground at the edge of the circuit board where the +5V power is attached.

Any unused TTL *input* pins should never be left *floating* (unconnected) because unterminated inputs are susceptible to noise, 'ringing', and output switching transients. Instead, all unused inputs should be connected to the +5V supply. Unused TTL *outputs* can be left floating.

REFERENCES

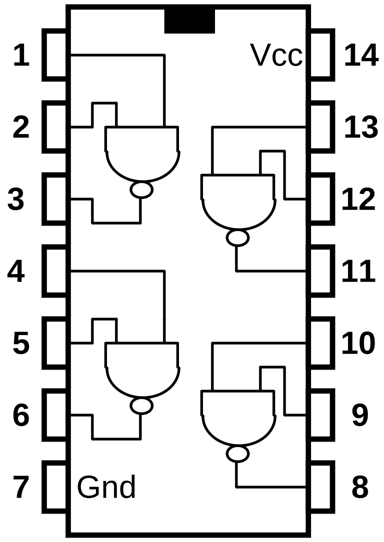
Sedra and Smith, *Microelectronic Circuits*, 3rd ed., Holt, Rinehart, and Winston, 1991.

Don Lancaster, *TTL Cookbook*, Howard W. Sams Co., 1991.

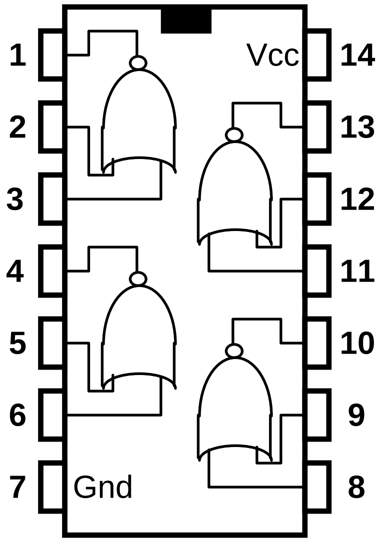
See also manufacturer's data books, such as the Texas Instruments *TTL Data Book for Design Engineers*, to find specifications and applications information. Data books are available in the IEEE Lounge and the Engineering Library.

Pin diagrams (top views) for several standard TTL ICs are shown below (there are *hundreds* of others!). Note that the pin numbers begin at the notched end of the DIP and proceed counter clockwise. Sometimes the manufacturer places a dot near pin 1 to aid in identifying the correct orientation.

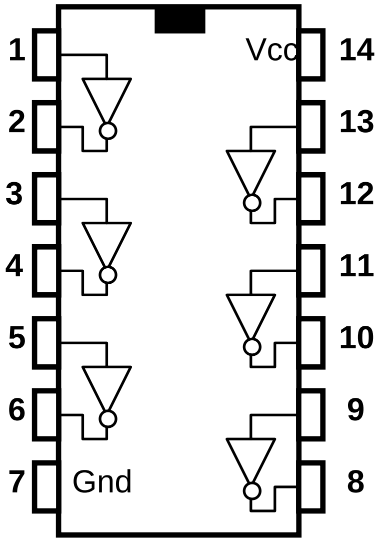
7400
Quad 2-Input NAND



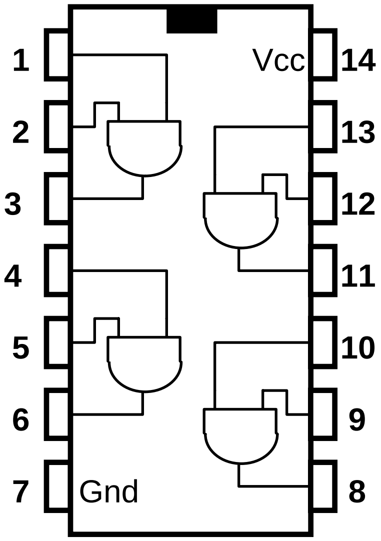
7402
Quad 2-Input NOR



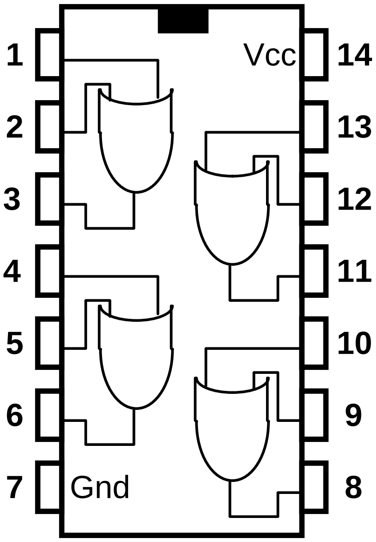
7404
Hex Inverter



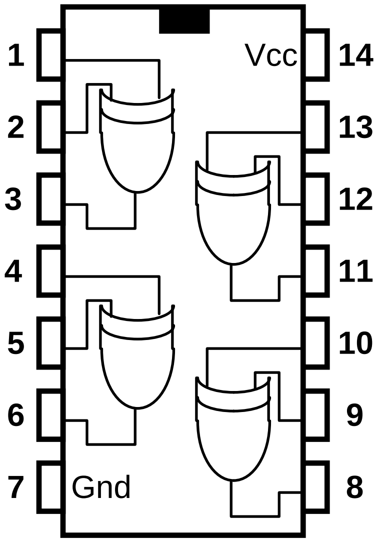
7408
Quad 2-Input AND



7432
Quad 2-Input OR



7486
Quad 2-Input XOR



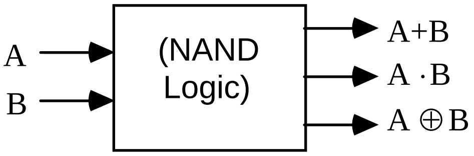
EQUIPMENT

TTL components
Heathkit Trainer
Multimeter
Oscilloscope

PRE-LAB PREPARATION

(I) It is sometimes useful to implement the basic combinational functions using combinations of other functions (recall *functional completeness*). For example, it may be possible to minimize the total number of ICs required in a design by making use of ostensibly unused gates. *Using only 2-input NAND gates*, show how each of the seven other basic combinational logic functions can be implemented.

(II) It is obvious that we can simultaneously generate 2-input AND, OR, and XOR functions using three chips: 7408 ($0.35 each), 7432 ($0.35 each), and 7486 ($0.45 each). However, in this simple example only one gate is used on each chip and the total cost is $1.15. Instead, show how the 2-input logic functions AND, OR, and XOR can be implemented simultaneously (as depicted below) with only two 7400 ICs ($0.29 each), for a total cost of $0.58.



(III) Imagine you work for an amusement park. A very simple safety module is needed for the "Super LaLaPalooza WhizBanger" thrill ride. The module has three inputs: [A]= seat-occupant sensor (1=occupied, 0=unoccupied), [B]= seatbelt sensor (1=belt fastened, 0=unfastened), and [C]= an operator-ready button (1=operator ready, 0=operator not ready). The ride must operate *only* when (i) the operator is ready AND the seat is unoccupied, or (ii) when the seat is occupied AND the belt is fastened AND the operator is ready. Your circuit must generate a '1' output only in these cases and a '0' in all other cases to prevent the ride from operating in an unsafe manner.

(a) Give the truth table and Boolean function for the circuit. You may indicate any 'don't care' conditions with an 'X'.

(b) Design the circuit using basic logic functions. Simplify the circuit as much as possible to minimize the required number of gates.

(c) Now show how the design can be implemented using only ICs selected from the 74-series TTL parts shown previously ('00, '02, '04, '08, '32, '86). How many different 74xx part numbers are required for your design of part (b)? How many individual chips are required? What is the total number of IC pins? Can you design a circuit that <u>minimizes the number of chips</u> required?

(IV) A standard 74-series TTL gate *input* requires 40µA of current in the high state, and 1.6mA of current in the low state. The standard 74-series TTL gate *output* can provide 0.4mA in the high state and 16mA in the low state. Note that in either case the fanout (number of inputs driven by a single output) is 10. A 74ALS-series gate *input* requires 20µA in the high state and 0.2mA in the low state, while its output can provide 0.4mA in the high state or 4mA in the low state.

(a) Determine the fanout of a standard TTL output driving a 74ALS input. Choose the worst case (minimum) by checking both the high and low states.

(b) Determine the fanout of a 74ALS output driving a standard TTL input. Again, choose the worst case.

(V) LOOKUP a data book for the 74LS family of TTL chips and record the nominal values for the terminal voltages and currents and for the propagation delay times. You will need to compare these to your measurements in the lab!

EXPERIMENT

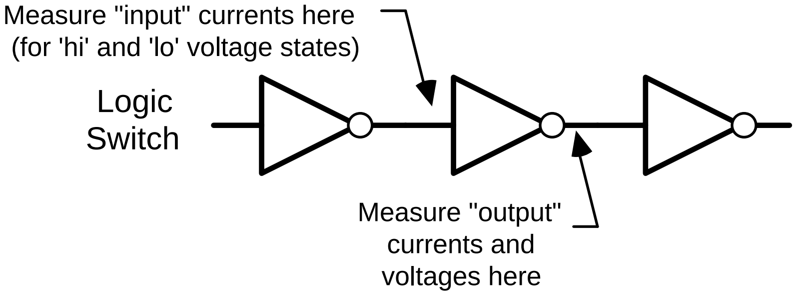
This experiment uses the Heathkit powered breadboard "trainers". The trainer has an internal power supply system with ±12 volt, +5 volt, and 0 volt (ground) outputs. We will use the +5 volt and ground outputs to power the TTL circuits. *When working with electronic devices ALWAYS assemble and verify the circuit with the power OFF. Once the circuit has been checked, then apply the power.* ICs can be damaged by incorrect voltage connections. Work carefully and methodically.

(1) Use two '00 ICs to implement simultaneously the combinational logic functions OR, AND, and XOR from part (II) of the pre-lab. Assemble your circuit on the Heathkit trainer breadboard. Use two of the data switches for the A and B inputs, and observe the output states using the light emitting diodes (LEDs).
Demonstrate your circuit for the lab TA.

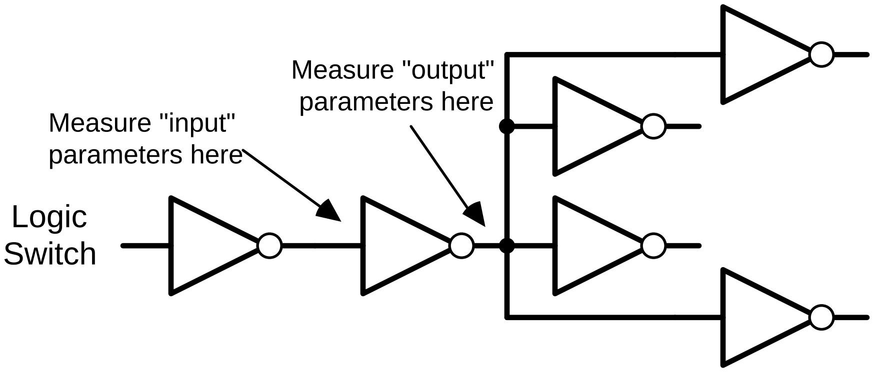
(2) Assemble the safety control circuit you designed in part (III) of the pre-lab. Use the data switches for the three input signals and observe the output using one of the LEDs. Verify the operation of the circuit using all possible input combinations and demonstrate the results for the TA.

(3) Connect three inverters ('04) in series and measure V_{OH}, V_{OL}, I_{IH}, I_{IL}, I_{OH}, and I_{OL} under these conditions for the middle inverter, as indicated below. *NOTE* that the

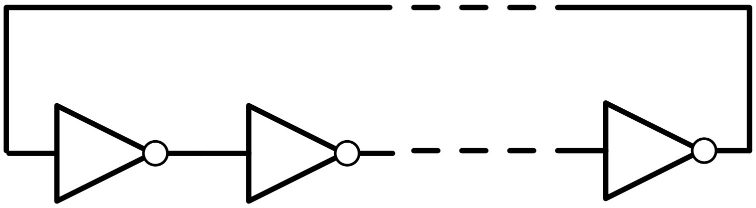
direction of the current has a defined meaning! Apply the required input signal using one of the data switches on the trainer.



(4) Now connect four load inverters to the middle inverter, as shown next, and repeat the measurements of part 3.



(5) One way to estimate the propagation delay for a single inverting gate is to connect several gates in a loop, measure the oscillation period, and divide by the number of gates. Use two 7404 ICs to make an oscillator loop with as many gates as possible (do you need an odd or even number of inversions?). Use the oscilloscope to observe the oscillation anywhere in the loop, then determine the period of oscillation and the propagation delay per gate.



RESULTS

(a) Show the circuit designed for part 1 of the experiment. Did it operate?

(b) Describe your design for the circuit of part 2. How did you attempt to minimize the design? How many chips did you need? Did the circuit behave as expected?

(c) Present your measurements from part 3. Compare the results you obtained to the expected values from TTL databooks. Did your measurements fall within the expected limits?

(d) Present your measurements from part 4. Compare the results to part 3, and discuss any differences. Do the measurements correspond to your expectations?

(e) Describe your measurements from part 5. How do the results compare to the expected propagation delay from a databook?

(f) In what ways should this experiment be changed?

Revised 7/93

Lab # 4
TITLE: Introduction to Sequential Digital Circuits Using TTL

ABSTRACT

Sequential digital circuits exhibit some form of *memory*: the present output depends upon both the current and previous input and output values. This allows digital operations involving the order in which the circuit inputs are applied. Transistor-transistor logic (TTL) integrated circuits include a number of sequential building blocks that are considered in this experiment. *This introduction is continued from Lab # 3.*

INTRODUCTION AND THEORY

The basic sequential circuit element is the *flip-flop*. We will define the flip-flop as a logic element with one or more inputs and two outputs, Q and $\overline{Q}$. The Q output is the *true* output, while the $\overline{Q}$ output is the *false* or *complementary* output, i.e., if Q=1, $\overline{Q}$=0, and vice versa. Changes in the output levels, or *states*, occurs either *asynchronously* or *synchronously* depending upon the design of the flip-flop circuit. Asynchronous flip-flops can change state whenever the inputs change state, while synchronous flip-flops operate with an additional input signal, the *clock*, which allows state transitions only during a specific portion of the clock signal. Flip-flops with synchronous operation will be assumed in this experiment.

The clock signal is typically a pulse waveform. The time between pulses is the *clock period* and the frequency of the pulses is referred to as the *clock rate*.

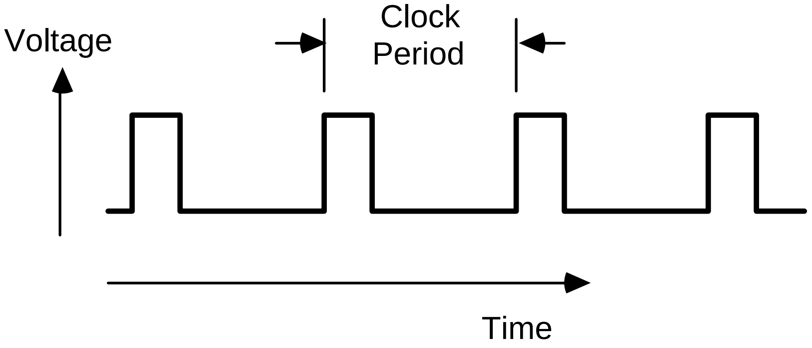


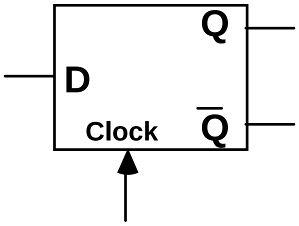
Figure 4.1: Example of a Clock Signal

Flip-flop Types

There are four basic types of flip-flops: D, T, SR, and JK. The behavior of these flip-flops is typically represented in the form of *signal state tables*, as shown here

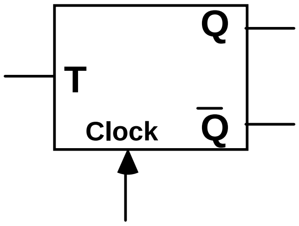
D Flip-flop

Present State	Next State	
	D = L	D = H
Q = L	L	H
Q = H	L	H



T Flip-flop

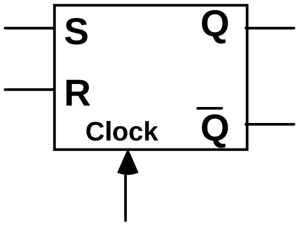
Present State	Next State	
	T = L	T = H
Q = L	L	H
Q = H	H	L



SR Flip-flop

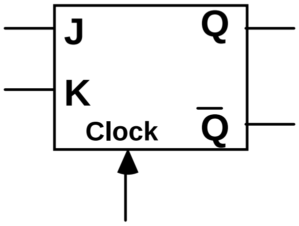
Present State	Next State			
	S = L R = L	S = L R = H	S = H R = L	S = H R = H
Q = L	L	L	H	†
Q = H	H	L	H	†

†: undefined conditions



JK Flip-flop

Present State	Next State			
	J = L K = L	J = L K = H	J = H K = L	J = H K = H
Q = L	L	L	H	H
Q = H	H	L	H	L



The signal state tables above indicate the *new* output state after the next clock signal, given the specified present state and present input conditions. Note that the JK flip-flop is similar to the SR flip-flop, except the JK flip-flop allows both inputs to be in the high state simultaneously: this condition results in the next output being the complement of the present output.

Another common way to specify flip-flop operation is a *transition table*. The transition table indicates the input conditions required to cause the specified sequence of states. This information is useful when designing sequential networks because we often need to determine which input combination is necessary to make the *next* state of the flip-flop something specific. Transition tables for the basic flip-flops are given below ('d' indicates *don't care*):

T Flip-flop

Present State	Next State	T
0	0	0
0	1	1
1	0	1
1	1	0

D Flip-flop

Present State	Next State	D
0	0	0
0	1	1
1	0	0
1	1	1

JK Flip-flop

Present State	Next State	J	K
0	0	0	d
0	1	1	d
1	0	d	1
1	1	d	0

SR Flip-flop

Present State	Next State	S	R
0	0	0	d
0	1	1	0
1	0	0	1
1	1	d	0

State Tables and State Diagrams

More complicated sequential logic circuits can be created by combining several flip-flops together. Since each flip-flop can represent two distinct states, *N* flip flops together can represent 2^N distinct states. The behavior of these multi-flip-flop networks is conveniently summarized in a *state table*. The state table indicates the *present* state and the *next* state and output(s) for every combination of inputs. For example, consider an arbitrary sequential logic circuit with 4 states (S), 3 inputs (I), and 2 outputs (Y). An *example* state table might be:

Present State	Input States		
	I_1	I_2	I_3
S_1	S_2/Y_1	S_1/Y_2	S_1/Y_1
S_2	S_2/Y_2	S_3/Y_2	S_4/Y_2
S_3	S_1/Y_2	S_2/Y_1	S_3/Y_2
S_4	S_1/Y_1	S_1/Y_2	S_4/Y_1

This example state table indicates, for instance, that if the present state is S_3 and the present input is I_2, then after the next clock signal the state will be S_2 and the output will be Y_1.

A state table can also be represented graphically as a *state diagram*. The state diagram corresponding to the *example* state table is shown in Figure 4.2.

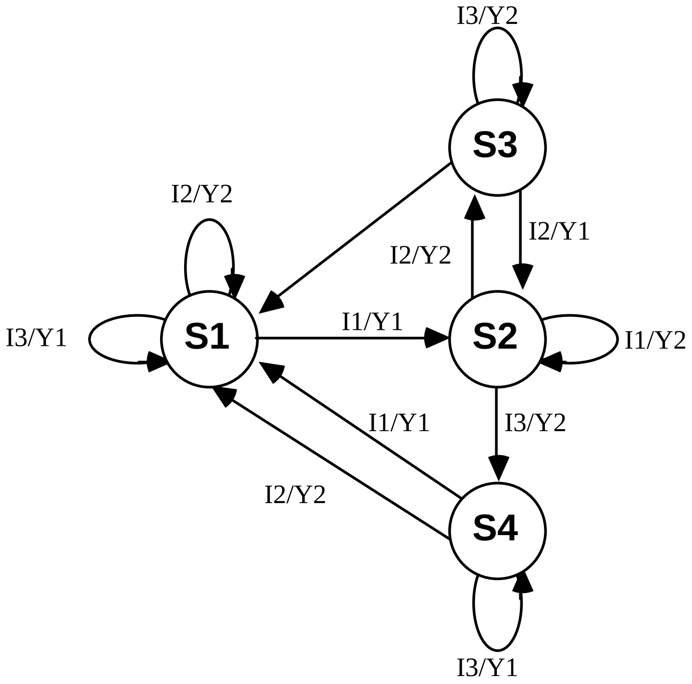


Figure 4.2

The circles represent each of the four states, while the arcs (arrows) represent the state transitions. Each arc is labeled *Input/Output*, indicating the *input required to make the state transition and the output value after the transition*. Arcs which begin and end on the same state circle indicate that no state transition occurs with the specified input.

A special example of a sequential circuit is a *counter*. The state diagram for a counter is a loop: each present state has a transition to one and only one next state. For example, the state diagram for a two-bit binary counter is:

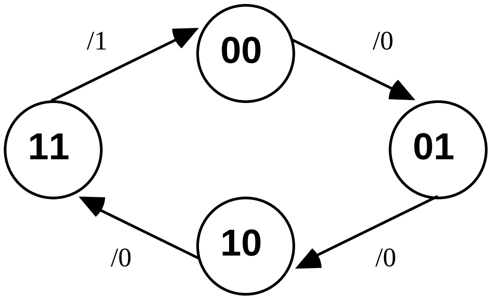


Figure 4.3

In this simple example the counter changes state on every clock input, so no input conditions need to be specified. This example counter has one output signal, however, which is set to '1' when the counter reaches '00' and set to '0' otherwise. *Note* that counters can be specified in general with arbitrary sequences of states, i.e., it is not necessary to follow a binary count sequence from state to state.

Converting a State Table to a Flip-flop Logic Circuit

A general sequential network can be represented as shown in Figure 4.4. Note that the network involves *inputs*, *outputs*, *flip-flops*, and *combinational logic*.

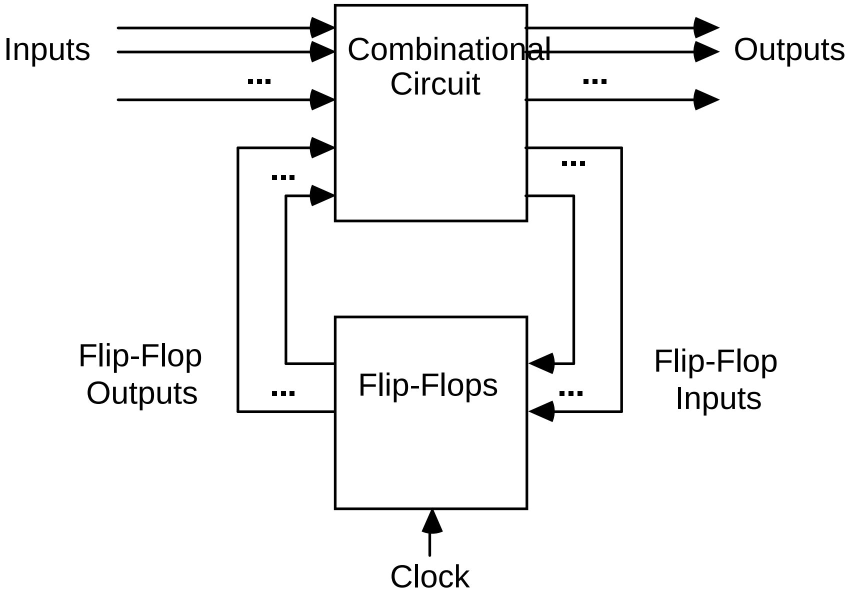


Figure 4.4

Consider the simple two-bit binary counter discussed previously, which we now wish to implement using JK flip-flops. In this example the network diagram reduces to the circuit shown in Figure 4.5.

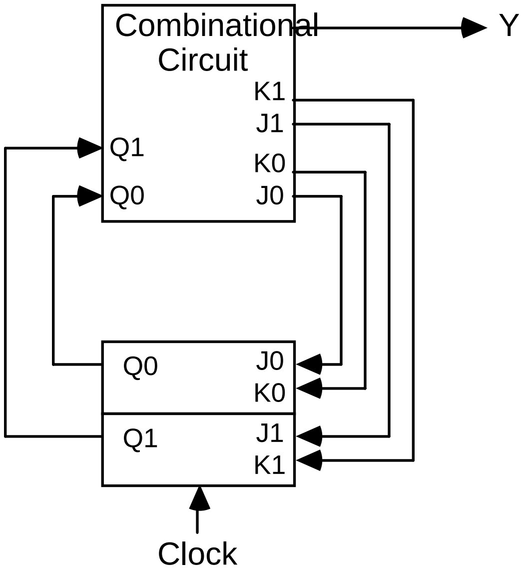


Figure 4.5

Thus, we need to <u>design a combinational circuit that takes the present state</u> [Q_1, Q_0] <u>and produces the required flip-flop inputs,</u> J_0, J_1, K_0, and K_1<u>, and output signal, Y</u>. This process can be accomplished by writing truth tables for the required output and flip-flop inputs. For this example the extended truth tables are:

Present State	Output
Q_1 Q_0	Y
0 0	1
0 1	0
1 0	0
1 1	0

Note that the output Y is $\overline{Q_0 + Q_1}$ (NOR). The JK flip-flop inputs are chosen so that the flip-flop outputs (Q_0 and Q_1) follow the proper transition sequence. Comparing the required JK values (and don't care inputs) to the present state values reveals that

$J_0 = \overline{Q_0}$, $K_0 = Q_0$
$J_1 = Q_0$, $K_1 = Q_0$,

which is very nice because the function $\overline{Q_0}$ is already available if the JK flip-flop has complementary outputs. So in this simple example the binary counter network reduces to the circuit of Figure 4.6.

Present State	Next State	F-F #1 Input	F-F #0 Input
Q_1 Q_0	Q_1 Q_0	J_1 K_1	J_0 K_0
0 0	0 1	0 d	1 d
0 1	1 0	1 d	d 1
1 0	1 1	d 0	1 d
1 1	0 0	d 1	d 1

d= don't care

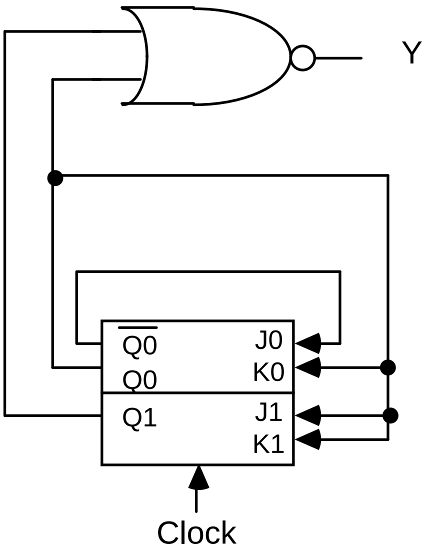


Figure 4.6

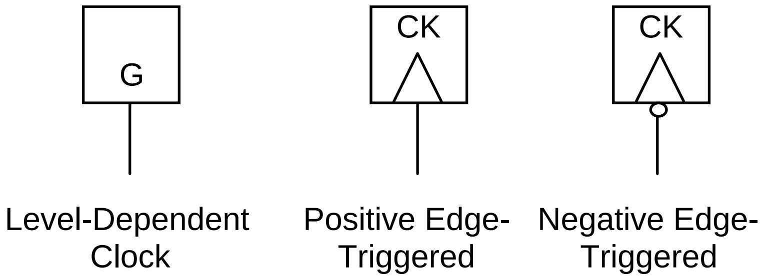
A similar procedure is followed if D, T, or SR flip-flops are utilized. Note that the minimization process on the combinational logic is often more complex than in this example, but Karnaugh maps or other techniques can be employed directly.

Sequential TTL Circuits

MSI circuits that implement D and JK flip-flop functions are available in TTL form. The TTL flip-flops have similar voltage and current characteristics as the TTL combinational gates described in the previous experiment. Several example chip pinouts are included in the References section.

Some TTL flip-flops are controlled by a *level dependent clock*, meaning that the flip-flop is able to change state as long as the clock signal is in the 'high' state. A level dependent clock acts simply as an *enable* input, so flip-flops with level-dependent clock control are sometimes referred to as *latches*.

Most TTL flip-flops are *edge-triggered*, which means that the output transition is initiated by either the rising edge or falling edge of the clock signal. Flip-flops that are clocked by the rising edge are called *positive edge triggered*, and those clocked by the falling edge are called *negative edge triggered*. Edge-triggered flip-flops require the inputs to be held constant (at an unchanging '0' or '1' level) for a short interval prior to the clock edge (the *setup time*), and remain constant for a short interval after the clock edge (the *hold time*). In simple logic circuits with only a few gates it is usually trivial to meet the setup and hold time specifications, but complicated circuits at high clock rates must be designed to ensure the proper signal timing if reliable operation is desired (and it usually is!). The common schematic symbols for clock inputs are shown below.



Level-Dependent Clock

Positive Edge-Triggered

Negative Edge-Triggered

Figure 4.7

TTL flip-flops often contain additional features. For example, the 7474 IC is a *Dual D-type Positive-Edge-Triggered Flip-Flop with Preset and Clear*. This means that the 7474 contains two D flip-flops with positive edge triggering, and two additional inputs: a *preset* signal which forces the flip-flop output to the '1' state, and a *clear* signal which forces the output to the '0' state. Both the preset and clear signals of the 7474 are *active-low*, meaning that they perform their function when pulled to the low ('0') state. Thus, if the 7474 is to be used as a normal D flip-flop the preset and clear pins should both be connected to the positive supply voltage. When in doubt, refer to the TTL data book information.

Debugging Sequential TTL Circuits

The process of constructing and verifying an electrical circuit is a fundamental methodology of the electrical engineering art. Unfortunately, it is also a process prone to careless errors. It is very important to make the task easier for yourself by thinking ahead.

The suggested approach is to construct the circuit in *small stages*, verifying correct operation after each stage is installed. This process requires some planning to decide where to start and how to choose the best testing procedure. For example, a sequential design involving some flip-flops and a combinational network could be constructed by first assembling the combinational network and verifying its operation by manually supplying the input patterns. Next, the flip-flops would be tested, followed by testing of the complete circuit. This approach has the advantage that errors or problems can be localized to the stage where the circuit stops working properly. If the stages are chosen to be small enough, any problems can be corrected quickly.

If a circuit does not seem to work properly *always check the obvious things first*: make sure the power supply is on, double check the input and output connections, and look for any loose wires or IC pins without connections! If the problem still cannot be identified, try to *isolate* the problem to a particular chip or interconnection by checking signal voltages at each pin. Finally, *make sure the design actually implements the function you are expecting*. As a last resort, check the integrated circuit by itself to make sure the IC is not defective or damaged.

The construction and verification process is also simplified by:

(i) turning power OFF whenever changing connections or adding new wires or ICs.
(ii) keeping all interconnection wires as short as possible.
(iii) aligning all ICs so that the notches (pin 1) are in the same direction.

(iv) using some sort of color coding for specific signals (yellow wires for clock, red wires for +5 volts, etc.).

(v) using stick-on labels to identify the pinouts of each chip.

REFERENCES

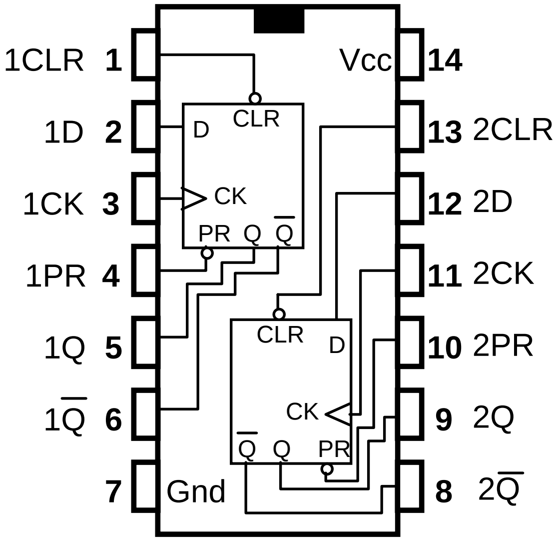
Sedra and Smith, *Microelectronic Circuits*, 3rd ed., Holt, Rinehart, and Winston, 1991.

Don Lancaster, *TTL Cookbook*, Howard W. Sams Co., 1991.

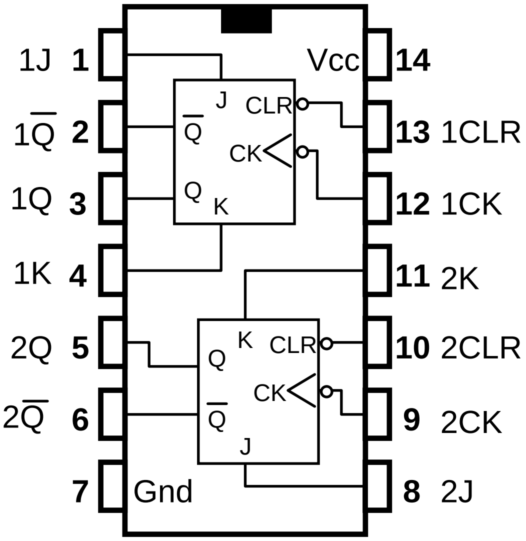
See also manufacturer's data books, such as the Texas Instruments *TTL Data Book for Design Engineers*, to find specifications and applications information. Data books are available in the IEEE Lounge and the Engineering Library.

Two example TTL circuits (D and JK flip-flops) are shown next. Other TTL flip-flop circuits are available with different features and pinouts. Refer to a TTL data book for more information. You can also investigate the cost and availability of TTL chips by looking in the Digi-Key or Jameco distributor catalogs.

74LS74
Dual D-type
Positive Edge-
Triggered Flip-Flops
with Preset and Clear



74LS107
Dual JK-type
Negative Edge-
Triggered Flip-Flops
with Clear



EQUIPMENT

TTL components Heathkit Trainer Multimeter

PRE-LAB PREPARATION

(I) It is possible to "make" a T flip-flop using a JK flip-flop. Show how the JK inputs should be connected to act like a T, including any additional external gates that are required.

(II) It is possible to "make" a D flip-flop using a JK flip-flop. Show how the JK inputs should be connected to act like a D, including any additional external gates that are required.

(III) Ordinary *binary* sequence counters have the property that more than one of the bits changes from '0' to '1' or '1' to '0' between states of the count sequence, e.g., 011 to 100, 111 to 000, etc. The various bits may require different amounts of time to change between levels, resulting in a "ragged" count transition that can cause trouble in certain high speed applications. Another type of counter organizes the bits differently so that the count order *has only one bit changing from one state in the count sequence to the next*. This type of counting order is called a *Gray code*. For example, a two-bit (4 state) Gray code count sequence could be: 00 01 11 10; 00 01 11 10; 00 ... Note that only one of the bits is different between adjacent states.

(a) Determine a three-bit (8 state) Gray code sequence.

(b) Sketch the state diagram (a loop), assuming a single output that is set to '1' when the state code is '000' and set to '0' otherwise. Also create a state table for the output and the flip-flop inputs, and then use Karnaugh maps or some other minimization technique to obtain logic expressions for the three required flip-flop input signals and the '000' output signal. Assume that *D-type flip-flops* are to be used here.

(c) Design a TTL logic circuit that implements your Gray code sequence using three D-type flip-flops (7474) and the available 2-input TTL combinational logic ICs described in Lab #3 (such as 7400, 7408, 7432, ...). Sketch a schematic of the actual TTL circuit showing all gates and required connections.

EXPERIMENT

(1) Verify the operation of the D (74LS74) and JK (74LS107) flip-flops. Refer to the pinouts in the reference section and assemble simple test circuits on the Heathkit breadboard. Use the data switches for the flip-flop inputs, the logic pulse switches for a manual clock signal, and observe the outputs using the LEDs (light-emitting diodes). You can also connect the breadboard's internal clock generator to the flip-flop to observe the outputs with repetitive clocking. Remember that the preset and/or clear inputs must be connected to the 'high' supply voltage (+5V) for normal flip-flop operation.

(2) Connect a JK flip-flop in such a way that it operates as a T flip-flop (Pre-lab part I). Verify the operation of the circuit.

(3) Next, connect a JK flip-flop (and any additional logic elements) in such a way as to act like a D flip-flop (Pre-lab part II). Verify the operation of the circuit.

(4) Now carefully assemble the 3-bit TTL Gray code sequence generator designed in part III of the pre-lab. Work steadily and meticulously to avoid wiring errors. Connect any unused TTL gate inputs to +5V. Connect the three flip-flop outputs and the single network output ('1' when state is '000', '0' otherwise) to the LEDs on the breadboard. First use one of the manual logic pulse switches as the clock signal to verify that the circuit is operating properly, then use the internal clock generator. Demonstrate your circuit for the lab TA.

RESULTS

(a) Present your circuits and results for parts 1, 2 and 3. How did your results in the lab compare to your pre-lab expectations? Did you encounter any difficulties in wiring and testing your circuits?

(b) Present your Gray code sequence generator. Explain your design and any design choices you made to minimize the complexity of the circuit or the number of components. Did you need to make any changes to your design in order to make the actual circuit work?

(c) In what ways should this experiment be changed?

Revised 4/92

Lab # 5

TITLE: Resistors: Simplification of Series and Parallel Networks

ABSTRACT

This experiment examines several useful methods for simplifying electrical circuits and networks. Investigations of *series* and *parallel* combinations of resistors are used to show voltage and current relationships. The concept of an *equivalent circuit* is also discussed.

INTRODUCTION AND THEORY

Ohm's Law indicates a linear relationship between voltage and current in a resistor. This property allows ostensibly complicated circuits to be reduced to simpler *equivalent* circuits. Equivalent circuits allow us to divide a complex network into smaller parts that are more easily described, modeled, and manipulated than the entire circuit itself.

Resistors connected in *series* must all carry the same current (Kirchhoff's current law applies at each node). The voltage across each of the resistors is given by Ohm's law: V=IR. Thus, the *total* voltage across a series of resistors is the sum of the voltages across the individual resistors.

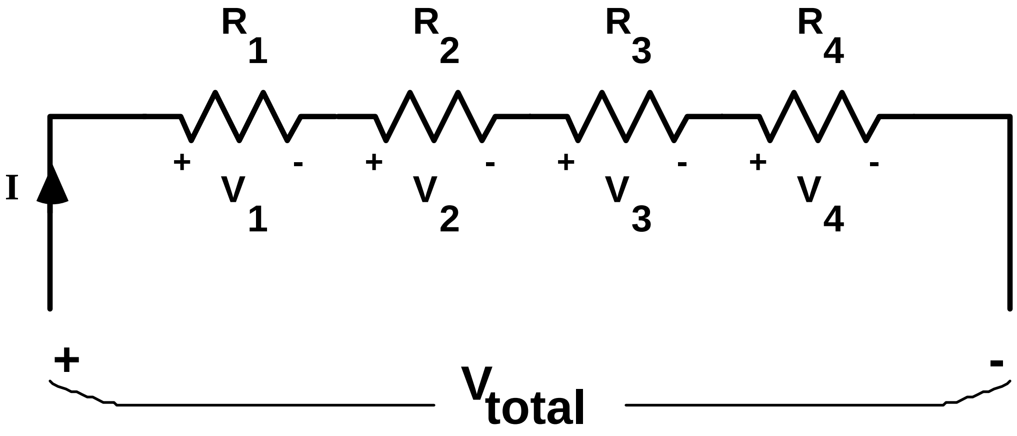


Figure 5.1

For Figure 5.1 above this yields:

$V_1 = I \cdot R_1, \quad V_2 = I \cdot R_2, \quad V_3 = I \cdot R_3, \quad V_4 = I \cdot R_4$

$V_{total} = V_1 + V_2 + V_3 + V_4 = I \cdot (R_1 + R_2 + R_3 + R_4)$

$R_{effective} = R_1 + R_2 + R_3 + R_4$

The equivalent circuit for a set of series resistors is simply the *sum* of the resistances.

Resistors connected in *parallel* must all have the same voltage across them because they are all connected between the same node pair. The current through each resistor is known from Ohm's law: I=V/R. Thus, the total current supplying the parallel resistors is the sum of the currents in the individual resistors.

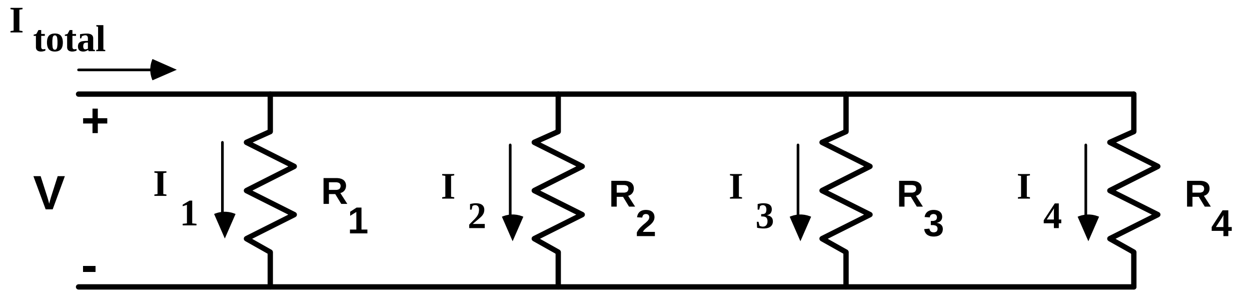


Figure 5.2

For Figure 5.2 this yields:

$I_1 = V/R_1, \quad I_2 = V/R_2, \quad I_3 = V/R_3, \quad I_4 = V/R_4$

$I_{total} = I_1 + I_2 + I_3 + I_4 = V \cdot (1/R_1 + 1/R_2 + 1/R_3 + 1/R_4)$

$R_{effective} = (1/R_1 + 1/R_2 + 1/R_3 + 1/R_4)^{-1}$

The equivalent circuit for a set of parallel resistors is the *reciprocal of the sum of the reciprocals* of the individual resistances.

A useful "rule of thumb" to remember for series and parallel resistor combinations is:

- The effective resistance of series resistors is *more than the largest resistor*.
- The effective resistance of parallel resistors is *less than the smallest resistor*.

This is just a quick way to verify that the results of an actual calculation are reasonable.

REFERENCES

See sections 2.4 through 2.6 of the text by J. David Irwin, *Basic Engineering Circuit Analysis*, 4th ed., Macmillan Publishing Co., 1993 (pp. 45-64).

EQUIPMENT

Resistor kit
DC Power Supply
Multimeter

PRE-LAB PREPARATION

(I) Simplify the network in Figure 5.3 to a single, effective resistance.

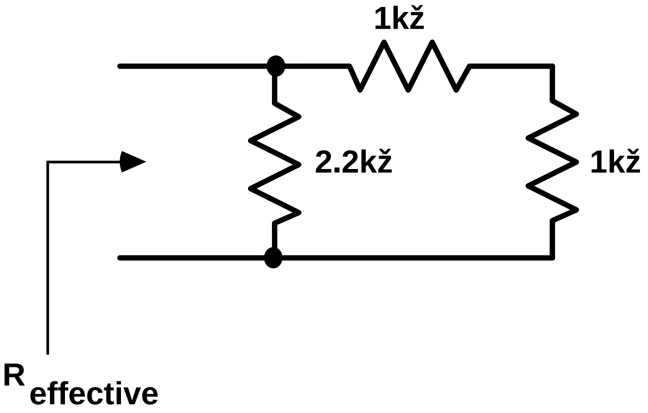


Figure 5.3

If a source of 10 volts is connected across the input to the circuit, what current will flow from the source? (*Hint*: use the effective resistance).

(II) A simple combination of two resistors in series is sometimes called a *voltage divider* because the voltage across the individual resistors depends upon the two resistor values. Using the circuit of Figure 5.4, determine a mathematical formula relating the voltage V_B (across R_B) to the applied voltage, V. (*Hint*: one way to start is to use the effective resistance to find the current).

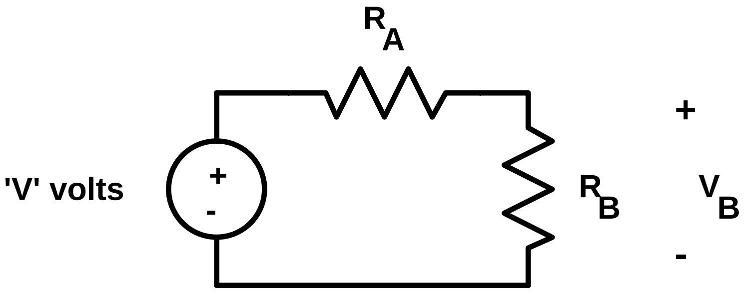


Figure 5.4

(III) If only 470Ω and 220Ω resistors happen to be available and we need a resistance of 370Ω, determine a combination of one 470Ω and two 220Ω resistors that will theoretically come within 1Ω of the desired 370Ω. Now with ±5% tolerance of the individual resistors used, what could be the maximum and minimum effective resistance of your circuit due to resistor tolerance?

EXPERIMENT

(1) Consider the parallel resistor circuit of Figure 5.2. Use the following nominal resistor values: $R_1 = R_2 = 1k\Omega$, $R_3 = 2.2k\Omega$, and $R_4 = 330\Omega$. Measure the individual resistor values using the multimeter, then construct and measure the effective resistance of the parallel circuit.

Next, adjust the bench power supply to +5V DC and attach the supply to the circuit. Measure the current conducted by each resistor.

(2) Use the multimeter to measure the value of the individual resistors for the circuit of Figure 5.3. Then construct the circuit and measure the effective resistance.

(3) Construct the voltage divider circuit of Figure 5.4 using nominal resistors $R_A = 10k\Omega$ and $R_B = 1k\Omega$ (measure and record the actual values used). Attach the signal generator in place of the voltage source 'V' and use the oscilloscope to adjust the

generator to produce a 10 volt peak-to-peak sinewave at 1kHz frequency. Observe the applied voltage from the generator and the voltage V_B simultaneously using the two input channels of the 'scope. Record the peak-to-peak amplitude of the voltage V_B.

Next, repeat the voltage measurements with $R_A = R_B = 1M\Omega$.

(4) Construct a circuit with 370Ω nominal resistance using the resistor combination determined in the pre-lab. Measure the effective resistance of your circuit and the values of the individual resistors. Note that the measured value will probably differ from 370Ω because the actual resistors are not exactly the nominal values.

RESULTS

(a) Present your measurements from part 1. Compare the measured effective resistance to the predicted value for the *nominal* resistor values, and to a prediction obtained mathematically using the measured individual resistances. Also compare the measured current in each resistor with a predicted value from Ohm's law using the measured resistor values. Explain your results.

(b) Present your measurements from part 2. Compare to predicted values using both the nominal and measured individual resistances. Discuss the credibility of your measurements and predictions.

(c) Present your measurements from part 3. Does the measured voltage follow your expectations from the voltage divider equation derived in the pre-lab? Explain any discrepancy. (*Hint:* what is the effective resistance of the oscilloscope input? How does this affect the measurement?).

(d) Explain your measurements for part 4. Does the effective resistance correspond well to the predicted value, given your measurements of the individual resistors?

(e) How would you improve this experiment?

Revised 7/94

Lab # 6
TITLE: Nodal Analysis of Simple Networks

ABSTRACT

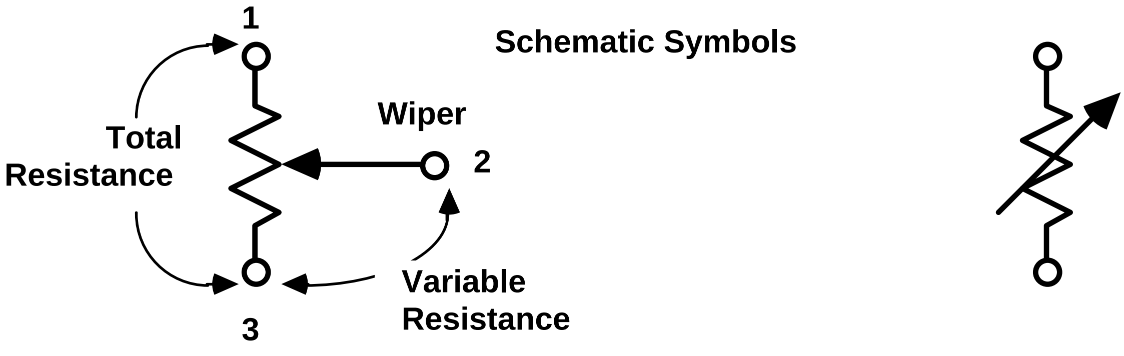
This laboratory assignment considers the application of *nodal analysis* to electrical circuits containing resistors and voltage sources. Calculated values of circuit currents and voltages are compared to measurements made in the lab. Circuit relationships involving Kirchhoff's current law (KCL) and Ohm's law are investigated.

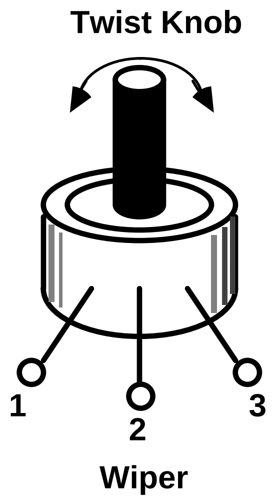
INTRODUCTION AND THEORY

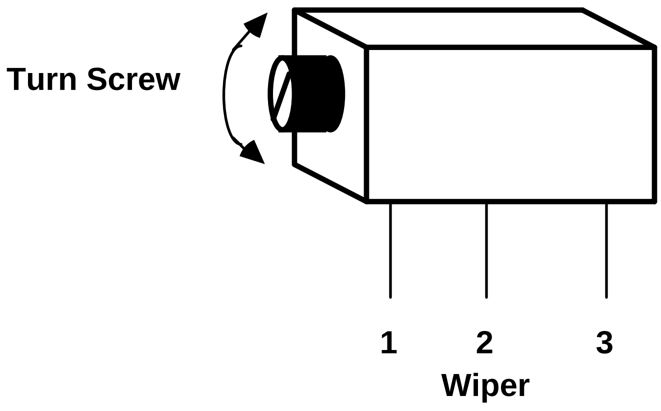
Recall that electrical voltage is a measure of the potential *difference* between two nodes in a circuit. The two nodes are referred to as a *node pair*. All node voltages in a circuit are typically specified with respect to an arbitrary reference node. The reference node is often called the *ground* node, although it may or may not actually be connected to the physical ground line in a practical circuit. The reference node is usually indicated with the ground symbol: ⏚ .

Analysis of electrical networks involves determination of node voltages and loop or branch currents. *Nodal Analysis* refers to the technique of writing equations where the unknown quantities (mathematical variables) are the node voltages of the circuit. Kirchhoff's current law is used to define the equations at each node in the circuit, using currents obtained by Ohm's law.

This lab introduces the use of *adjustable resistors*, or *potentiometers*. The resistance of a potentiometer (or 'pot') is set by mechanically moving a *wiper* across a resistive surface: the length of the resistive surface between either end and the wiper determines the resistance value. Pots are identified by the total resistance from end to end, so a 1kΩ pot provides a variable resistance from near zero up to 1kΩ. Potentiometers come in several styles, including rotary knobs, sliders, and screw-adjustments. Some common circuit symbols and styles are shown below.







REFERENCES

See section 3.1 of the text by J. David Irwin, *Basic Engineering Circuit Analysis*, 4th ed., Macmillan Publishing Co., 1993 (pp. 89-112).

EQUIPMENT

Resistor kit
1kΩ variable resistor (potentiometer)
DC Power Supply
Multimeter

PRE-LAB PREPARATION

(I) Determine the node voltages and branch currents for the circuit given in Figure 6.1. If the actual resistance values are within ±5% of the indicated values, calculate the maximum and minimum node voltages due to the resistor tolerance.

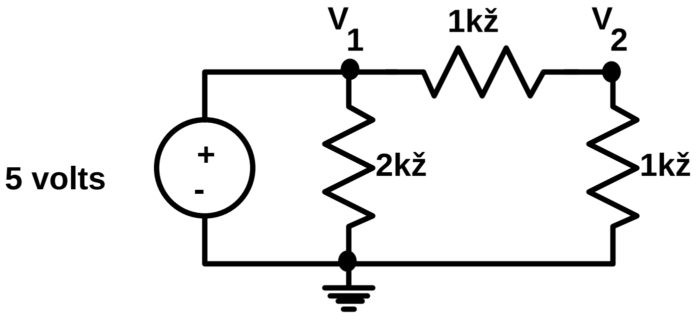
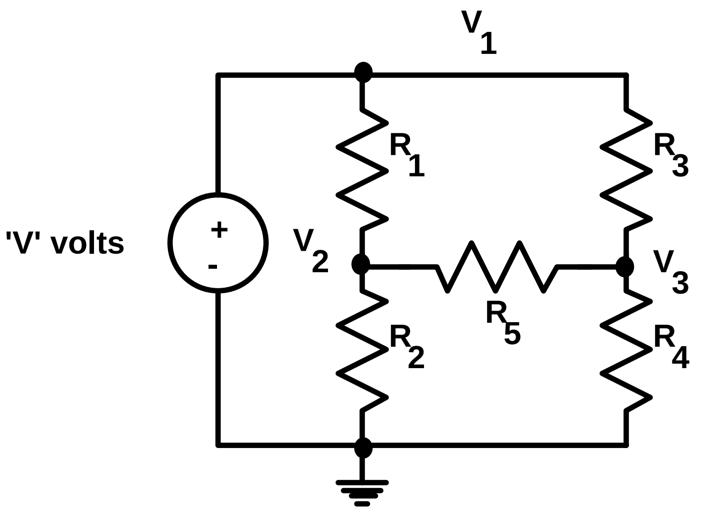


Figure 6.1

(II) Consider the circuit shown in Figure 6.2. This arrangement is called a *bridge* circuit, and is used for several instrumentation and measurement purposes. One useful property of the bridge circuit is the so-called *balance* condition that occurs when the relationship among the bridge resistors ("legs") is such that $(R_1/R_2) = (R_3/R_4)$. Note that in the balance state the node voltages V_2 and V_3 are equal, meaning that the current in R_5 must be zero. Also note that the balance condition depends only upon the resistor ratios, not the applied voltage, V.



$R_1 = 470\Omega$	$R_2 = 220\Omega$
$R_3 = 1k\Omega$	$R_4 = 470\Omega$
$R_5 = 1k\Omega$	'V' = +15 volts

Figure 6.2

The bridge circuit can be used to determine the value of an unknown resistor if several known resistors are available. For example, if the R_3 leg was an unknown resistor we could use various combinations of known resistors for the R_1, R_2, and R_4 legs until the balance condition was achieved. We would then know that $R_3 = R_4 \cdot (R_1/R_2)$. Before electronic resistance meters were available most resistance measurements had to be made with resistance bridge circuits.

Using the resistance and voltages given in Figure 6.2, determine ALL node voltages and branch currents in the circuit. Using the voltage and current values, calculate the power dissipated by each resistor.

(III) Determine a *new* value for R_4 (leaving the other resistors unchanged) that will balance the bridge circuit. Assume an adjustable resistor is used so that any resistance value can be obtained.

EXPERIMENT

(1) Construct the circuit of Figure 6.1. Measure (with the multimeter) and record the resistance of each resistor you use. Use the multimeter to adjust the power supply. Measure and record *all* node voltages and branch currents.

(2) Construct the bridge circuit of Figure 6.2, again recording the resistance of each resistor and using the multimeter to adjust the power supply voltage. Measure and record all node voltages and branch currents.

(3) Next, replace resistor R_4 with a 1kΩ variable (adjustable) resistor. Use the multimeter to measure the voltage across resistor R_5 while you slowly vary the value of R_4. Adjust the value of R_4 until the multimeter indicates zero volts across R_5. Carefully remove the variable resistor from the circuit and measure its resistance using the multimeter.

(4) Now replace R_2 with a nominal 100Ω resistor (measure and record the actual resistance). Place the variable resistor back in the R_4 position and adjust until the multimeter indicates zero volts across R_5. Carefully remove the variable resistor and measure its new value.

RESULTS

(a) Present your measurements for part 1. Compare your measurements to the predicted values obtained using nodal analysis. Include a labeled sketch of your circuit showing measured component values, voltages, and currents.

(b) Present your measurements of the bridge circuit used in part 2. Provide a labeled sketch of the circuit and your measurements.

(c) Were you able to balance the bridge using the variable resistor in parts 3 and 4? What value was required? How did your measured values compare to the predicted ratios?

(d) One possible use of the bridge circuit with a variable resistance in one leg is in position detection for mechanical systems (See Figure 6.3). How would you design a circuit that would allow an operator to place a linkage at a precise angle of 45° using a bridge circuit with a potentiometer for position feedback?

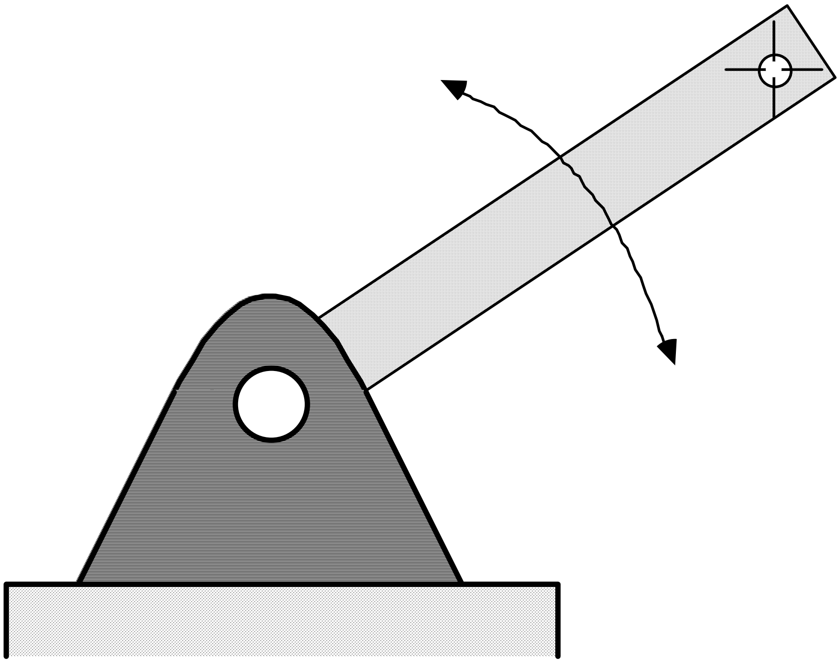


Figure 6.3

(e) What would you change in this experiment to make it better?

Revised 7/94

Lab # 7
TITLE: Loop Analysis of Simple Networks

ABSTRACT

Loop analysis of electrical circuits containing resistors and voltage sources is considered in this experiment. Calculated values of circuit currents and voltages are compared to measurements made in the lab. Circuit relationships involving Kirchhoff's voltage law (KVL) and Ohm's law are examined.

INTRODUCTION AND THEORY

Circuit equations involving Kirchhoff's current law (KCL) are used in nodal analysis to determine the unknown node voltages. In *loop analysis*, on the other hand, the unknown quantities are loop currents and the circuit equations are written using Kirchhoff's voltage law (KVL).

To begin the loop analysis procedure each loop of the network is assigned an unknown current. The direction chosen for the current in the loop is arbitrary: the calculated current will turn out to be *negative* if the actual current flows in the direction opposite to the one chosen for the analysis. The loop equations are written by applying KVL to each loop: the sum of the voltages around each loop must be zero. In any resistors shared by adjacent loops the net current in the component consists of the sum or difference of the two loop currents, depending on whether the two currents are defined flow in the same or different directions. Once the set of current equations is solved the calculated loop currents can be used to determine the voltage across each of the resistances using Ohm's law.

REFERENCES

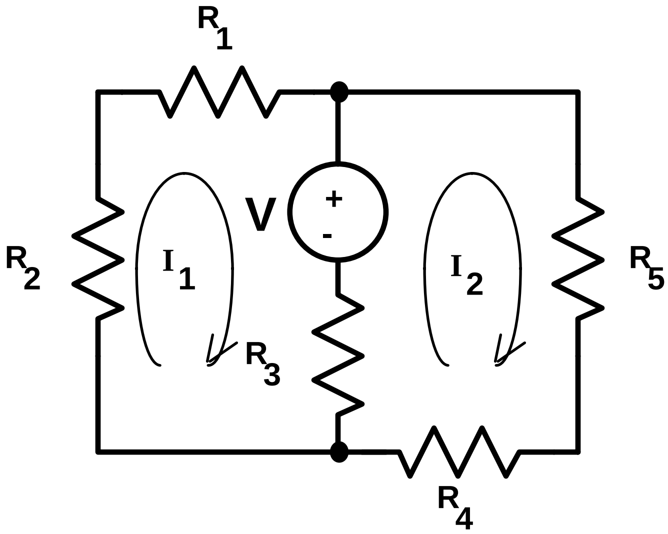
See section 3.2 of the text by J. David Irwin, *Basic Engineering Circuit Analysis*, 4th ed., Macmillan Publishing Co., 1993 (pp. 113-126).

EQUIPMENT

Resistor kit
DC Power Supply
Multimeter

PRE-LAB PREPARATION

(I) Use loop analysis to determine the loop currents and branch currents for the circuit of Figure 7.1 using hand analysis (calculator). Also calculate the resistor voltage and the power dissipation for each resistor. Verify your results using a symbolic mathematics computer program (such as MAPLE).

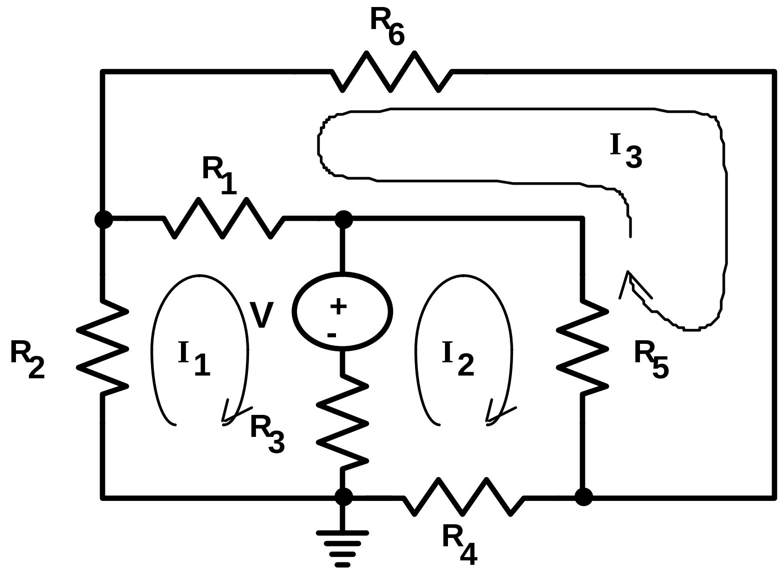


R_1 = 10ž	R_2 = 100ž
R_3 = 100ž	R_4 = 1kž
R_5 = 330ž	'V' = +10 volts

Figure 7.1

(II) What is the variation in the current through R_1 if R_5 varies ±10% around 330Ω? (Assume the other resistors (R_1 thru R_4) do not vary).

(III) Determine all loop currents and branch currents for the circuit of Figure 7.2 using a symbolic math program. Note that the circuit is the same as Figure 7.1 except resistor R_6 = 1kΩ has been added to the circuit. What is the current supplied by the voltage source? What is the power supplied by the voltage source?



R_1 = 10ž	R_2 = 100ž
R_3 = 100ž	R_4 = 1kž
R_5 = 330ž	R_6 = 1kž
	'V' = +10 volts

Figure 7.2

EXPERIMENT

(1) Assemble the circuit of Figure 7.1 using the specified nominal resistors (measure and record the actual values!). Use the multimeter to adjust the power supply for 10 volts output. Measure the current in each branch of the circuit, i.e., the current in R_1 & R_2, in R_3, and in R_4 & R_5.

(2) Now replace R_5 with *two 330Ω resistors in parallel* and re-measure the current in each branch of the circuit. Next, replace R_5 with *three 330Ω resistors in parallel* and measure the branch currents again. Be sure to measure and record the actual resistance of the nominal 330Ω resistors use use.

(3) Assemble the circuit of Figure 7.2. Note that the circuit is the same as Figure 7.1 except resistor R_6 = 1kΩ has been added to the circuit. Measure the current in each resistor of the circuit. Also measure the voltage of each node in the circuit with respect to the labeled ground reference node.

RESULTS

(a) Present your measurements from part 1. What are the measured loop currents? Compare the measured branch currents from part 1 to a mathematical prediction using loop analysis for both the nominal (pre-lab) and the measured resistor values. Calculate the power dissipated in each resistor and the total power dissipation. Compare this to the power (volts x current) delivered by the 10 volt supply. Are they equal? Is the power dissipation in each resistor less than its 0.25 watt power rating?

(b) Present your measurements from part 2. What was the measured resistance for the two nominal 330Ω resistors in parallel? For three in parallel? Calculate the mathematically expected loop currents using loop analysis with both the measured and nominal resistance values (pre-lab) and compare to your loop current measurements. Discuss the differences.

(c) For the circuit of Figure 7.2 (part 3), compare the predicted values of loop current from the pre-lab (assuming the nominal resistors) with the measurements made in the lab with the actual resistors. What is the percent difference [$100\% \times (I_{predicted} - I_{actual})/I_{predicted}$] between the predicted currents and the measured currents? Are the differences within the resistor tolerance range? Also compare the measured node voltages with predictions based on Ohm's law.

(d) What would you change about this experiment to make it more interesting?

Revised 7/94

Lab # 8
TITLE: Operational Amplifiers

ABSTRACT

The use of operational amplifiers as circuit building blocks is explored in this experiment. Basic properties of op amps are observed using simple resistive networks. A few of the practical limitations of op amp integrated circuits are also introduced and compared with several of the ideal assumptions.

INTRODUCTION AND THEORY

Operational Amplifiers ("Op Amps") are an extremely important component for a wide range of low power electronic circuits. The term *operational* refers to the use of op amps in electronic circuits which perform arithmetic *operations* on the input voltages (or currents) applied to the circuit. Although the concept of an operational amplifier dates back to the era of World War II, the development of *integrated circuits* (ICs) from the 1960's to the present has resulted in a large number of op amp types and features. Many general purpose op amps in IC form cost less than 50 cents. In fact, the *socket* the op amp IC plugs into may cost as much or more than the amp itself!!

The op amp is depicted schematically as shown in Figure 8.1. The figure shows the two op amp inputs: "-" for the inverting input and "+" for the non-inverting input; the op amp output, and the power supply connections. NOTE that the power supply connections are not always shown in diagrams, but they must be included in the actual circuit.

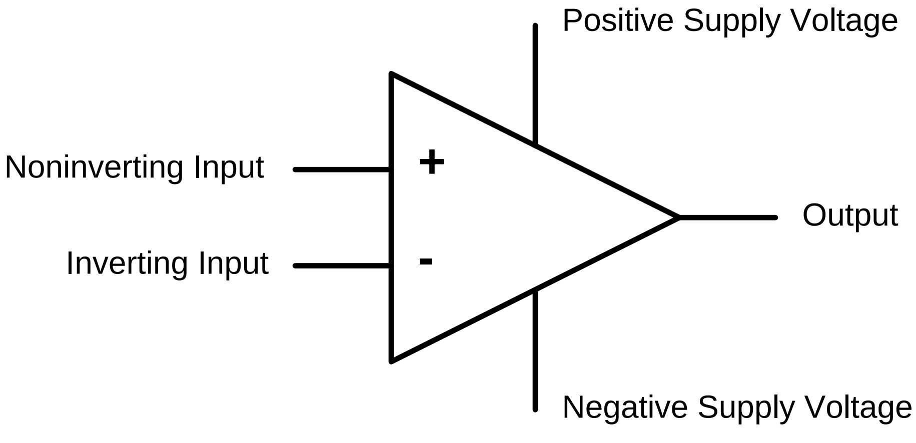


Figure 8.1

The integrated circuit op amp used in this experiment is shown in Figure 8.2. The circuit is contained in a *dual in-line package*, or DIP for short. The DIP has a notch or stripe at one end to indicate the correct orientation of the circuit. The standard part number is usually printed on the top of the DIP. Note that the pin numbers are assigned in counter clockwise order beginning at the notch end. In addition to the two inputs, the two power supply pins, and the output, notice that this particular op amp has three other pins: one labeled *NC*, meaning no connection, and two labeled *offset null*. The offset null pins allow us to make small adjustments to the internal currents in the IC in order to force the output voltage to be zero (null) when the inputs are both zero in order to compensate for the anticipated manufacturing variations from chip to chip. We will not need to use the offset null feature in this experiment, so no connections will be made to the offset null pins. It is also important to realize that there is no "ground" pin on the op amp: the amp receives its ground reference via the external components and connections of the complete circuit.

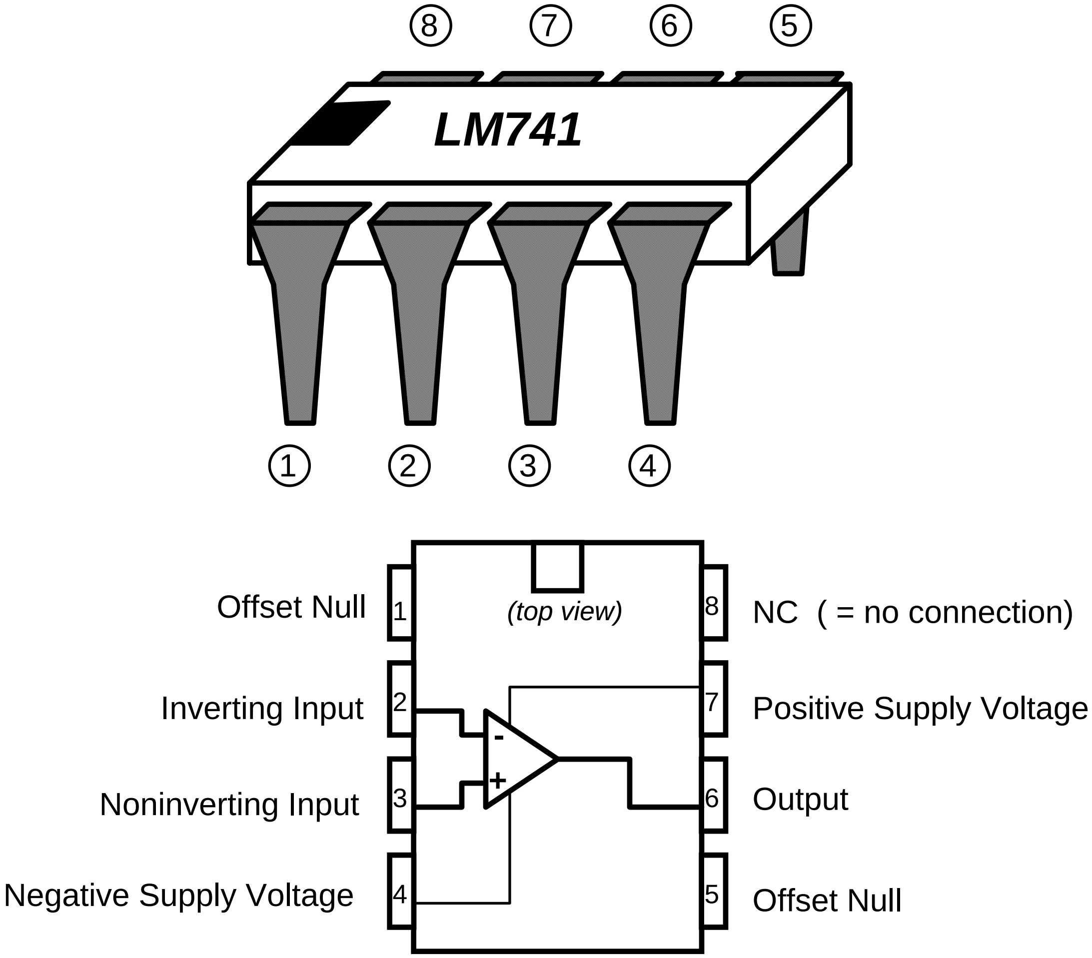


Figure 8.2

While the particular IC used in this experiment contains a single op amp, many other IC types are produced which contain two or more op amps in a single DIP package.

Integrated circuit op amps behave very much like the conceptually "ideal" op amp used in circuit analysis. There are some important limitations to keep in mind, however. First, the supply voltages cannot exceed some maximum rating, typically ±18 V DC. The op amp will usually operate using lower voltage supplies, *but exceeding the maximum rating will destroy the IC*. Second, the output voltage from an IC op amp is usually limited to be a volt or two smaller than the power supply voltages, e.g., the output voltage swing of an op amp with ±15 V supplies is, perhaps, ±13 V. Third, the output current from most op amps is limited to 30 mA or so, meaning that the load resistance attached to the output must be large enough that no more than the maximum current flows when the output voltage is maximum.

IC op amps have many other characteristics that will be considered in subsequent electrical engineering courses. *When in doubt about the limitations of an op amp it is best to refer to the manufacturer's data sheet*.

It is common to include *power supply bypass capacitors* in op amp circuit designs. Capacitors are charge storage elements which will be discussed in more detail later in this course. *Bypass* refers to the good design practice of placing capacitors across the power supply connections to help stabilize the DC power supply voltages and "bypass" any noise or interference on the supply lines to ground.

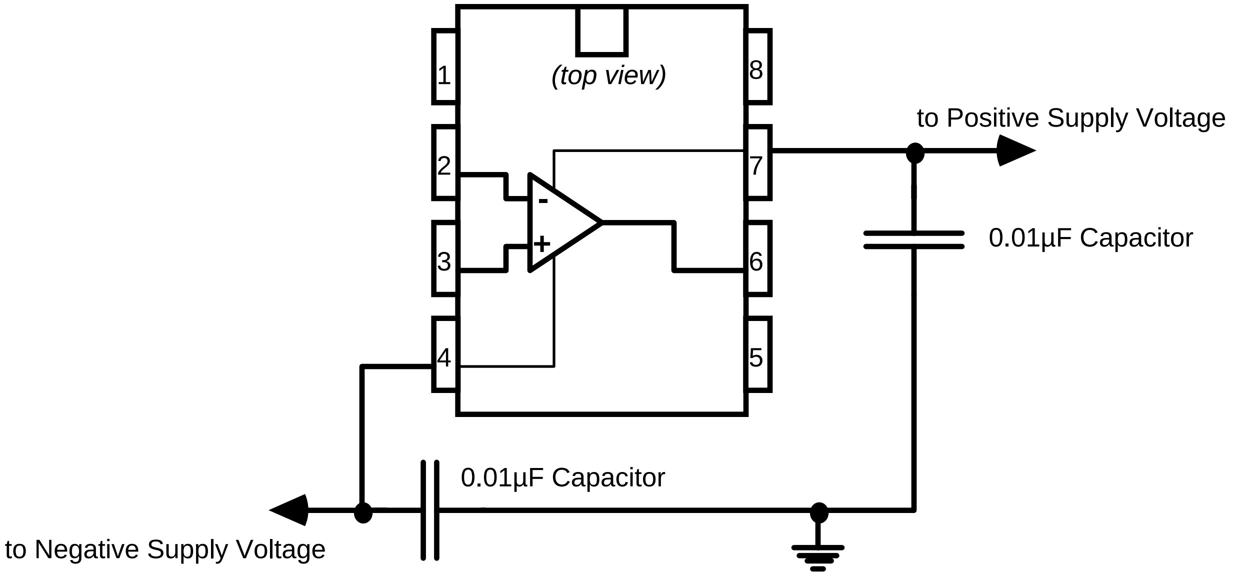


Figure 8.3

This means connecting small (typically 0.01µF) capacitors between the positive supply voltage and the circuit ground and between the negative supply voltage and circuit ground, as shown in Figure 8.3. The bypass capacitors should be placed as close to the IC as possible.

Many different types of op amps are available from commercial manufacturers. The type 741 op amp used in this lab was originally introduced by Fairchild Semiconductor in 1968, so it is an "old", reliable, well-understood, and inexpensive IC. *The 741 is by no means the best op amp for every purpose*: more recent designs reflect the advances of integrated circuit technology that have taken place over the last 20+ years. The 741 op amp is used here because it is a good example of the so-called *general purpose* operational amplifiers that are used in everything from radios and wireless telephones to car engine control systems and the space shuttle.

REFERENCES

See section 3.4 of the text by J. David Irwin, *Basic Engineering Circuit Analysis*, 4th ed., Macmillan Publishing Co., 1993 (pp. 138-148).

For op amp comparisons and circuits, see, for example: Walter Jung, "*IC Op-Amp Cookbook*, 3rd ed., Howard W. Sams Co., 1986.

See also manufacturer's data books, such as Signetics, National Semiconductor, Motorola, PMI, Linear Technology, etc. for op amp specifications and applications information. Data books are in the IEEE Lounge and the Engineering Library.

EQUIPMENT

Resistor kit
DC Power Supply
Multimeter
Heathkit Trainer
741 Op Amp IC
Signal Generator
Oscilloscope
0.01µF bypass capacitors

PRE-LAB PREPARATION

(I) Determine the voltage gain of the circuits in Figure 8.4 a and b. Use the "ideal" op amp model.

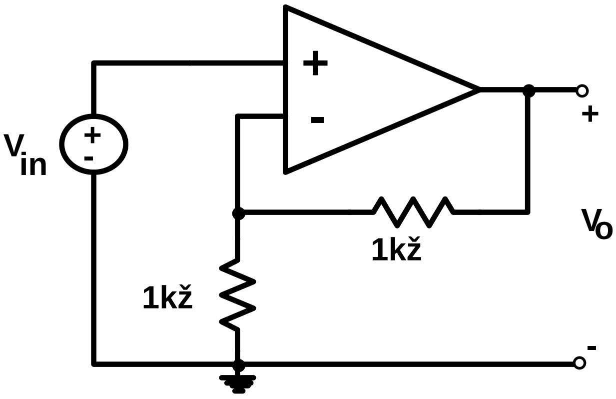


Figure 8.4a ∅

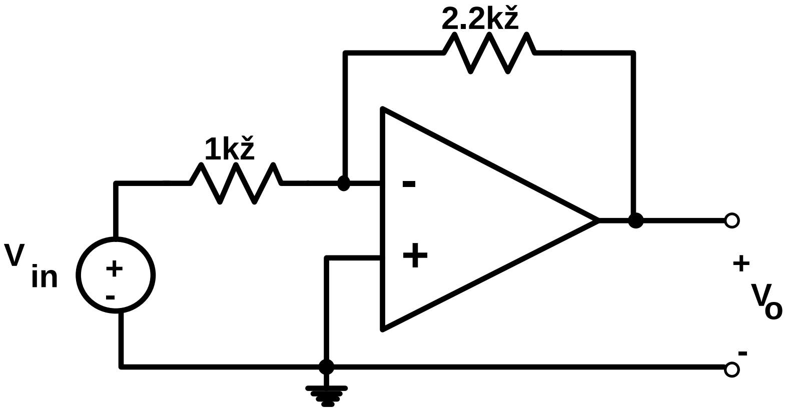


Figure 8.4b ∅

(II) The circuit shown in Figure 8.5 is referred to as an *inverting summer*. Calculate the output voltage in terms of the three input voltages assuming an "ideal" op amp.

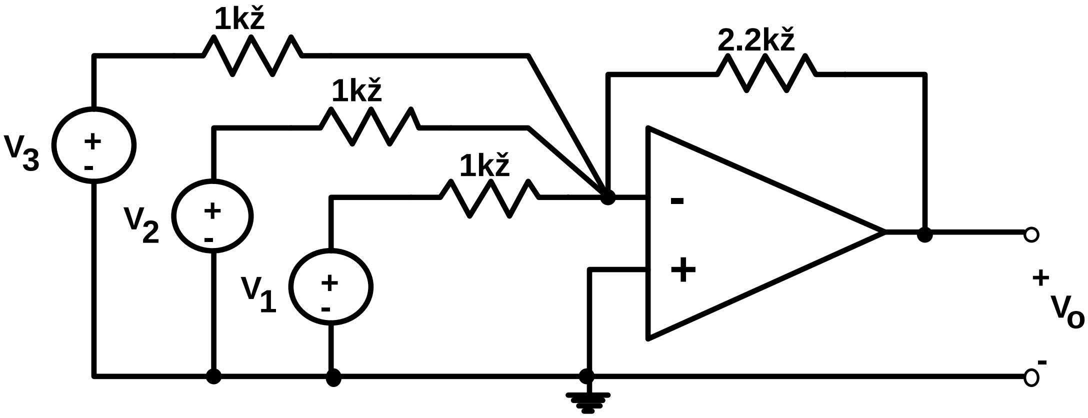


Figure 8.5

EXPERIMENT

This experiment uses the Heathkit powered breadboard "trainers". The trainer has an internal power supply system with ±12 volt, +5 volt, and 0 volt (ground) outputs. We will use the ±12 volt outputs to power the op amp circuits. *When working with electronic devices ALWAYS assemble and verify the circuit with the power OFF. Once the circuit has been checked, then apply the power.* ICs can be damaged by incorrect voltage connections. Work carefully and methodically.

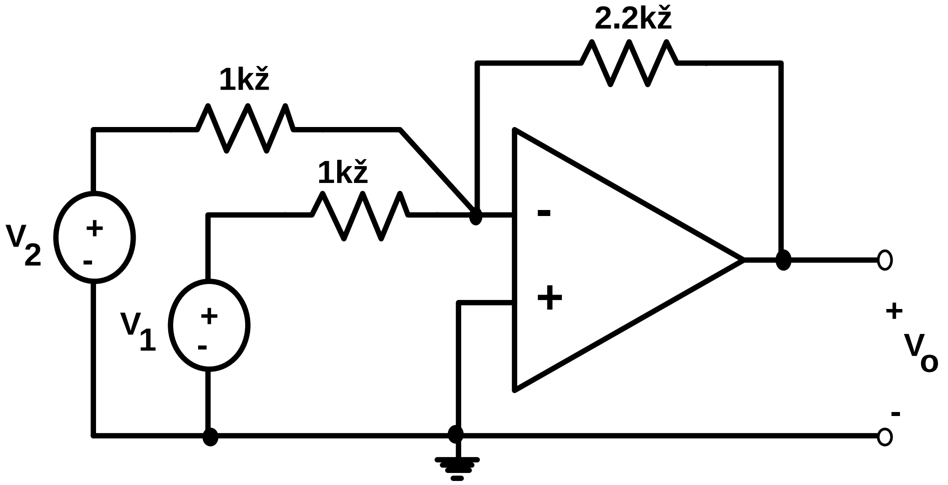
(1) Assemble the circuit of Figure 8.4a on the Heathkit trainer breadboard. Measure and record the actual values of the nominal 1kΩ resistors. Remember the power supply connections: use V+ = +12 volts to IC pin 7, use V- = -12 volts to IC pin 4. Include 0.01μF bypass capacitors between +12V and ground and between -12V and ground, placing the capacitors as close to the IC as possible. Use the bench power supply for V_{in} . *Note that V_{in} is referenced to ground: make sure you have the negative terminal of the bench supply connected to the ground on the trainer board.* Record the output voltage as you carefully vary V_{in} between 0 volts and 7 volts in 0.5 volt steps.

Now again record the output voltage as you vary V_{in} between 0 volts and -7 volts in -0.5 volt steps. (How can you use the bench supply to produce negative voltages with respect to ground?)

(2) Assemble the circuit of Figure 8.4b on the trainer breadboard. Measure the resistors used. As for the previous circuit, measure and record the output voltage as you vary the input voltage from -7 to +7 volts in 0.5 volt steps.

(3) Now use the signal generator in place of the bench power supply for V_{in} (still the circuit of Figure 8.4b). Observe the signal generator signal on channel A of the oscilloscope and the op amp output on channel B. Make sure the 'scope inputs are set for DC coupling. Adjust the signal generator for a sinewave at ~500 Hz frequency, and adjust the amplitude so that the signal *out of the op amp* is 4 volts peak-to-peak. Determine the input peak-to-peak voltage.

(4) Modify the circuit of Figure 8.4b by adding another input as shown next:



Use the signal generator for V_1 and the bench power supply for V_2. Start with V_2 (the bench power supply) set to zero, and readjust the signal generator if necessary so that the op amp output is a 4 volt peak-to-peak sinewave at ~500 Hz. Display the bench supply voltage (V_2) on 'scope channel A and the op amp output on channel B. Observe how the output of the op amp changes as the bench supply voltage is varied between 0 and 10 volts. Record the maximum and minimum values of the output voltage when the bench supply is ±1, ±2, and ±5 volts.

RESULTS

(a) Present your measurements from part 1 in the form of a graph with V_{in} as the abscissa and V_o as the ordinate. Label your plot. What is the slope of the curve? Does it pass exactly through the origin? How does the result compare to your predictions using the ideal op amp model?

(b) Present your measurements from part 2, again as a V_o vs. V_{in} plot. Discuss the results.

(c) What was the input peak-to-peak voltage required in part 3 to get a 4 volt peak-to-peak output waveform? How does this relate to your results from part 2?

(d) Describe the function of the circuit used in part 4. What is the mathematical relationship between the output voltage and the two input voltages? Present your measurements for the various DC input voltages.

(e) What can be done to improve this experiment?

Revised 7/94

Lab # 9

TITLE: Design and Circuit Simulation Using SPICE

ABSTRACT

This lab previews several circuit design and analysis issues. Simulation of electrical circuits using digital computer software is an important tool of modern circuit design. This experiment applies the widely used circuit simulator SPICE to DC analysis of electrical circuits. Although SPICE was developed for and is particularly useful in the design of integrated circuits, it can be a helpful aid in discrete circuit analysis as well.

INTRODUCTION AND THEORY

Most careers in electrical engineering involve at least to some extent the design of electrical apparatus: control circuits, power distribution systems, industrial processes, software programs, interface electronics, and so forth. Unfortunately for the student, it is not possible to learn the "art" of design by simply reading textbooks–although that is one way to gather information. Many students (and working engineers!) are uncomfortable with design concepts at first because the possible solutions and problems seem endless: one may feel initially that the design choices to be made are arbitrary at best and overwhelming at worst.

Design involves a combination of *theory, analytical skills, creativity, and practical experience*. Theory and analytical skills are what you are taught in the classroom. These are the elements that separate a BSEE degree from an Associates degree. Creativity involves a good understanding of the strengths and limitations of electrical engineering components, concepts, and technology. Of all the design ingredients, *practical experience* is perhaps the most important element. Thus it is important to take advantage of any opportunity you may have to do some practical circuit design and testing. One can also take advantage of the experience of others by asking questions and studying examples of practical circuits in books, magazines, etc.

Some design specifications do not directly involve the electrical performance of the circuit. In no particular order, these additional specifications might include

- size and weight (physical attributes)
- compatibility with standards or other devices
- time deadline for the design to be complete
- no circuit adjustments during fabrication
- minimizing cost of the parts
- minimum number of circuit elements to inventory
- upgrade capability
- operating temperature range
- availability of parts
- ease of manufacturing
- power consumption
- minimum number of interconnections
- electromagnetic compatibility
- safety
- reliability
- ease of testing
- ease of repair
- modularity

...etc...

SPICE is an acronym for *__S__imulation __P__rogram - __I__ntegrated __C__ircuit __E__mphasis*. It was developed at the University of California - Berkeley as a way to test and evaluate designs for integrated circuits before actually fabricating the ICs: it is far cheaper to discover and correct problems using software simulation than to fabricate and test each new design. Versions of SPICE are available for almost every type of computer, from small PCs to workstations to supercomputers. Although many other circuit simulators are now available, SPICE was the first to gain widespread acceptance and to become an industry standard.

SPICE is based on a set of software *models* for various circuit elements. Built-in models include independent and dependent voltage and current sources, resistors, inductors, capacitors, transformers, and semiconductors (diodes, transistors, etc.). All of the models are described by various user-specified parameters. For example, the resistor model is essentially Ohm's law, where the user specifies the resistance value. Like most SPICE models, the resistor model contains more sophisticated features, like a temperature coefficient to indicate the change of resistance as a function of temperature. In general the SPICE models contain reasonable *default* values so that the user need only specify the parameters of interest.

SPICE is able to perform several different types of simulation: DC, AC small signal, transient, sensitivity, noise, distortion, etc. This lab will use only DC analysis, but it is important to be aware that the real power of SPICE lies in the analysis of complicated circuit parameters and interactions.

The version of SPICE available on the department's microcomputers, PSpice, has several features that may be different from the original SPICE versions which were designed to run on mainframe computers. In particular, the PROBE feature is a useful way to obtain plots and graphs of the simulation results. To use PROBE, simply include a line in your normal SPICE input (circuit) file:

```
.PROBE
```

Inclusion of the ".PROBE" line instructs PSpice to generate a special output file containing the simulation data. Then, running the PROBE program with the simulation data file allows the display of any SPICE output variable using high-resolution graphics. NOTE that PSpice will generate the PROBE data automatically without including the ".PROBE" line if you run your simulation from within the PSpice window system (ps.exe).

PSpice on the PCs includes a *library* of several "subcircuits". The subcircuits are accurate models of complicated components such as op amps. One of the available models is the **uA741**, which is a model of an actual 741 op amp. To use the library include a line in your input file:

```
.LIB EVAL.LIB
```

This causes PSpice to load in the subcircuit definitions from the file eval.lib. To use the 741 subcircuit you need to make the following connections:

```
XOPAMP n1 n2 n3 n4 n5 uA741
```

where *`n1`* = noninverting input
`n2` = inverting input
`n3` = positive power supply
`n4` = negative power supply
`n5` = output
are the node numbers from your circuit.

The subcircuit name `XOPAMP` is not important, but you must use a name beginning with "X", which is the SPICE convention for subcircuit calls. The `uA741` name tells SPICE that you will be using the 741 model from the `EVAL.LIB` file.

It is vital to realize that a circuit simulator such as SPICE is only a tool: **YOU** must specify the circuit interconnections, choose the parameters, and evaluate the results. This means that you must know what to expect in the SPICE output, and what circuit parameters to change if the results are not acceptable. Sitting at a terminal and randomly changing parameters in hope of coming up with good results is *not* the strategy of an electrical *engineer*. In short, you must still provide the design, correctly specify the circuit for SPICE, and interpret the results. This means maintaining good documentation and program comments, performing a quick hand analysis to verify that the SPICE results are reasonable, and understanding the limitations of the SPICE models.

REFERENCES

See chapter 4 of the text by J. David Irwin, *Basic Engineering Circuit Analysis*, 4th ed., Macmillan Publishing Co., 1993 (pp. 166-186).

SPICE User's Guides available from UNL IEEE Student Branch.

Paul W. Tuinenga, *SPICE: A Guide to Circuit Simulation and Analysis Using PSpice*, 2nd Ed., Prentice-Hall, 1992.

For references on electronic design refer to the numerous electronic design books and periodicals in the library. Also, don't be afraid to look at "hobbyist" materials for simple, practical design basics.

EQUIPMENT

Resistor kit
0.01µF bypass capacitors
741 Op Amp IC
Multimeter
DC Power Supply
Heathkit Trainer

PRE-LAB PREPARATION

(I) The circuit in Figure 9.1 depicts a voltage source with internal resistance $R_S = 10k\Omega$ driving a $1k\Omega$ load resistance R_L. Sketch the circuit labeling all components and nodes, then write a SPICE program to determine the voltage V_o and the current through R_L for $-10 \leq V_{in} \leq +10$ volts. Use the SPICE results to calculate the power dissipation in R_L. Compare the SPICE results to a hand analysis of the circuit.

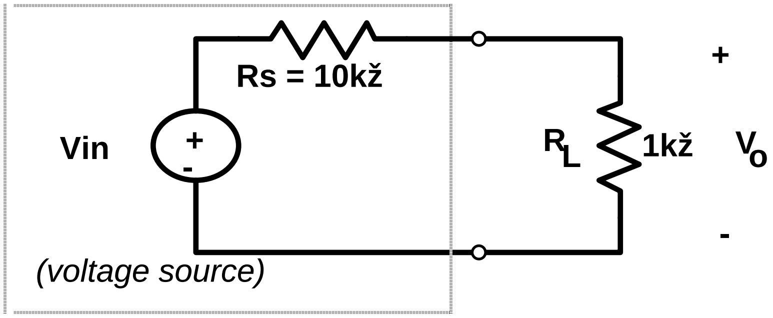


Figure 9.1

(II) A simple "ideal" model of an op amp can be implemented in SPICE using a voltage-controlled voltage source (Figure 9.2).

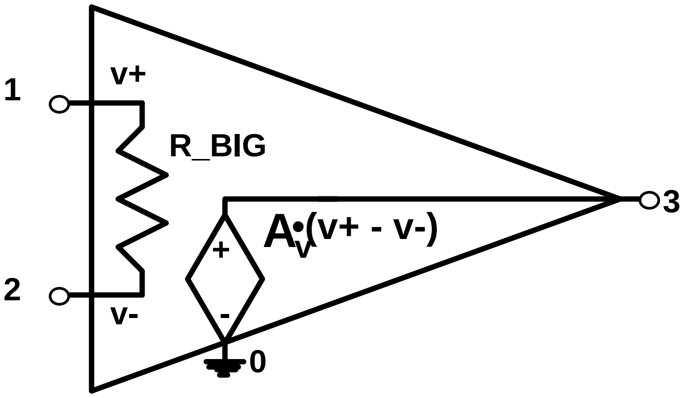


Figure 9.2

The voltage-controlled voltage source (VCVS) is indicated in SPICE by using 'E' as the first letter of the source name. With the nodes labeled as in Figure 9.2:

```
EOPAMP 3 0 1 2 1MEG
```

This means that the VCVS is connected with its positive end at node 3, negative end at node 0 (ground), and is controlled by the voltage at node 1 minus the voltage at node 2. The VCVS output voltage is 10^6 = 1 million (1MEG or 1E6) times the input voltage difference. This op amp has a large input resistance (R_BIG), zero output resistance and a voltage gain of 10^6. NOTE that this simple model can be modified to include output resistance and other parameters.

In Figure 9.3 an "ideal" op amp voltage follower is placed in between the source and the load (see Figure 9.1). Using the voltage-controlled voltage source approach described above for an ideal SPICE op amp, determine the voltage V_o and the power dissipation (V_o^2/R) in R_L in this case for V_{in} varying from -10 to +10 volts in 0.5 volt steps. Use a voltage gain of 10^6. Provide a labeled sketch of your SPICE circuit, and compare the results at V_{in} = -10, 0 and +10 volts to a hand analysis of the circuit.

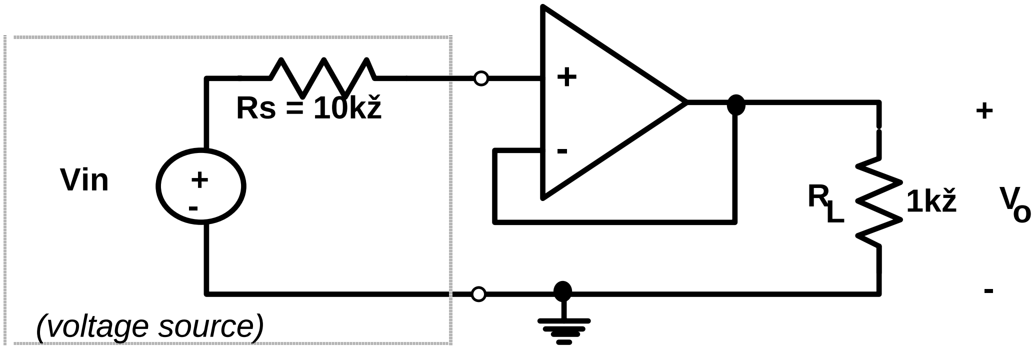


Figure 9.3

(III) Perform another SPICE analysis of the circuit of Figure 9.3, but with the voltage gain of the op amp set to 10^2 instead of 10^6. Explain the results.

EXPERIMENT

(1) Construct the circuit of Figure 9.1. Use the bench supply in series with a nominal 10kΩ resistor for the voltage source model. Measure the resistors used and adjust the bench power supply using the multimeter. Vary the input voltage from -10 to +10 volts in 0.5 volt steps, and measure the voltage and current for the 1kΩ resistor (R_L) and compare to the results obtained in the prelab with SPICE.

(2) Construct the circuit of Figure 9.3 using the Heathkit trainer and a type 741 op amp. Use the ±12 volt supplies on the trainer to power the op amp. Vary the input voltage from -10 to +10 volts in 0.5 volt steps. Carefully measure the voltage and current in R_L and the voltage at the input to the op amp for each step.

(3) Now construct an op amp circuit with a nominal gain of -10 (inverting amplifier) and apply a series of input voltages ranging from -2 to 2 volts in 0.2 volt steps. Record the output voltage at each value of input voltage.

RESULTS

(a) Explain your results for part 1. Perform another SPICE analysis using the measured values of the resistors instead of the nominal 10kΩ and 1kΩ values. How do the measured and predicted values agree in each case?

(b) In part 2 the op amp voltage follower is used to "match" the high internal resistance of the voltage source to the lower resistance of the load resistor. Compare the measured results for part 2 to the results for part 1. Perform another SPICE analysis using the measured values of the resistors instead of the nominal values. How do the measured and predicted values compare for this part?

(c) Perform a SPICE analysis on the circuit of part 3, using the simple "ideal" VCVS op amp and the measured values of the resistors. How do the SPICE results and the measurements compare?

(d) Use the 741 op amp functional model (`uA741`) available with the PSpice program and re-analyze the circuit of part 3. How do the SPICE results using the functional model compare to the lab measurements and to the simple VCVS op amp model?

(e) Were the procedures for this lab easy to follow? How would you improve this experiment?

Revised 7/94

Lab # 10
TITLE: Thévenin and Norton Equivalent Circuits

ABSTRACT

It is often possible to simplify the analysis of a complicated circuit with an equivalent Norton or Thévenin circuit. This approach also is used to divide a circuit into linear and nonlinear parts. The linear part can be simplified to a single voltage/current source and an equivalent resistance. Also incorporated into the lab is a sensitivity analysis (w/SPICE) of an equivalent circuit.

INTRODUCTION AND THEORY

Linear circuits may be replaced by a single source (v_{oc} or i_{sc}) and an *equivalent* (Thévenin) resistance (R_{th}). R_{th} is determined by removing all independent sources (short voltage sources and open current sources). V_{oc} is found by measuring the open circuit voltage across the output and i_{sc} is found by measuring the current between the shorted output connections. The Thévenin and Norton circuits are equivalent and if one is known, the other is also easily determined. Actually measuring the short circuit current in a real circuit is often not recommended (the circuit may not be designed to handle the high current) and may damage the circuit. Circuits with **no** independent sources require a different technique; a source must be connected to the output and the current or voltage measured.

Sensitivity analysis determines what effect a variation in component value will have on the desired output. All components used in the lab have tolerance ranges; most resistors are ±5% and some types of capacitors may vary ±25%. Components with smaller tolerances cost more so it is often necessary to determine how much variation can be allowed and the circuit still work. One method of sensitivity analysis is available with SPICE. Using the .SENS statement the effect of component variation on any circuit parameter can be determined. This analysis is only valid for **small** variations in circuit component values.

REFERENCES

Chapter 5 of*Basic Engineering Circuit Analysis*, 4th edition, by J. David Irwin. Macmillan Publishing Co., 1993.

SPICE A Guide To Circuit Simulation & Analysis Using PSpice, 2nd Ed., by Paul W. Tuinenga. Prentice Hall, 1992.

Digi-Key Catalog, by Digi-Key Corporation, printed bi-monthly.

EQUIPMENT

Multimeter	Resistor kit (100,220,470,560,etc)
DC power supply	Breadboard (or Heathkit trainer)

PRE-LAB PREPARATION

(I) Find the Thévenin and Norton equivalent circuits for Figures 10.1 & 10.2. Now with V= +10 volts, determine the power absorbed by the 100 Ω resistor if the output terminals were shorted (to determine i_{sc}).

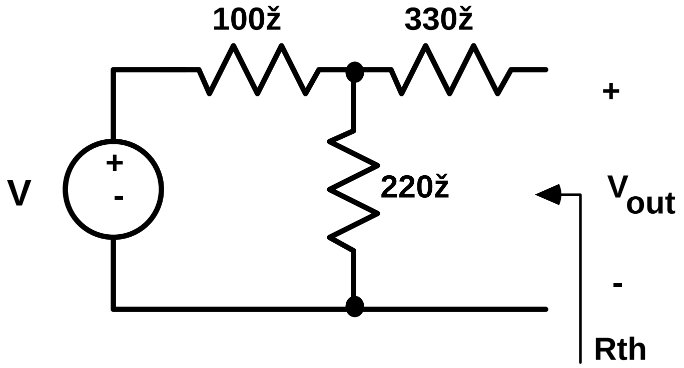


Figure 10.1

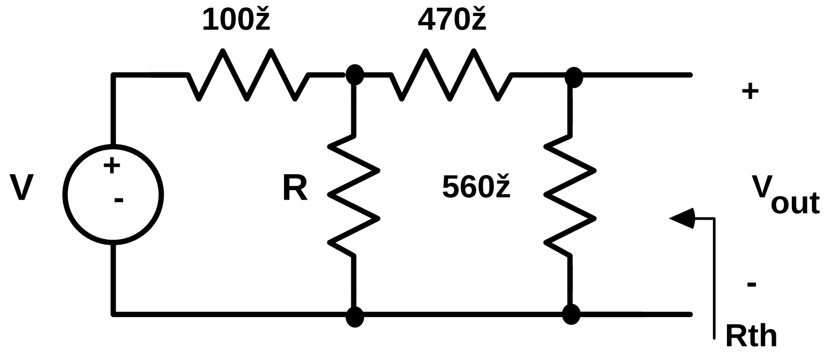


Figure 10.2

(II) Compare the price of 5% tolerance carbon and 1% (any type) 0.25 watt resistors using the latest Digi-Key catalog.

(III) Calculate the minimum and maximum V_{th} for Figure 10.1 using 10% tolerance resistors.

(IV) Do a sensitivity analysis of circuit 1 using SPICE. A 1 volt increase (to 11 V) of the power supply results in how much variation of the output voltage?

(V) If V_{oc}=5 V and R_{th}=275 Ω (circuit 2), determine the power supply voltage of the original circuit and the value of R. Use the Digi-Key catalog to determine the value of the closest 1% and 5% tolerance 0.25 watt carbon resistors available.

EXPERIMENT

(1) Construct circuit 1 (using a 100 Ω resistor with large enough power rating to absorb i_{sc}). Measure v_{oc} and i_{sc} at the output. *Remember to measure the resistance of each resistor used.*

Now imagine that you are unable to measure i_{sc} because a 100 Ω resistor with a large enough power rating is unavailable (or pretend the power supply might be damaged). Take an additional measurement that will allow you to construct the Thévenin equivalent.

(2) Place the following "loads" across the output and record voltage across and current through. (Use 100 Ω, 220 Ω, 330 Ω and 470 Ω)

(3) Construct a Thévenin equivalent source (using results of part 1a or 1b) and repeat part 2 (measure current *through* and voltage *across* 4 different "load" resistors.)

RESULTS

(a) Compare the calculated Thévenin equivalent (prelab) with the experimentally determined Thévenin equivalent. Discuss any differences between them and possible sources of the discrepancy.

(b) Sketch the Thévenin equivalent circuit determined experimentally in part 1. Use this equivalent circuit and calculate the *expected* value of current and voltage across the 4 different load resistors. Compare expected voltage and current in the loads with the *actual* measurements. Discuss discrepancies between calculated values and the actual measurements. Also sketch the Norton equivalent circuit for circuit 1.

(c) Discuss circuit sensitivity, component tolerance, and component cost as seen by an electronics manufacturer/designer.

(d) The internal impedance of the power supply is stated to be less than 0.02Ω at DC. Is the internal impedance large enough to affect the Thévenin equivalent circuit here?

(e) How would you improve this experiment?

Revised 7/94

Lab # 11
TITLE: Superposition

ABSTRACT

The superposition theorem is applied to a simple **linear** circuit containing more than one source. The experiment will show superposition can be applied to voltage and current but not to power. Also, a battery is used as a power source to demonstrate a non-ideal voltage source.

INTRODUCTION AND THEORY

The principle of superposition allows complicated circuits with multiple sources to be analyzed one source at a time. The principle states (Irwin, page 186) that:

> "In any linear circuit containing multiple independent sources, the current or voltage at any point in the network may be calculated as the algebraic sum of the individual contributions of each source acting alone."

While determining the current or voltage from any one source, all other *independent* voltages are replaced by shorts and *independent* current sources are opened. *Dependent* sources remain in the circuit. The most important limitation of superposition is the network MUST be linear. In other words, the composite value of voltage or current in any circuit element is the *sum* of the effects of the individual sources.

REFERENCES

Basic Engineering Circuit Analysis, 4th edition, by J. David Irwin. Macmillan Publishing Co., 1993. Chapter 5.

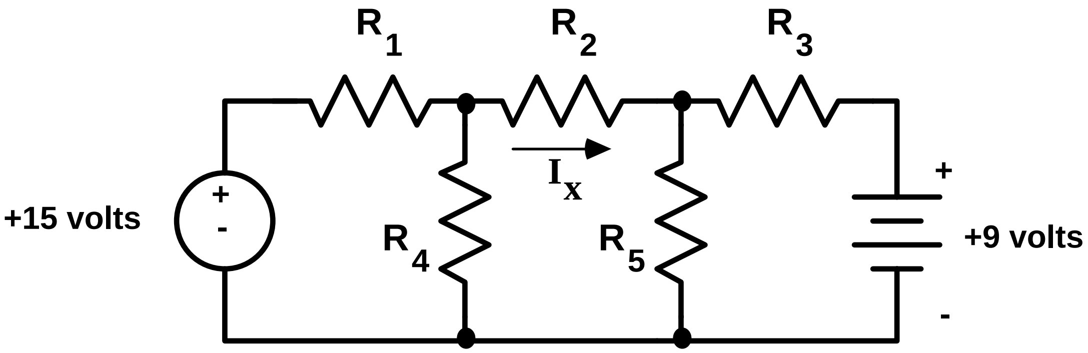
SPICE A Guide To Circuit Simulation & Analysis Using PSpice, 2nd Ed., by Paul W. Tuinenga. Prentice Hall, 1992.

EQUIPMENT

Multimeter
DC power supply
Breadboard (or Heathkit trainer)
Resistor kit (100,220,470,560,etc)
9 volt battery

PRE-LAB PREPARATION

(I) Use superposition to calculate the current I_X in the circuit of Figure 11.1.



R_1 = 100ΩR_2 = 50Ω
R_3 = 100ΩR_4 = 220Ω
R_5 = 470Ω

Figure 11.1

(II) Verify the results of the superposition method (used in step 1) by finding I_X using mesh analysis.

(III) For the circuit of Figure 11.1, find the value of R_4 which will cause I_X to equal zero.

(IV) Use the superposition method (as in part I) to find:

a. Power dissipated in R_2 if the 9 volt battery is replaced with a short circuit.
b. Power dissipated in R_2 if the 15 volt supply is replaced with a short circuit.
c. Power dissipated in R_2 with both sources active in the circuit.

Compare the sum of the powers due to each source acting individually with the power delivered to R_2 with both sources in the circuit. Explain the results.

EXPERIMENT

(1) Determine the *internal resistance* of the 9 volt battery by:

a. Measuring the open terminal (V_{OC}) voltage of the battery.
b. Using a multimeter to find the exact resistance of a 220 Ω resistor.
c. Connecting the 220 Ω resistor across the battery and measuring V_L, the load voltage across the resistor.

d. Calculating or measuring the current through the 220 Ω resistor.
e. Using these values, determine the resistance.

(2) Measure the resistance of all resistors for Figure 11.1 and then assemble the circuit.

(3) Measure the value of the current I_X.

(4) Replace the 15 volt power supply with a short circuit and then measure I_X.

(5) Return the 15 volt supply to the circuit and remove the 9 volt battery. Since the battery has a significant internal resistance, replace the battery with resistors (or variable potentiometer) equal to the resistance determined experimentally in part 1. Measure I_X.

(6) Return the battery to the circuit and replace R_4 with a resistor of value calculated to reduce I_X to zero (from prelab). Measure V_{R4}, V_{R5} , and the current through R_2. Adjust the value of R_4 to reduce the current I_X to zero, then record the value of R_4.

(7) Measure the resistance of a 470 Ω resistor and connect it across the DC power supply. Measure the current through the resistor as the supply is varied from 1 to 10 volts (1 volt increments). Use the multimeter to adjust the voltage (not the internal supply meter).

RESULTS

(a) Apply superposition again (as in the prelab) but now include the internal resistance of the 9 volt battery. Compare I_X obtained (including the battery resistance) with the prelab and experimentally determined values of I_X. Comment on the differences. How would you model the battery as a Thévenin equivalent?

(b) Compare the voltages measured across R_4 and R_5 after R4 was replaced (part 6).

(c) Use a full page of your lab notebook to plot the data from part 7. On the same graph plot power delivered to the resistor versus current. Adjust the power scale so the graph uses most of the page. Referring to the graph, discuss the application of superposition to power in a linear circuit

(d) How would you improve this experiment?

Revised 7/94

Lab # 12
TITLE: Power Relationships in Simple Circuits

ABSTRACT

In many applications it is important to determine relationships between the power supplied by sources in a circuit and the power delivered to other circuit elements. This experiment considers the concept of *maximum power transfer* and other power calculations in simple resistive circuits.

INTRODUCTION AND THEORY

In electrical circuits it is often necessary to transfer a voltage, current, or power from a source to load. This could be a battery source connected to a light bulb load, an audio amplifier source connected to a loudspeaker load, etc. We are often interested in transfering the maximum amount of *power* from the source to the load, although many low power communication circuits are designed to transfer the maximum *voltage* from source to load.

An arbitrary connection between a source and a load can be depicted as in Figure 12.1. NOTE that the voltage source and source resistance R_S could actually be the *Thévenin equivalent circuit* for a more complicated network.

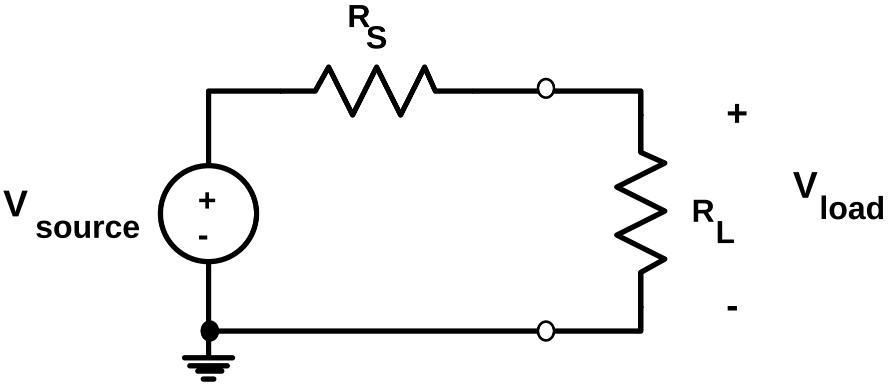


Figure 12.1

Case 1: Maximum voltage transfer

If it is desired to maximize the load voltage V_{load}, what relationship between R_S and R_L is necessary? Using voltage division:

$$V_{load} = \frac{R_L}{(R_S + R_L)} \cdot V_{source},$$

which means that **V_{load} is always less than V_{source}, and is maximized for $R_L >> R_S$**. So in order to maximize the voltage at the load we must design the circuit so that the load resistance is much greater than the source resistance. One way to accomplish this in electronic circuits (assuming the op amp can supply sufficient current) is to use a voltage follower between the source and the load: the op amp voltage follower has a high input resistance and a low output resistance.

Case 2: Maximum current transfer

If we need to supply the maximum available current from the source to the load we need a different relationship between R_L and R_S. Specifically:

$$I_{load} = \frac{V_{source}}{(R_S + R_L)}.$$

Assuming the source resistance is fixed, **the load current is maximized for $R_L << R_S$**.

Case 3: Maximum power transfer

To maximize the power delivered to the load, assuming the source resistance is fixed, we need to maximize the power expression with respect to R_L:

$$P_{load} = \frac{V_{load}^2}{R_L} = \left(\frac{V_{source}}{R_S + R_L}\right)^2 \cdot R_L$$

and solving $\frac{dP_{load}}{dR_L} = 0$.

The result is that **maximum power transfer occurs for $R_L = R_S$**.

REFERENCES

See chapter 5 of the text by J. David Irwin, *Basic Engineering Circuit Analysis*, 4th ed., Macmillan Publishing Co., 1993.

EQUIPMENT

Resistor kit DC Power Supply Multimeter

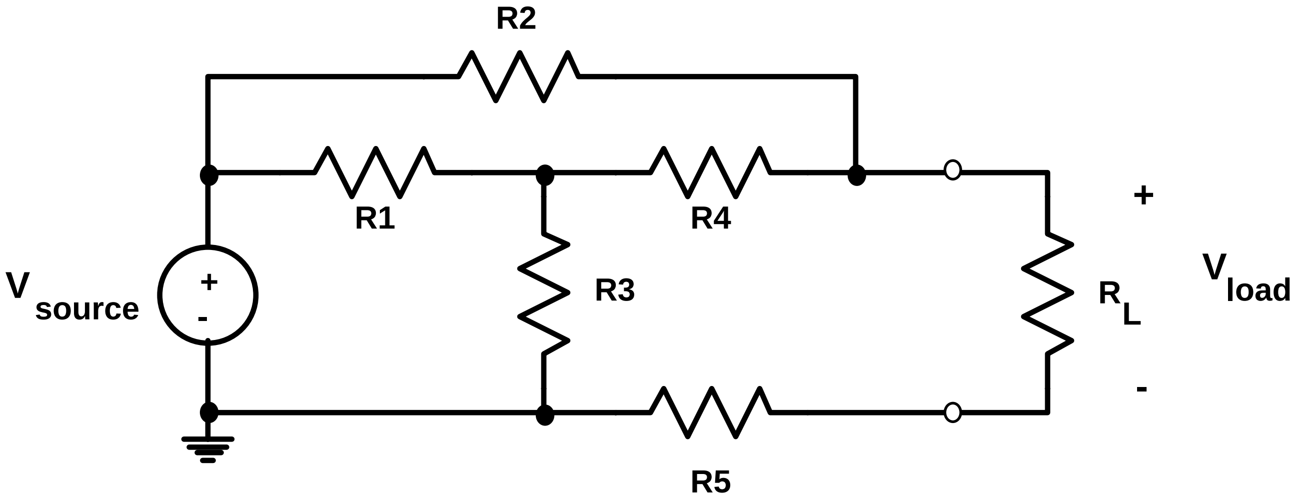
PRE-LAB PREPARATION

(I) Use a symbolic math program (MAPLE) to plot V_{load} vs. R_L/R_S for the circuit of Figure 12.1 over the range $0.05 \leq (R_L/R_S) \leq 20$. If it is required that 90% of the voltage V_{source} appear across the load, what is the minimum ratio R_L/R_S ?

(II) Use a symbolic math program to plot P_{load} vs. R_L/R_S for the circuit of Figure 12.1 over the range $0.05 \leq (R_L/R_S) \leq 20$.

(III) Verify for the maximum power transfer condition by solving $\frac{dP_{load}}{dR_L} = 0$.

(IV) For the circuit in Figure 12.2 determine a symbolic expression (in terms of R1, R2, ...) for the value of load resistance that will result in maximum power transfer from the source to the load. Then use the specified resistance values to calculate the power dissipated in *each* of the resistors with the optimum load resistor attached. Use a program to solve for the branch currents if you wish.



R1 = R2 = 330Ω R3 = 1kΩ
R4 = 2.2kΩ R5 = 470Ω

Figure 12.2

EXPERIMENT

(1) Assemble the circuit of Figure 12.1 using the bench power supply for V_{source}, R_S = 330Ω, and R_L = 1Ω. Use the multimeter to set V_{source} to 8 volts DC. Measure and record the voltage across R_L.

(2) Now replace the 1Ω load resistor in turn with the nominal resistor values (one resistor per value) in your lab kit (10Ω, 100Ω, 220Ω, etc.) and record the load voltage. Remember to record the actual values of the resistors you use!

(3) Construct the circuit shown in Figure 12.2 using the nominal resistors indicated (record the actual values used). Set the power supply to +10 volts. Using the symbolic expression you derived in the pre-lab and the measured values of your resistors, calculate the value of R_L for maximum power transfer. Construct a resistor of this value using a carefully adjusted 1kΩ potentiometer and/or resistors from your lab kit and attach it to your circuit. *Measure the voltage across your load resistor and the voltage across each of the other resistors in the circuit. Also measure the current flowing in the power supply.*

(4) Now replace the load resistor in turn with each of the nominal resistor values in your lab kit (1Ω, 100Ω, 220Ω, etc.) and record the load voltage in each case.

RESULTS

(a) Present a table of V_{load}/V_{source}, I_{load}, and P_{load} vs. R_L using your measurements from parts 1 and 2.

(b) Prepare a graph of V_{load}/V_{source} vs. R_L for the voltage measurements made in parts 1 and 2. *Use a logarithmic scale for the abscissa (R_L) axis.* How do your measurements compare to the mathematical expectations?

(c) Prepare a graph of I_{load} vs. R_L for the voltage measurements made in parts 1 and 2. Use a logarithmic scale for the abscissa (R_L) axis. Discuss the results.

(d) Prepare a graph of P_{load} vs. R_L using the measurements from parts 1 and 2. Again, use a log scale for R_L. Use your measurements to estimate the value of R_L at the maximum of the load power curve. How does this value compare to the predicted $R_L = R_S$ relationship?

(e) Present your results for parts 3 and 4. What was your calculated load resistor? What was the power dissipation in each of the resistors? What was the total power dissipation? How does the total power supplied by the bench power supply (V_{source} x I_{source}) compare to the sum of the power dissipated in each resistor?

(f) Give a table and a separate graph of V_{load}/V_{source}, I_{load}, and P_{load} vs. R_L using your measured data from parts 3 and 4. Discuss your findings.

(g) What needs to be improved about this experiment?

Revised 7/94

Lab # 13
TITLE: RL and RC Circuits

ABSTRACT

This lab exercise introduces basic RL and RC circuits. The output of any R, L, and C circuit can be analyzed to determine both the steady-state (forced) output and the transient (natural) response for any given input. This lab will include the analysis (including SPICE) and construction of simple RL and RC circuits.

INTRODUCTION AND THEORY

Simple R, L, and C circuits are often used for filtering, integration and differentiation of signals, and in oscillator/timer circuits. Although RL circuits have much in common with RC circuits, RC circuits are more commonly used in many practical applications at frequencies below a few MHz due to the smaller size and weight of capacitors compared to inductors. The analysis of RC and RL circuits is similiar and is discussed in depth in Chapter 7 of the text (Irwin, 3rd edition).

The solution for a circuit with only one storage device (capacitor or inductor) will always be a first-order differential equation (which can be solved several ways). In a series RL circuit, if $V_R << V_L$ (time constant L/R small compared to the **period** of the input signal) the voltage across the inductor (V_L) approximates a differentiator. If $V_R >> V_L$ (time constant L/R large compared to the period of the input signal) the voltage across the resistor (V_R) approximates an integrator.

REFERENCES

Basic Engineering Circuit Analysis, 4th edition, by J. David Irwin. Macmillan Publishing Co., 1993. Chapter 7.

EQUIPMENT

DC power supply
1 kΩ resistor
0.01µF bypass caps
0.4 H inductor w/~140 Ω internal impedance
Signal generator
0.1µF cap
Heathkit trainer
Oscilloscope
741 OP AMP

PRE-LAB PREPARATION

(I) A DC power supply (R = 600 Ω internal resistance) furnishes 10 V peak-to-peak to a 0.4 H inductor (which has 140 Ω *internal resistance*). At t=0, the switch moves from position 1 to position 2. (See Figure 13.1). The switch has a "make-before-break" action which means that the current in the inductor is **not** interrupted when the switch moves. NOTE: A 10 volt (peak-peak) square wave from the signal generator is actually a +5/-5 signal, i.e., varies about zero.

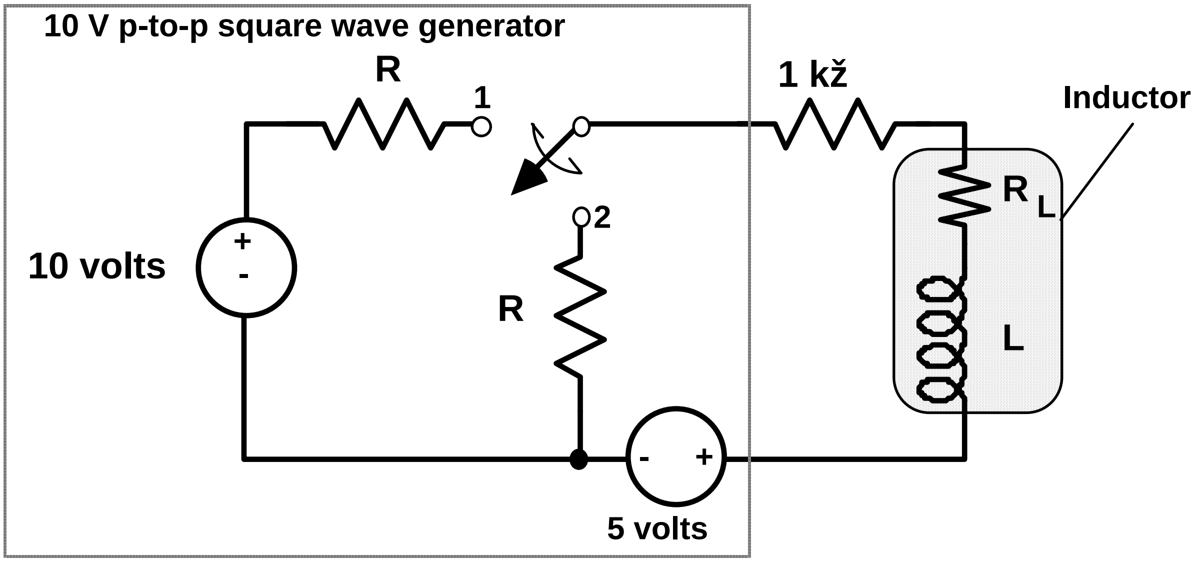


Figure 13.1

a. Find $i_L(t=0^-)$ and $V_L(t=0^-)$, the current and voltage measured at component "L" (not L + R_L) before the switch is moved.
b. Calculate the Thévenin equivalent resistance R_{TH} as seen by L.
c. What is the time constant for the circuit?
d. Find i(t) and $V_{L+R}(t)$ for t>0 where V_{L+R}=voltage across the actual physical inductor (L + R_L).
e. Find i(t=1TC, 2TC, 5TC) where TC=time constant.

(II) Model the circuit using SPICE (w/PROBE) to verify the time constant (TC) of each. Model the circuit with a square wave "pulse" generator instead of a DC power supply and switch. Initially set the "on" width of the square wave pulse ≅ 5 TC's. (f=1/(10TC). Then change the pulse period such that the frequency is first 10 times and then 1/10 of the initial frequency. Sketch and discuss the outputs which show either integration or differentiation. NOTE: Remember, in the lab you can not separate the inductor from the internal resistance: you will be measuring the voltage across an inductor **and** the internal resistance.

(III) Analyze the op amp differentiator circuit (C=0.1 μF) of Figure 13.2 (using section 7.5 of the reference). Assume a sine wave input (of arbitrary magnitude), and include a 600Ω source resistance with the signal generator.

(IV) Exchange the positions of the resistor and capacitor. Analyze the circuit. Assume a square wave input (of arbitrary magnitude).

(V) Model the op amp circuits of parts III and IV using PSpice. Assume an input voltage (from the signal generator) of 0.5 V. Obtain plots of V_{out} for a frequency of 500 Hz, 5 kHz, and 50 kHz.

(VI) What happens if the calculated output is greater than the op amp can produce (usually a little less than the magnitude of the power supply voltages)?

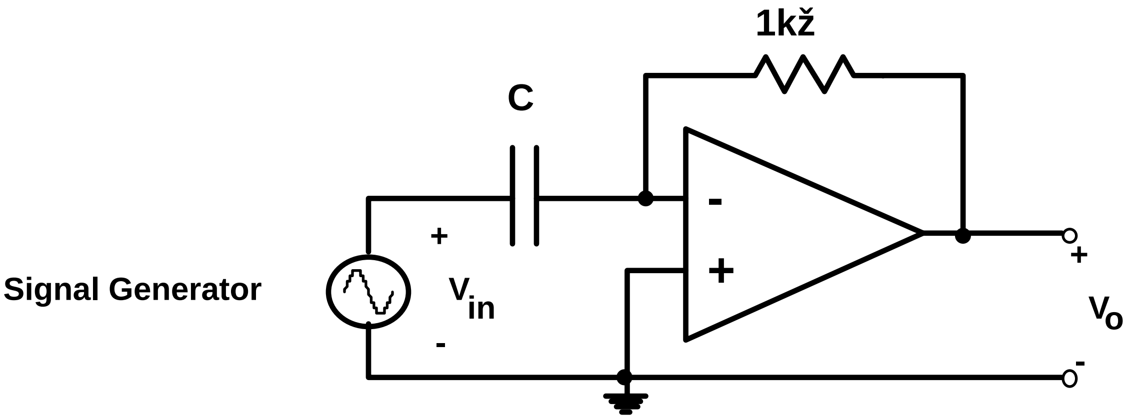


Figure 13.2

EXPERIMENT

Remember to use the multimeter and the inductance/capacitance meter to record the values for the resistors, capacitors, and inductors used in this experiment as you go along. Follow the meter directions.

(1) Connect the circuit shown in Figure 13.1 *but substitute the square-wave generator in place of the DC power supply and switch circuit*. Note the generator just happens to also have a 600 Ω internal impedance. The on/off action of the square wave generator also substitutes for the switch. Adjust the square wave output to **10 V P-P (peak to peak)**. Next, adjust the generator frequency so that the output reaches a maximum stable value during the positive half cycle. This should be a frequency=1/(10TC) Graph the voltage and current in the inductor as a function of time.

(2) Adjust the frequency to 10 times the setting from part 1 and graph the voltage across the inductor and the voltage across the 1kΩ resistor.

(3) Adjust the frequency to 1/10 of the setting from part 1 and again graph the voltage across the inductor and then the 1 kΩ resistor.

(4) Connect the circuit of Figure 13.2 (C=0.1 μF). Set the **square** wave generator to 0.5 V p-p and graph the output at 500 Hz. Change the input to a 1.0Vp-p **sine** wave and graph the output at 500 Hz, 5 kHz. and 50 kHz. Be sure the oscilloscope is set to DC coupling and observe the input on channel A and output on channel B. Also remember the op amp requires a positive and negative power supply (pin 7 pos, pin 4 neg, pins 1,5,&8 are not used).

(5) Exchange the position of the resistor and capacitor. What function should the circuit implement? Set the square wave generator to 1.0 Vp-p (before connecting to the OP AMP circuit) and graph the output at 100 Hz and 1 kHz. Then connect the largest resistor available in your kit across the capacitor and note what happpens.

RESULTS

(a) Determine the time constant of the inductor circuit using the current or voltage graphs from the experiment. Compare the hand calculations, SPICE simulations, and experimentally determined values of the time constant. Discuss any **significant** differences or possibilities of error. What effect does the internal resistance of the inductor have on the time constant?

(b) Why are RL circuits seldom used, except in radio frequency circuits?

(c) Use both PSpice and lab results of parts 1, 2, and 3 to discuss when integration or differentiation effects were observed and under what conditions. Remember that the period T (of the square wave) =1/frequency.

(d) Present your results for the RC op amp circuits. What is the effect of the internal 600Ω resistance of the signal generator?

(e) Discuss why there is a DC voltage present in the integrating circuit (unlike the output graph shown in the textbook for the integrating circuit).

(f) Discuss how frequency, input amplitude, values of R_1 and C, and the op amp affect the differentiator/integrator output. Are there frequency limitations of these simple integrating/differentiating op amp circuits?

(g) If you were required to re-write the steps of this experiment, what would *you* do differently?

Revised 7/94