

BASIC CIRCUIT ANALYSIS

ELEC 273

LABORATORY MANUAL

Originally prepared by N. Suresh

Basic Circuits & Systems Laboratory
Department of Electrical & Computer Engineering
Concordia University

April 2018

**PLEASE READ IMPORTANT INFORMATION REGARDING
EMERGENCY PROCEDURES ON THE LAST PAGE**

Safety

Engineers are often required to use hand and power tools in constructing prototypes or in setting up experiments. Specifically, electrical engineers use test instruments to measure the electrical characteristics of components, devices, and electronics systems.

These tasks are interesting and challenging, but they may also involve certain hazards if one is careless in his/her work habits. It is therefore essential that students learn the principles of safety at the very beginning of their career and that they practice these principles.

Safe work requires a careful and deliberate approach to each task. Before undertaking an experiment, students must understand what to do and how to do it. They must plan everything, setting out tools, equipment, and instruments on the workbench in a neat and orderly fashion. Extraneous items should be removed, and all cables should be securely fastened.

General Safety Rules

The first rule of personal safety is always:

Think First!

This rule applies to all industrial workers as well as to those working with electricity. Develop good habits of workmanship. Learn to use tools correctly and safely. Always study the job at hand and think through your procedures, your methods, and the applications of tools, instruments, and machines before searching. Never permit yourself to be distracted from your work, and never distract another worker engaged in hazardous work. **Don't be a clown!** Jokes are fun and so is "horsing around", but not near moving machinery or electricity. There are generally three kinds of accidents which may occur to electrical students and technicians- electric shock, burns, and equipment-related injuries. You're knowing and studying about them, and observing simple rules will make you a safe person to work with. You could personally be saved from painful and expensive experiences.

Electric shocks

What about electric shocks? Are they fatal? The physiological effects of electric currents can generally be predicted with the chart shown in Fig. 1:

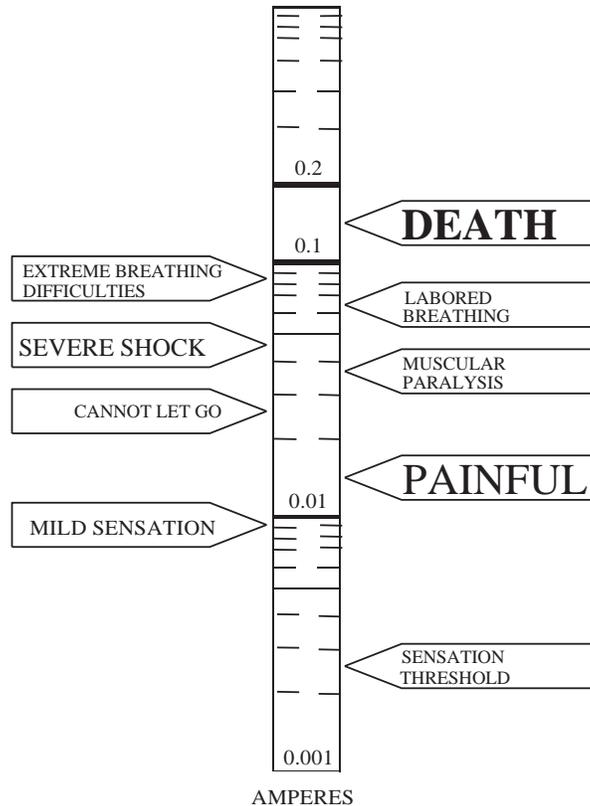


Fig. 1 Physiological effects of electrical currents.

Notice that it is the current that does the damage. Currents above 100 mA, or only one tenth of an ampere, are fatal. A workman who has contacted currents greater than 200 mA may live to see another day if given rapid treatment. Currents less than 100 mA can be serious and painful. A safe rule: **Do not place yourself in a position to get any kind of shock.**

Nine rules for safe practice and avoiding electric shocks:

1. **Work with one hand behind you or in your pocket.** A current between two hands crosses your heart and can be more lethal than a current from hand to foot. A wise technician always works with one hand. Watch your TV serviceman.
2. Be sure of the condition of the equipment and the dangers it can present **before** working on it. Many sportsmen are killed by supposedly unloaded guns; many technicians are killed by supposedly "dead" circuits.
3. **Never** rely on safety devices such as fuses, relays, and interlock systems to protect you. They may not be working and may fail to protect you when most needed.
4. **Never** remove the grounding prong of a three-wire plug. This eliminates the grounding feature of the equipment making it a potential shock hazard.
5. **Do not work on a cluttered bench.** A disorganized mess of connecting leads, components and tools only leads to careless thinking, short circuits, shocks, and accidents. Develop systematized and organized work habits.
6. **Do not work on wet floors.** Your contact resistance to ground is greatly reduced on a wet floor. Work on a rubber mat or an insulated floor.
7. **Do not work alone.** It is just good sense to have someone around to shut off the power, to give artificial respiration, or to call a doctor.
8. **Never talk to anyone while working.** Do not let yourself be distracted. Also, don't talk to someone who is working on dangerous equipment. Do not be the cause of an accident.
9. **Always move slowly** working around electrical circuits. Violent and rapid movements lead to accidental short circuits and shocks.

Burns

Accidents caused by burns, although usually not fatal, can be painfully serious. The dissipation of electrical energy produces heat.

Four rules for safe practice and avoiding burns:

1. **Resistors get very hot**, especially those that carry high currents such as the ones in this lab. Watch those five- and ten-watt resistors. They will burn the skin of your fingers. Stay away from them until they cool down.
2. Be on guard for all **capacitors which may still retain charges**. Not only can you get a dangerous and sometimes fatal shock, you may also get a burn from an electrical discharge. If the rated voltage of electrolytic capacitors is exceeded or their polarities reversed they may get very hot and may actually burst.

General Laboratory Rules

1. Students must treat the lab equipment, their colleagues, and the Lab Demonstrator TA with due respect. No containers of liquids (including water) are allowed on the lab benches. Large personal articles including coats, jackets, bags, etc., should be left under the desk or at the front of the room if necessary. Eating or drinking is strictly forbidden in the lab by university regulations. Telephones should be silenced and no calls are permitted in respect of your colleagues.
2. Every student must prepare for each experiment and must actively participate in performing the experiment. Do not waste your partner's and the Lab Demonstrator's time by being unprepared for the lab. Note that the function of the Lab Demonstrator is to assist and to answer technical questions, but not to perform the experiment for you.
3. The lab bench workspace must be kept clean. Scrap paper, pencil and eraser shavings must be cleaned up and deposited in the appropriate recycling or garbage can. *Disciplinary action will be taken against any student who misuses the lab, such as making marks on lab benches or on equipment, braiding patch cords etc.* Students must not be a disturbance to others in the lab.
4. In the event of a fire or any emergency requiring evacuation of the building, students must follow the Lab Demonstrator's directives and immediately leave the building via the stairwells whose locations are indicated on a map posted in the lab. The emergency procedures are reproduced at the end of this manual for reference.
5. Any apparently faulty equipment should be reported to the Lab Demonstrator. Under no circumstances are students allowed to try to repair any of the lab equipment.
6. All instruments switched on by the student should be switched off at the end of the session. Each station has a designated patch-cord rack and any cords used by the student must be returned to the designated rack. Do not remove any connections that were already made and which you have been instructed not to remove.

CONTENTS

		Pages
Preliminary Information	Lab Rules, Report Format, Grading Scheme, Missed Lab Procedure, Computer Simulation Requirements	2 - 8
Experiment # 1	Familiarization With Lab Instruments Basic Measuring Techniques using the Digital Multimeter, Function Generator and Digital Storage Oscilloscope	9 - 22
Experiment # 2	Circuit Theorems Verification of Superposition, Thevenin, Norton and Maximum Power Transfer Theorems	23 - 36
Experiment # 3	Operational Amplifier Circuits Investigation of Operational Amplifier Circuits with DC and AC signals	37 - 48
Experiment # 4	Dynamic Response of R-C, R-L and R-L-C Circuits Transient Response, Frequency Response, Resonance and Impedance	49 - 72
Experiment # 5	AC Power Measurements Measurements with resistive, inductive and capacitive loads, Power factor correction of a inductive load	73 - 82
<hr/>		
Appendices		
	A Descriptions of Measuring Equipment DMMs: Fluke 8010A, Agilent 34405A Function Generator: Instek GFG-8216A Digital Storage Oscilloscope: Tektronix TDS320	83 - 92
	B Operational Amplifier Basics, Description of the op-amp modules used in the laboratory	93 - 96
	C Introduction to Complex Numbers, Phasors and Impedance	97 - 102
	D Descriptions of equipment used in the ac power experiment	103 - 106
	E A brief note about SPICE simulation	107 - 108
	F Table of standard common electrical symbols	109 - 110
	Expectations of Originality Form, Emergency Instructions	111 - 114

Introduction

The laboratory room is H-822 in the Hall Building.

The purpose of the laboratory portion of the course is to:

- Illustrate the material presented in the ELEC 273 classroom by means of experiment,
- Enable the student to become familiar with basic electrical measurements and measuring instruments
- Give the student practice in engineering data analysis, presentation, report writing and teamwork.

Regarding the Schedule and Pre-lab Preparation

The topic of each laboratory experiment may be ahead of that being studied in the classroom. However the content is not beyond the understanding of an entry-level university student. The student must make a sincere attempt to understand the experiment description and procedure before coming to each lab. The experiment descriptions provided herein should be sufficient introductory material to conduct the experiment, with further relevant information found in the appendices. For best preparation students should additionally consult external references including the course text book or web as needed.

Pre-Lab Quiz

In order to promote the essential preparedness mentioned above, a pre-lab quiz is held during the first 15 minutes of the lab. The short quiz is typically a single question pertaining to the lab to be performed and counts for 10% of the total lab report grade. After the quiz, a brief lecture about the lab is provided by the Lab Demonstrator TA.

Lab Structure

The lab has ten bench stations with the same equipment. Students may work in pairs, but not groups of three unless allowed by the TA in cases where more than 20 students attend the session. Five experiments, listed under “Contents” at the beginning of this manual, will be performed in alternate week lab sessions (every two weeks) in the fall and winter terms, or weekly in the summer term.

Although working in groups of two, **each student must individually prepare and submit all lab reports.**

General Lab Procedure

Before the Lab

- Read the introductory material so as to be ready to conduct the lab efficiently in the available time.
- Consult external references if needed.
- Prepare for the pre-lab quiz.
- Print the pre-lab and Data Table sheets to bring to the lab if you wish to keep your lab manual intact.

During the Lab

- Submit your lab report from the previous lab.
- Arrive before the lab start time so as not to miss any time for the quiz.
- Perform the lab experiment and record all the required data into the provided Data Tables.
- Note observations you may want to investigate further or comment on in your report
- Obtain printouts where instructed.
- Obtain the Lab Demonstrator's signature on all your data and printouts.
- Leave the equipment, bench and lab room clean in accordance with the Lab Rules

After the Lab

- Prepare the report in accordance with the information that follows. It is due at the start of the next lab.

Lab Report Regulations

Lab reports must use a standard cover page available here:

http://users.encs.concordia.ca/~pbipin/ELEC273/lab_report_cover.pdf

All information on the cover page must be completed. The Expectations of Originality statement must be signed and dated. (Refer to Appendix for further detail.) Marks will be deducted for a report submitted without the cover page above, or if it is only partially filled.

The DATA TABLES (complete with the TA's signature) provided in this manual at the end of each experiment should be detached and included with the lab report. Alternatively the blank pre-lab and data table sheets can be obtained here:

<http://users.encs.concordia.ca/~pbipin/ELEC273/ELEC273LabManual/index273LabManual.html>

The rest of the report (Introduction, Procedure, Discussion, etc.) may be either word-processed or hand-written if legible. It is not necessary to write a detailed procedure as it is already stated here in the manual; it should be summarized as much as possible. Instead, the report should concentrate on meaningful Results, Discussion, and Conclusion sections to indicate that the

student has grasped the main concepts of the experiment. No photocopied data will be accepted. Additional computer-processed results such as graphs and spreadsheets could also be included.

Although student pairs will be expected to have similar lab results, no other part of the submitted report is expected to be shared between student bench pairs or class colleagues.

The paper report must be submitted to the Lab Demonstrator TA at the start of the next lab. No late submissions are accepted and no other manner of submission is possible. The marked report is returned to the student at the following lab session. The last report (#5) has a submission deadline of only one week. It must be submitted at the lab test or by a pre-arranged meeting. The graded last report may be retrieved at the end of the session from the Lab Coordinator's office. Unclaimed reports will be retained for one term (~ 3 months).

Lab Report Grading Scheme

Towards consistent marking across course sections, the Lab Demonstrator has been advised to use the following marking guidelines.

Objective, Introduction, Procedure	Brief, original, and well-worded	10 %
Results and Discussion	Comparison of experimental results with theoretical and simulation results. Should include error analysis, discussion on the agreement or non-agreement between experiment and theory	50 %
Conclusion	Brief and meaningful summary of what was learned from the experiment. Appropriate to mention particular observations here.	20 %
Presentation	Cover page is included and complete, signed Expectation of Originality statement, content is well-organized	20 %

Lab Test

A mandatory lab test is conducted during the regular lab time in the week following the fifth experiment. Students perform this test individually. Towards the end of the term, the test schedule is announced by email and posted at the lab room. The 45-minute test involves measurements and or calculations that the student should be familiar with from one of the experiments. The lab manual and an ENCS-approved calculator are the only permitted materials. Late arriving students do not receive extra time.

Missed Labs & Make-Up Procedure

Since a lab grade (combination of reports and test) of at least 50% is required to pass the course, it is advisable to avoid missing any labs.

If the student foresees that they will not be able to attend their regular lab section, they should contact the Lab Coordinator as soon as possible to inquire about temporarily joining another section if space permits. (Always include your name, student ID and course section code in your communications.)

If the student has already missed their section, they should also contact the Lab Coordinator to inquire about any make-up possibility.

A student who is authorized to attend another section, such as in the cases above, should submit their materials (pre-lab or report) to the TA present. The submission must show:

- their registered section code, and
- the code of the section they attended instead, with
- the name of the Lab Demonstrator TA it was submitted to.

In the case of unavoidable grave circumstance such as illness or accident, an absence may be disregarded in calculating the lab grade provided that an authentic document (doctor's or hospital certificate, police report, etc.) is furnished.

Computer Simulation Requirements

Comparison of the experimental data with results obtained from a SPICE simulation of the circuit tested in each experiment is a requirement for this lab (see ‘Marking Scheme’ above). SPICE (Simulation Program with Integrated Circuit Emphasis) is a standard circuit simulation software, originally developed at the University of California at Berkeley. Numerous commercial circuit simulation program packages based on SPICE are available for use on the various computer platforms. The “schematic capture” versions of the program in which a graphical user interface is used to provide the input in graphical form, have been developed. In these versions, no programming knowledge is required. The cursor is used to “drag and drop” circuit components (using their schematic symbol) from available “device libraries” on to a workspace in order to assemble a virtual circuit. Pull-down menus are then used to set all the device and simulation parameters and to select the desired output(s). Knowledge of SPICE is NOT a substitute for learning basic circuit analysis techniques. However, such on-screen wiring of circuits may be helpful to the student in learning topological facts of circuits, such as series and parallel connections, current directions, and voltage polarities. Simulation can confirm lab results as well as provide answers to problems.

Students are required to include SPICE-simulated results for lab circuits in experiments 2 - 5, although SPICE is not a part of the course material. The exposure gained will be beneficial since simulation will be formally required in later courses. Any available SPICE software may be used for your lab simulations. Many companies provide free demo versions or time-limited trial versions. Many sell a low-cost academic or hobby version. These demonstration version are usually limited in functionality, for example, the number of circuit components used per design is limited, or the ability to save or print may be restricted. However, these should still be adequate for the circuits encountered in this course. If the save or print function is disabled, typically it is still possible to take a screen shot (print screen) of the relevant display for use in the lab report.

Although the terms and commands may vary depending upon the exact software used, there are some terms used to define the SPICE excitation waveforms as well as analysis commands that have common features. These are given in Appendix E.

Regarding Schematic Diagrams

As future professional electrical engineers, you are obliged to be able to properly draw and read electrical circuit schematic diagrams. The symbols used in circuit diagrams have been standardized by international and North American organizations such as the International Electro-technical Commission (IEC) and the Institute of Electrical & Electronics Engineers (IEEE), respectively. Students are urged to learn about these standards by reading the “IEEE Standard 315-1975 (Reaffirmed 1993)” and associated documents that can be found online. Some basic standard symbols are given in the Appendix F. Students should practice drawing diagrams properly in their lab reports as well as in their other course submissions.

EXPERIMENT 1

FAMILIARIZATION WITH LAB INSTRUMENTS

OBJECTIVE : To become familiar with the measuring instruments and other equipment used in the lab: Digital Multimeters (DMMs), Function Generator (FG), Digital- Storage-Oscilloscope (DSO) , DC Power supplies .

PRE-REQUISITES: 1. Elementary knowledge of current and voltage variables in electrical circuits and of sinusoidal waveform terminology.
2. Reading and making an attempt to understand the tutorial material given below, before coming to the lab session.

PART (A) : TUTORIAL

Current and Voltage measurement: Both DC and AC current and voltage and measurements are performed in the lab using the Fluke Model 8010A and the Agilent Model 34405A digital multimeters (DMMs) which closely approximate ideal meters. Ideal ammeters and voltmeters, respectively represented by the circular symbols (with the letters A and V within) shown in Figure 1.1 below, do not disturb circuit conditions when they are connected. The ‘ideal ammeter’, when inserted in the path of the current being measured, behaves like a ‘short circuit’ (ie as if no disruption was made by the insertion). The ideal voltmeter, when connected to the terminals across which the voltage difference is being measured, does not draw any current and is equivalent to an ‘open-circuit’. For example, the front panel connections for the 8010A and 34405A DMMs, in the ammeter and voltmeter modes respectively , and their circuit equivalents, are also shown in the figure.

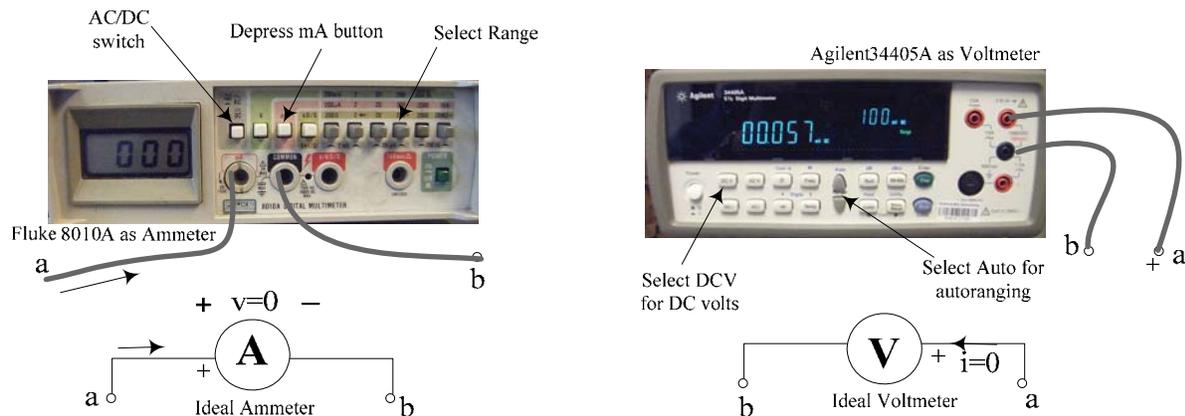


Figure 1.1: Ideal ammeter and voltmeter symbols and their corresponding equivalent connections and settings for the Fluke 8010A and Agilent 34405A DMMs

Resistance measurement: DMMs are also capable of directly measuring other quantities such as resistance, conductance, capacitance, frequency etc. The ‘Ohm-meter’-function is particularly useful to measure the resistance of an isolated resistor as well as for checking open-circuits and short-circuits (continuity checks). For example, the Fluke 8010A set to measure R is shown in Figure 1.2.

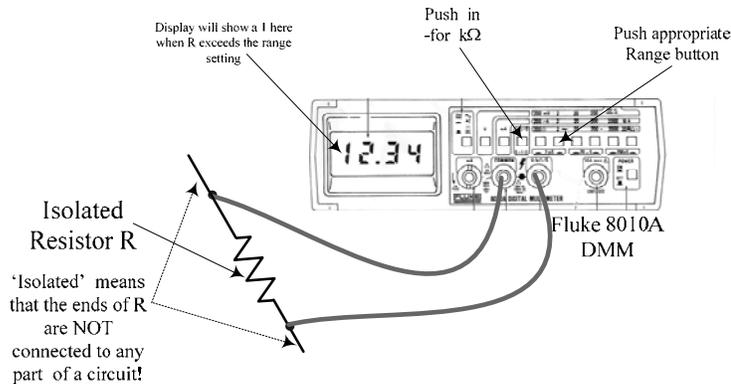
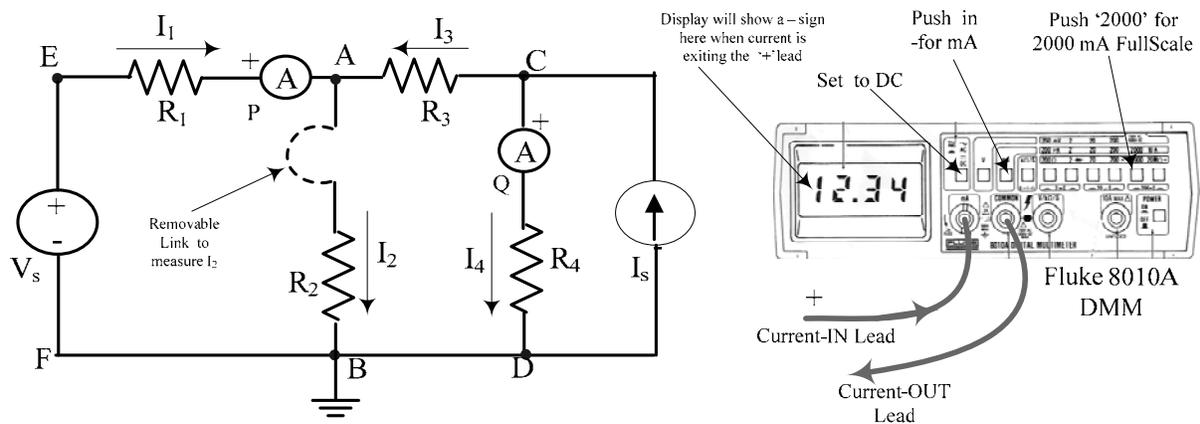


Figure 1.2: Fluke 8010A DMM in the Ohmmeter mode

The DMM terminals used for R measurement are usually the same as those used for voltage. **To avoid possible damage, a DMM should never be left in the Ohmmeter-mode, after use.**

The Agilent 34405A can additionally measure capacitance, frequency and temperature (using a ‘thermistor’ temperature probe). Another distinction between the above two meters is that the various measurement ranges in the 8010A are manually selected whereas the 34405A is an auto-ranging DMM.

Current measurement: The current in any circuit branch (ie through the element constituting that branch) is obtained by inserting an ammeter in series with the branch. It is therefore necessary to break a connection to obtain a current measurement. For example, consider the circuit shown in Figure 1.3. Ammeters P and Q are shown inserted in branches EA and CD to measure the currents I_1 and I_4 respectively.



Ammeter P is connected to measure the current I_1
 Ammeter Q is connected to measure the current I_4

Figure 1.3: DC current measurement [using the Fluke 8010A as example]

To facilitate the ammeter insertion in any branch, a wire-link such as the dotted link shown in the branch AB, may be provided when constructing the circuit. It is then a simple step to remove each link and substitute an ammeter to measure the current at that particular point. The ammeter must be inserted in the proper reference direction to show the correct sign in the display : In the 8010A DMM, if the current leaves the '+' input lead, a '-' sign will be displayed ahead of the numerical value. To measure AC currents, the same terminals are used but the AC/DC mode button is pressed in for AC.

When used in the 'ammeter' mode, DMMs usually have a fuse incorporated for over-current protection. If no indication is obtained in the 'ammeter' mode, this fuse is most probably burnt and must be replaced (by your lab instructor). In the 34405A a separate front-panel 1.25 Amp fuse is used for over-current protection. In the 8010A, a 2A fuse is contained within the '+' current input terminal, which is a special part [**Please note: It is important to make sure that this part is always re-inserted in the DMM whenever it is taken out as it is not a replaceable component !**]

To avoid possible damage to itself as well as to other circuits, a DMM should never be left in the ammeter-mode, after use.

Voltage definitions & measurement: A 'branch voltage' is defined as the voltage across the element constituting the circuit branch. It can be measured by connecting a voltmeter in parallel with the branch. For example in the same circuit as above, shown again in Figure 1.4, voltmeter M is connected to measure the branch voltage between nodes 'A' and 'C', designated by the double-subscripted quantity V_{AC} , where node 'A' is taken to be positive with respect to node 'C', by convention. Branch voltages can be more conveniently obtained by measuring the node voltages, which are the voltages between every node and an arbitrarily-chosen reference node, called the 'ground' [In Figure 1.4 node 'B' is indicated as being the 'ground' by its connection to the associated ground symbol.] Node voltages are easily measured by 'probing' every node using a 'grounded' voltmeter such as voltmeter N in Figure 1.4,

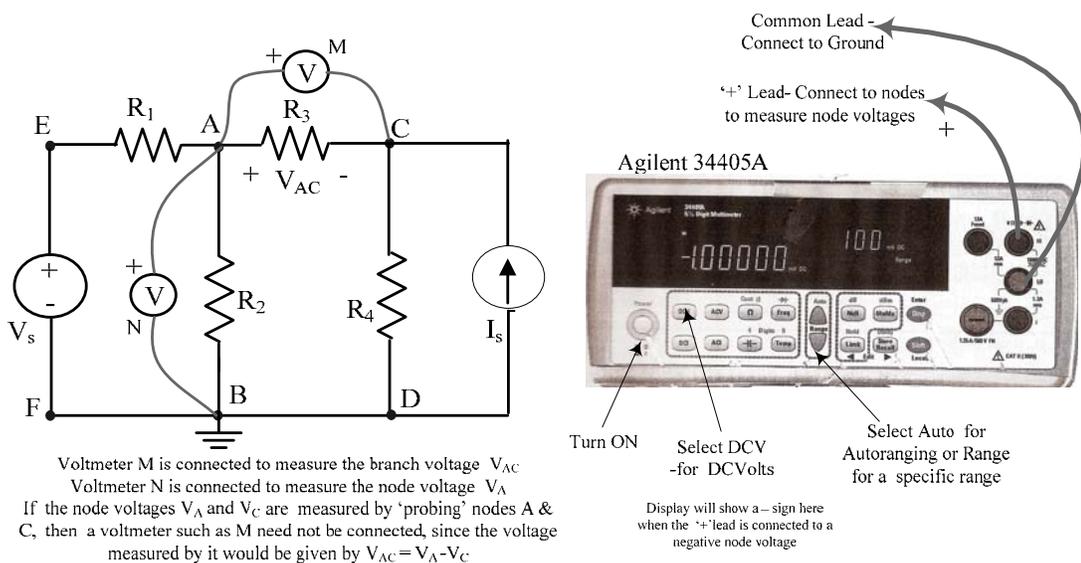


Figure 1.4 : DC voltage measurement [using the Agilent 34405A as example]

Node voltages are written as single-subscripted voltages such as V_A , V_C , V_E etc., where it is understood that $V_A = V_{AB}$, $V_C = V_{CB}$, $V_E = V_{EB}$ etc. when 'B' is the chosen ground. Using Kirchoff's Voltage Law (KVL) it can be shown that, for any 'non-grounded' branch such as AC, the branch voltage V_{AC} is given by the difference between the corresponding node voltages, i.e. $V_{AC} = V_A - V_C$ or $V_{CA} = V_C - V_A$. The potential-difference between any pair of nodes, even if there is no element connected between the two nodes, can be found by measuring the node voltages. Node voltages can have positive or negative values. In measuring node voltages, it is important to ground the negative input terminal of the voltmeter so that the correct sign of the node voltage will be displayed ahead of the numerical value.

DC Voltage & Current Supplies: The Power Supply Panel (PSP) provides DC source voltages and currents needed in the experiments. It is located on top of the lab station hutch at its centre. The PSP is energized by turning on a master power supply (the black-coloured unit located on the centre pillar of the hutch) and it is active when the green PWR ON indicator light is glowing. The front panel of the PSP is shown sketched in Figure 1.5 and a photograph of it is seen in Figure 1.6. The PSP provides a 1.1 volt -to-10 volt continuously variable voltage source and a 5.6-to-50 mA continuously variable current source, each varied by means of a 10-turn control knob.

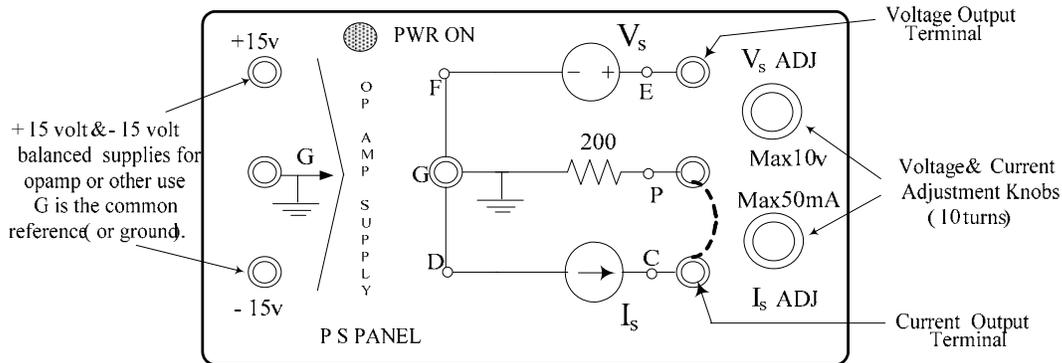


Figure 1.5: Front view of the PSP

The variable current source may be converted into an additional variable voltage source, if necessary, by bridging terminals P and C (shown as a dotted line between P & C in Figures 1.5 & 1.6) using a short patch cord. The PSP also provides 'balanced' +15v / 0 / -15v power supplies required in experiments involving Operational Amplifiers. All the PSP outputs are referenced to the (black) ground terminals labelled G (for Ground)



Figure 1.6: Front view photograph of the PSP

Experimental Circuit for DC experiments: The resistor circuit shown in Figures 1.3 and 1.4 is available in the lab, constructed on an old-fashioned aluminum chassis ('R-Chassis') shown in Figure 1.7. It is used in this experiment for DC measurements. The actual circuit connections between the various terminals on the chassis can be easily examined visually. The circuit diagram of the R-chassis with the resistance values used, is pasted on the top side of the chassis, as seen in the photograph. The resistor values as well as the patch cord connections to be made, including current-measurement links (around the node A), are also shown in the figure. The value of R_L in the circuit is variable from 100Ω to 300Ω using a 10-turn potentiometer (20Ω per turn starting from 100Ω)

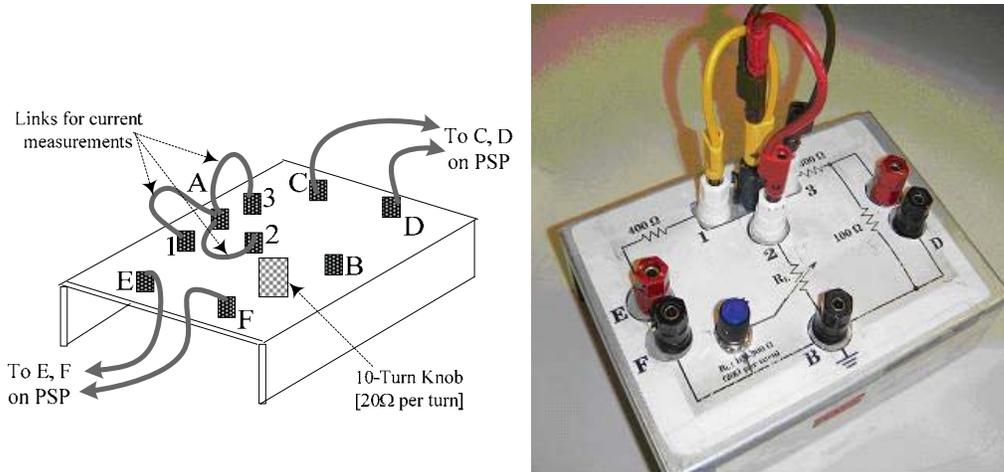


Figure 1.7 : The 'R-Chassis'

The complete circuit used for DC measurements, showing the exciting voltage and current sources (from the PSP), but not the DMMs, is seen in Figure 1.8. With node B(F,D) used as the ground, V_s and I_s are first set to specific known values. The node voltages V_E, V_A, V_C are then successively measured using the Agilent DMM and the currents I_1, I_2, I_3 are successively measured using the Fluke DMM.

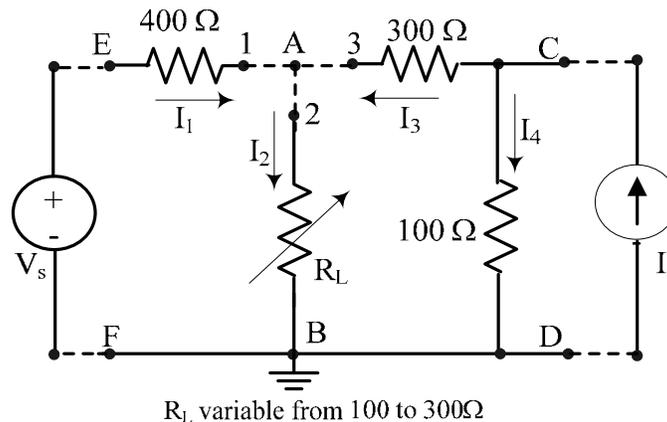


Figure 1.8 : The circuit used for DC measurements

AC (sinusoidal) waveform generation, display & measurement: Periodic voltage waveforms such as square-wave, triangular-wave, sinusoidal and pulse signals produced by electronic function generators (FG) are used as test input signals. The Instek GFG-8216A Function Generator used in the lab can provide sinusoidal (& square-wave) signals upto a frequency of 3 MHz. The signals are displayed and measured using a Tektronix TDS320 Digital Storage Oscilloscope (DSO). Details of both these instruments, such as the function of each button or knob, are given in the Appendix, which the student should read for maximum benefit.

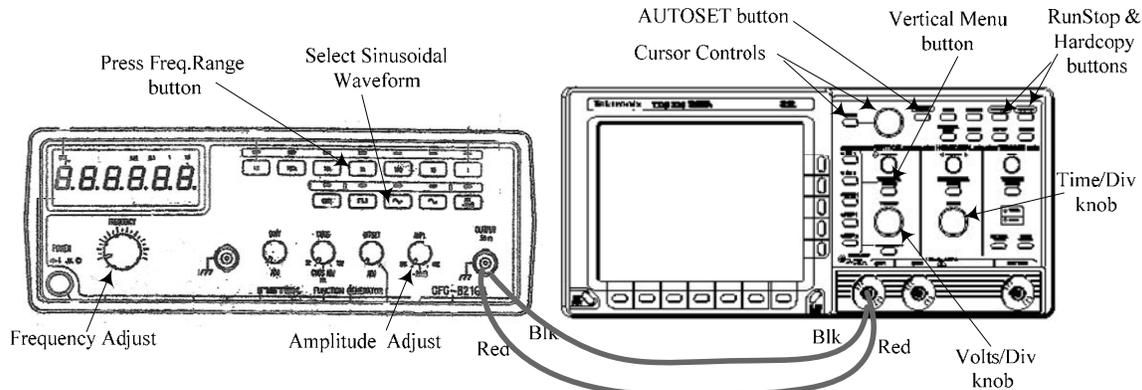


Figure 1.9: Main FG & DSO controls identification & basic interconnection

The student should first make the direct connection shown in Figure 1.9 between the FG and DSO (using banana-plug patch cords available in the lab), switch both instruments on, and obtain a waveform display on the DSO: Pressing CH1 or CH2 on the DSO (on the group of buttons to the left of the VERTICAL MENU button) will cause the corresponding trace to appear. Then pressing the AUTOSET button will produce a display which is ‘optimally sized’ for the applied signal level. Any given trace may be removed by selecting it and then pressing the ‘WAVEFORM OFF’ button. The DC zero level of each trace is indicated by the small arrowhead along the left edge of the graticule. The student should investigate the effects of manipulating the main FG controls identified in Figure 1.9, on the waveform observed on the DSO. The student should also ‘experiment’ with the DSO controls: When the VERTICAL MENU button on the DSO is pressed, various options such as AC/DC input coupling, channel inversion, full or 20 MHz bandwidth, Finescale, DC offset become available for the selected trace. These options are selected using the horizontal and vertical rows of buttons that border the screen, as well as the GP knob. The DSO also provides two reference channels (to store waveforms for comparison purposes) and a ‘MATH’ channel which displays the result of an arithmetical operation performed on the two active channel signals, such as $M = \text{Ch1} - \text{Ch2}$.

AUTOMATIC MEASUREMENT: In addition to displaying waveforms, the DSO can automatically measure periodic-signal parameters such as peak or RMS amplitudes, Period and Frequency. In the above setup, although the FG has its own frequency display, it is preferably to use it only as a guide, and to obtain the actual frequency from a DSO measurement.

To investigate the automatic periodic-signal measurement capability of the DSO, obtain a **sinusoidal** display showing a few cycles of the waveform: Press the MEASURE button on the DSO and then press the “Select Measurement for CH1” on the bottom menu buttons. Measurement choices appear alongside the vertical menu buttons. Press the last button

(‘More 1 of 6’) to see other measurement choices. The onscreen measurement readings will appear within the graticule area, but they can be moved off screen to the right of the graticule (by pressing CLEAR MENU) before a printout is made. **The indicated ‘measured’ values are the correct values to be used, as they are averaged over several cycles of the signal.**

CURSOR MEASUREMENTS: By pressing the CURSORS button, manually adjustable Vertical, Horizontal & Paired ‘cursors’ are available for the measurement of time-difference, voltage-difference or both, respectively. In particular, the V-bar cursors can be used to find the time-difference between two reference points which are on the same waveform or on different waveforms. The ‘paired’ cursors, which have tiny H-bars ‘riding’ on the waveform at the location of the V-bars can only be used on a single waveform. **To activate and use the cursors,** press CURSORS. A choice of H-bar (to measure voltage differences), V-bar (to measure time differences) or Paired Cursors (to measure both voltage and time differences) is available. The cursors are selected and positioned using the TOGGLE button and the GP knob, respectively, the solid-line cursor being the ‘active’ one. The above ΔV or Δt differences measured by the cursors appear on the top-right end of the screen, but may be moved off-screen by pressing CLEAR MENU. A typical display showing automated measurements of Peak-to-Peak and RMS magnitudes, Period, Frequency and Paired-cursor data is shown in Figure 1.10.

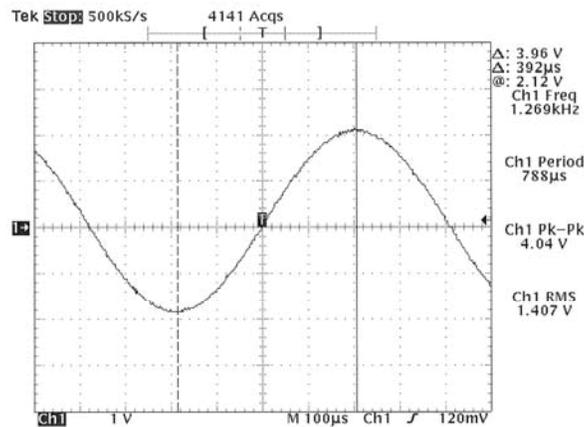


Figure 1.10: Typical display showing automated measured values and Paired- cursor data. [The small arrowhead appearing along the left edge of the graticule indicates the zero-position of each trace.]

The cursor data “@ 2.12V” indicates that the voltage level on the waveform at the location of the solid-line(active) cursor is 2.12 volts. Note that paired cursors can only be used on a single waveform.

DATA PRINTOUT: Every DSO in the lab is interfaced through a serial-data switch to the common printer. To obtain printout of a display, first store (‘freeze’) the display by pressing RUN/STOP and then press HARDCOPY. **The printer is programmed to print two copies of the stored display, together with station-identification.**

Experimental Circuit for AC experiments: An aluminum chassis (the ‘RLC Chassis’) on which R, L and C elements are mounted, as shown in Figure 1.11, is used in the lab for familiarization with the use of the FG and the DSO in AC measurements.

The value of R in the circuit is variable from 100Ω to 10100Ω using a 10-turn potentiometer ($\sim 1\text{ k}\Omega$ per turn, starting from 100Ω). The nominal values of L & C are $L = 47\text{ mH}$ and $C = 22\text{ nF}$. An RLC-meter (located on the Printer table) can be used to measure exact values.

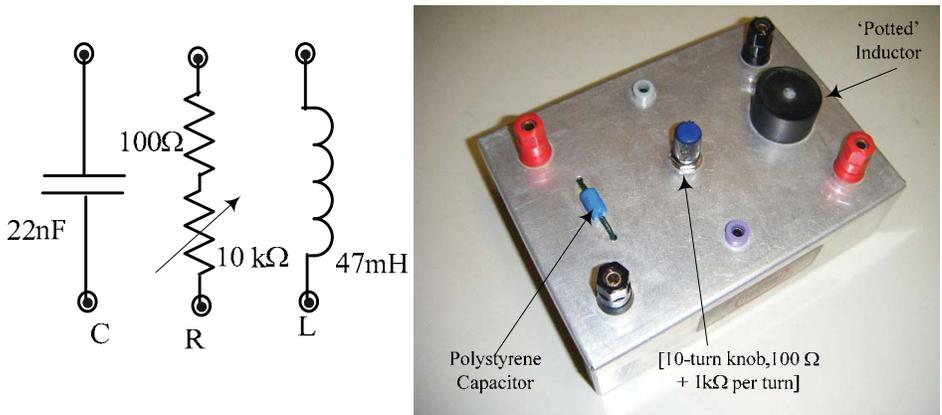


Figure 1.11 : RLC-Chassis and component values

PART (B): Procedure & Results

DC voltage & current measurements :

1. **Make sure that the PSP is turned OFF.** Make the patchcord connections shown in Figure 1.12 between the PSP and the R-Chassis using banana-plug patchcords of appropriate lengths and colours*

[* It is good practice to ‘colour-code’ your connections. Try to use different coloured cords and certain ‘standard’ colour-associations., eg Black or Brown for Ground , Red for ‘+’ power , and other colours for the other interconnections.]

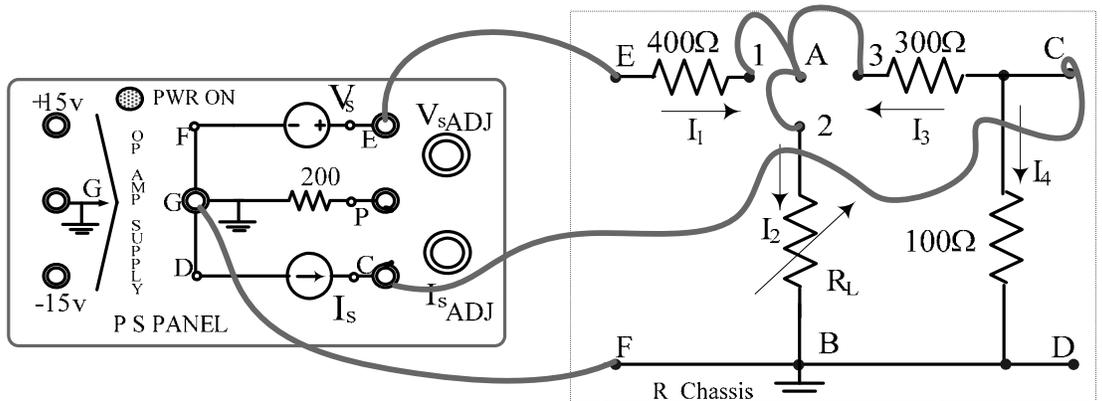


Figure 1.12

2. Set the value of R_L to a known value between 200Ω to 300Ω and then **turn the PSP ON**. Set the 'V_s ADJ' and 'I_s ADJ' knobs at 4 -to-8 turns from the extreme CCW (stop) position. This will result in V_s and I_s outputs in the range of 4~9 Volts and 8~9 mA , respectively.
3. Set the Fluke 8010A DMM as an ammeter in the 200 mA DC range Measure the branch currents I₁, I₂ and I₃ (by successively replacing the links around terminal A with the ammeter connected in the proper direction) and enter their values in the data tables provided. [Recall the current measurement method given earlier in Figure 1.3]
4. Set the Agilent 34405A DMM to the DCV-Auto mode and measure the node voltages V_E, V_A and V_C [with F(B,D) as ground] and enter their values in the LAB RECORD. [Recall the node voltage measurements given earlier in Figure 1.4]
5. **Results (DC) :**
 - (a) Use the current measurements of Step 3 to verify the KCL equation : $I_2 = I_1 + I_3$
 - (b) Use the node voltage measurements of Step 4 to obtain the branch voltages V_{EA}, V_{AB}, V_{AC} and V_{CD}; hence verify the validity of the KVL relations :

$$V_{EF} = V_{EA} + V_{AB} = V_{EA} + V_{AC} + V_{CD} \quad \text{and} \quad V_{AC} + V_{CD} + V_{BA} = 0.$$
 - (c) Use all the measured data (voltages & currents) to calculate I_s ,the current delivered by the current source.[Hint: $I_4 = V_c / 100, A$] and hence determine the total power P_{del} delivered by the PSP sources.
 - (d) Use all the measured data (voltages & currents) to calculate the total power dissipated P_{diss} by elements in the circuit and show that P_{diss} = P_{del} [of Step 5(c)] Give possible reasons for any difference observed in the two values of power.

=====

AC (sinusoidal) waveform measurements :

- Make the banana-plug patchcord connections shown in Figure 1.13 between the FG, the DSO and the RLC-Chassis using appropriate lengths and colours. The circuit obtained is a **series-RLC** circuit excited by a voltage source, with the voltage across C being the output voltage V_{out} . [Note that after connecting the FG to the DSO, as in Figure 1.9, all the other connections are made at the DSO terminals alone, since this results in minimum wiring clutter].

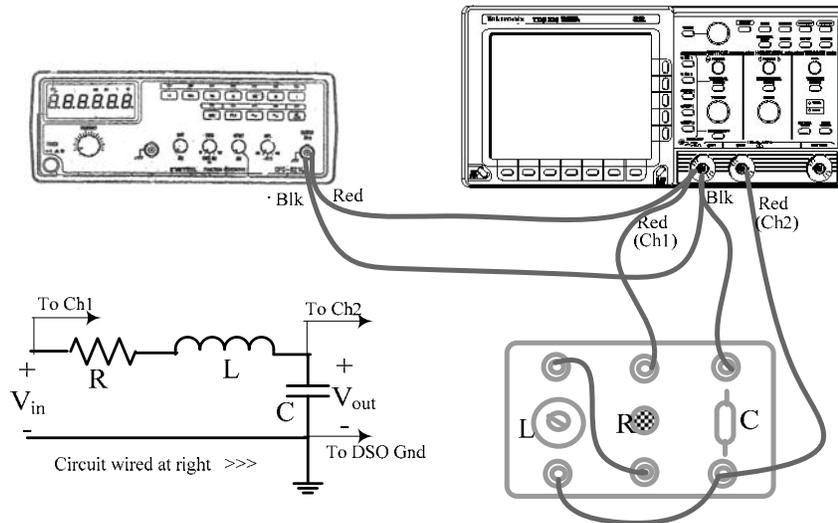


Figure 1.13

- Set the DSO to display Channels 1 and 2, with DC input coupling. Press the MEASURE button and successively select the following measurements to be displayed: Ch1 frequency and the RMS amplitudes of both channels. Also press CURSORS and select the V-bar cursors. Move the displayed values to the right by pressing CLEAR MENU.
- Set the FG output amplitude and frequency controls to display a sinusoidal Ch1 signal of about 2 volts RMS at a frequency of about 100 Hz. (Press AUTOSSET if necessary to obtain an optimally-sized display). Position the two traces so that there is no overlap. Set the R control on the RLC-Chassis to the minimum (ie 100 Ω).
- Vary the FG frequency over the range of 100 Hz to 10 kHz (as read on the DSO) and observe the changes in the output voltage (Ch2) amplitude and its phase, relative to the input signal. [It will be observed that significant changes in both voltage and phase-shift occur around and above the 'series resonant-frequency' given by $f_n = \frac{1}{2\pi\sqrt{LC}}$ Hz, which is approximately 5 kHz for the LC values used in the RLC-Chassis.].
- Obtain two typical printouts, one at $f_1 (\approx f_n)$, Hz and the other at $f_2 (\approx 1.5f_n)$, Hz. Both printouts must show the two RMS amplitude values and the V-bar cursors positioned to read the time-shift Δt between two adjacent waveform maxima (peaks) [or at the

minima (troughs), or at zero-crossings]. **The phase-shift ϕ is then given by $\phi = 360 f \Delta t$** , degrees, where f is in Hz. and Δt in sec. In each case, adjust the SEC/DIV knob until only one or two cycles of the waveform are displayed, so that the cursors may be positioned more precisely to find Δt .

A typical trace near the resonant frequency, with the cursors placed at the troughs, is shown in Figure 1.14.[For the waveforms in Figure 1.14, $V_{out}(Ch2)$ lags $V_{in}(Ch1)$ by $360(4866)(44.5 \times 10^{-6})$ degrees or $\sim 78^\circ$]

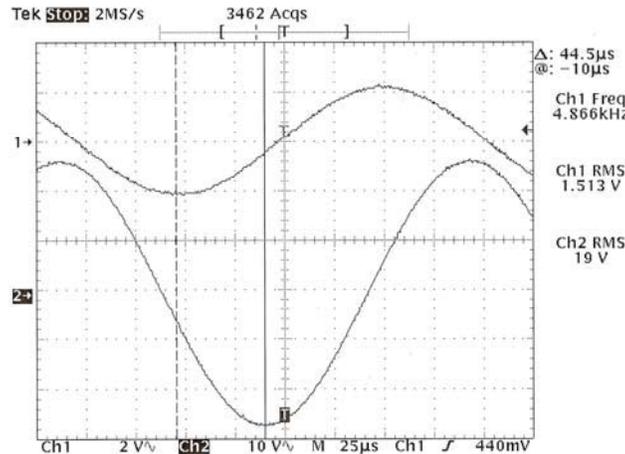


Figure 1.14

11. **Results (AC) :**

- The amplitude ratio V_{out}/V_{in} in Fig 1.14 is called the Voltage Gain A_v . From the two printouts of Step 10, obtain the values of A_v at the two frequencies f_1 & f_2 used.
- Calculate the phase shift ϕ (in degrees) at the above two frequencies, from the cursor readings.
- Compare the results of (a) and (b) with the corresponding theoretical values which are given by the following equations:

$$A_v = \frac{1}{\omega C \sqrt{R^2 + [\omega L - \frac{1}{\omega C}]^2}} \quad \text{and} \quad \phi = -90^\circ - \tan^{-1} \frac{(\omega L - \frac{1}{\omega C})}{R}, \text{ degrees.}$$

where $\omega = 2\pi f$ (radians/sec). Use the frequencies ω_1 and ω_2 and nominal values given for L & C given in Figure 1.11 and value used for R . Calculate the percent difference between the measured and theoretical values using the latter as reference.

Cursor measurements on dynamic responses

12. Next, press MEASURE and remove all measurements using the menu . Press CURSORS and select 'Paired Cursors'. Reset the FG output to obtain a square-wave Ch1 signal of about 6 volts peak-to-peak at a frequency of about 100 Hz. The waveform on Ch2 should also display a square-wave, but with an oscillatory part at the beginning of each half-cycle : Adjust the SEC/DIV knob until only the positive-going section of one cycle of the waveform is displayed, as shown in Figure 1.15 for example.

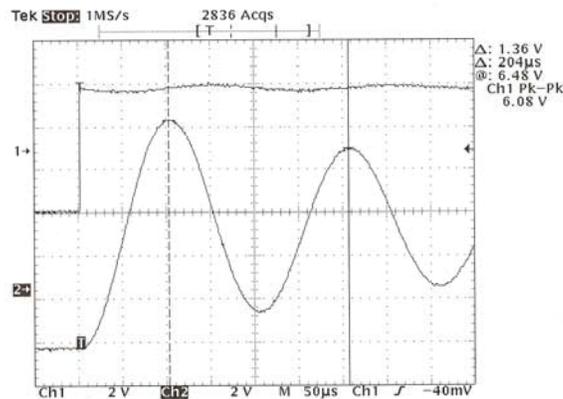


Figure 1.15

[The oscillatory part of the Ch2 signal in Figure 1.15 is called a 'damped sinusoidal response' (also known as the 'under-damped response' of a 'second-order circuit'). Step 13 below illustrates the measurement of typical parameters of such a response]

13. Position the paired cursors on the Ch2 trace and obtain printouts showing the following measurements :
- (a) Voltage between and time between the baseline and the first positive peak of the Oscillation.
 - (b) Voltage between and time T between the first two positive peaks of the oscillation [an example of this measurement is shown in Figure 1.15]
Determine the effective frequency of the damped signal, $f \text{ (Hz)} = 1/T \text{ (sec)}$

=====

DATA TABLES

[To be cut out & **pasted** into the Report booklet]

Step 2: Reference values used :

$R_L \approx \dots\dots\dots$ Ohms , V_s knob at $\approx \dots\dots\dots$ Turns, I_s knob at $\approx \dots\dots\dots$ Turns.

Step 3 : DC Currents:

$I_1 = \dots\dots\dots$ mA, $I_2 = \dots\dots\dots$ mA, $I_3 = \dots\dots\dots$ mA.

Step 4 : DC Node Voltages :

$V_A = \dots\dots\dots$ Volts, $V_E = \dots\dots\dots$ Volts, $V_C = \dots\dots\dots$ Volts.

TA Signature :

[Make sure to also obtain your TA's signature on the printouts of Steps 10 and 13]

BLANK

EXPERIMENT 2

CIRCUIT THEOREMS

OBJECTIVE : Experimental verification of the Superposition Theorem, the Thevenin & Norton Theorems and the Maximum Power Transfer Theorem

- PRE-REQUISITES**:
1. Complete familiarity with the use of the lab instruments, as acquired in Experiment #1 [**This familiarity on the part of the student will be assumed for all the remaining experiments**]
 2. Reading & making an attempt to understand the tutorial material given below, before coming to the lab session.

PART (A) : TUTORIAL

The theorems listed in the objective are very useful in simplifying the solution of problems in linear circuit analysis. The principles outlined by these theorems are simple and can be understood by reading the tutorial material which follows. In this experiment, the theorems will be verified for a resistive DC circuit, which is constructed on a breadboard by the student.

The Superposition Theorem applies to circuits which are excited by two or more independent sources, such as the one shown in Figure 2.1(a).

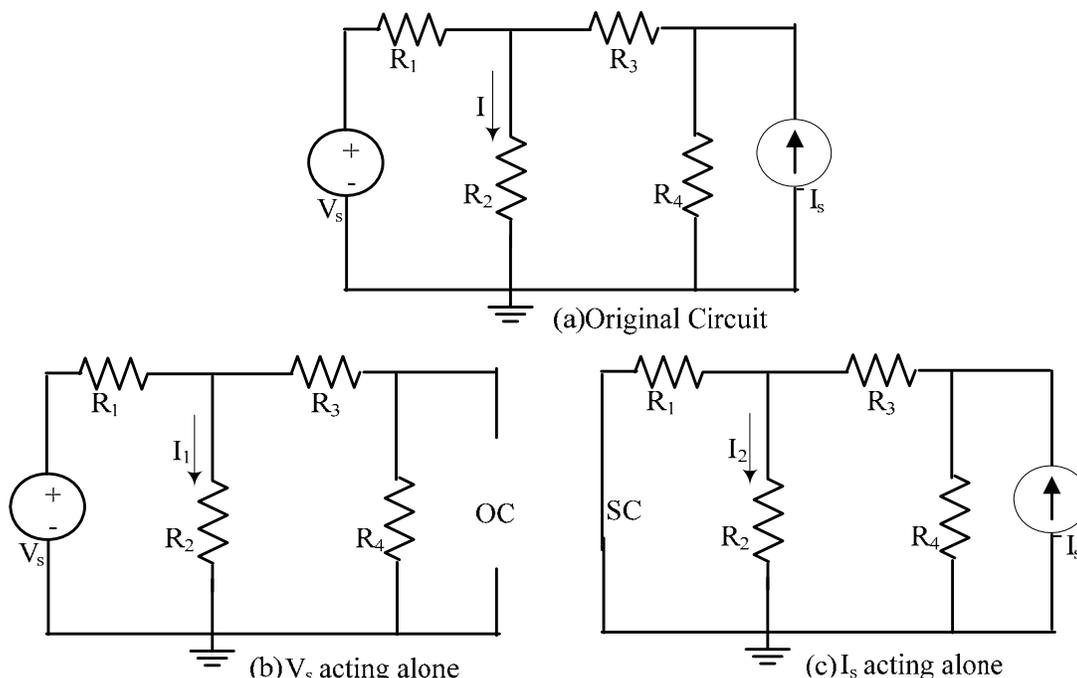


Figure 2.1

For such circuits, the theorem states that the current (or voltage) associated with any branch can be determined as the sum of the values of that branch current (or voltage) which will be produced by each of the exciting sources acting alone, with all other independent sources deactivated. For example the current I in Figure 2.1(a), is given by the sum of the currents I_1 & I_2 shown in Figures 2.1(b) & (c), respectively. The theorem simplifies the overall solution because each of the sub-circuits will contain only one independent source and will hence be simple to analyze using only basic steps such as voltage division, current division and Ohm's law.

Two facts must be noted in the application of the ‘de-activation’ of sources referred to above :

- (1) For each sub-circuit containing only one independent source, all other independent voltage sources are de-activated by replacing them with short-circuits (ie by setting their voltages to zero) and all other independent current sources are de-activated by replacing them with open-circuits (ie by setting their currents to zero),
- and
- (2) the source de-activations are only applicable to the independent sources : dependent-sources cannot be de-activated because setting their value to zero would be equivalent to constraining their controlling voltage or current to become zero, thereby changing the problem itself.

In view of (2) above, the use of superposition does not offer any simplification if the circuit to be analyzed contains dependent sources. The superposition principle is basic to all linear circuits and can be used as a test for circuit linearity.

Thevenin’s Theorem & Norton’s Theorem offer circuit simplification by replacing an entire portion of any given circuit, with a simple two-element equivalent circuit consisting of an independent source and a resistance. The essence of both theorems is graphically illustrated in Figure 2.2, which shows an (undefined) circuit, containing many sources, in which the problem is to find the voltage V_{ab} across a certain resistance R_L . According to Thevenin’s theorem, the remainder of the circuit connected to R_L can be replaced with a simple circuit [called the Thevenin Equivalent Circuit(TEC)] consisting of an independent voltage source V_T in series with a resistance R_T . According to Norton’s theorem, the same replacement referred to above, can be made with another simple circuit [called the Norton Equivalent Circuit(NEC)] consisting of an independent current source I_N in parallel with a conductance G_N . V_T and R_T are called the Thevenin Voltage Source and Thevenin Resistance respectively and I_N and G_N are called the Norton Current Source and Norton Conductance respectively. The resistance R_L associated with the current or voltage to be found is called the load resistance.

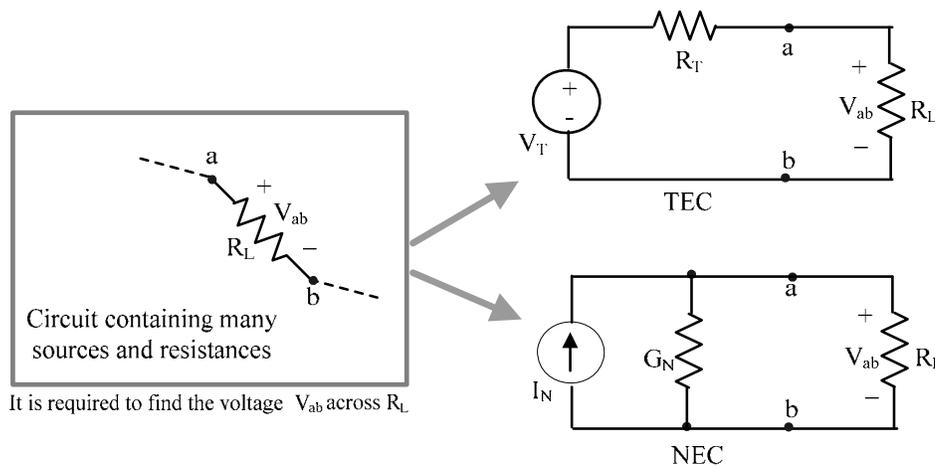


Figure 2.2: Graphical illustration of Thevenin’s & Norton’s theorems

When the circuit connected to R_L is replaced by a TEC or by a NEC, as shown in Figure 2.2, the required voltage V_{ab} is easily found using elementary methods such as voltage-division, current-

division, and/or Ohm's law : For example in the above circuits, $V_{ab} = V_T [R_L/(R_T+R_L)]$ by voltage-division, or $V_{ab} = I_N / G_{total} = I_N / (G_N + G_L)$ by Ohm's Law.

Thevenin's Theorem and Norton's Theorem can also be used to obtain the equivalent circuit across any given pair of terminals in a network, without specifying a 'load'.

It can be shown that the equivalent circuit parameters V_T (or I_N) and R_T (or G_N) may be obtained from only two measurements : the "open-circuit terminal voltage" $V_{ab}(oc)$, which is the voltage V_{ab} with R_L removed (disconnected) and the "short-circuit terminal current" $I_{ab}(sc)$, which is the current that would flow in a short-circuit placed between terminals 'a' and 'b'. This can be seen by considering the TEC and the NEC circuits in Figure 2.2. From the TEC, with terminals 'a' and 'b' open-circuited (ie with the load removed), the current, and hence the voltage-drop across R_T , will be zero thereby giving the result $V_T = V_{ab}(oc)$. When the terminals 'a' and 'b' are short-circuited, the current in the short circuit will be $I_{ab}(sc) = V_T/R_T$, and hence $R_T = V_T/I_{ab}(sc) = V_{ab}(oc)/I_{ab}(sc)$. From the NEC, $V_{ab}(oc)$ is also equal to I_N/G_N and $I_{ab}(sc) = I_N$, hence $G_N = I_{ab}(sc)/V_{ab}(oc) = 1/R_T$, ie R_T and G_N represent the same equivalent resistance element. Thus, only two measurements $V_{ab}(oc)$ and $I_{ab}(sc)$ are sufficient to determine both the TEC and the NEC.

In general, however, it is not advisable to attempt a practical measurement of $I_{ab}(sc)$, since its value could be destructively large if R_T (which is usually unknown *a priori*) happens to be very small. A safer alternative is to connect a calibrated variable resistance R_x to the terminals and then vary it (starting from a large value) until the terminal voltage V_x decreases to a value distinguishably less than the open-circuit value $V_{ab}(oc)$. Then, by voltage-division,

$$V_x = V_T \frac{R_x}{R_T + R_x} = V_{ab}(oc) \frac{R_x}{R_T + R_x}$$

Since the values of V_x , $V_{ab}(oc)$, and R_x are known, R_T can be calculated from the above equation. R_T can also be found by adjusting the calibrated resistance R_x until $V_{ab} = V_T/2 = V_{ab}(oc)/2$ which is the condition corresponding to $R_x = R_T$. This value of load resistance also results in maximum power being transferred to the load (see below)

The TEC and NEC are in fact the equivalent circuits of practical ('real') voltage and current sources, respectively and $R_T (= 1/G_N)$ is then called the *internal resistance* of the source. The internal resistance of a source is not a physically distinct component.

The Maximum-Power-Transfer (or MPT) Theorem states that the maximum power that can be delivered by a real source, whose TEC (or NEC) is known, to a load resistance R_L is given by $P_L(max) = (V_T)^2/4R_T = (I_N)^2/4G_N$ and that this power transfer to the load occurs when $R_L = R_T$. This result is derived by finding and setting $dP_L/dR_L = 0$ where $P_L = (V_L)^2/R_L = [V_T / (R_T + R_L)]^2 \cdot R_L$. Thus, for a variable load resistance R_L connected to a real source, the load power $P_L = (V_L)^2/R_L$ calculated and plotted against R_L , will show a maximum value when $R_L = R_T$. Since for $R_L = R_T$, $V_{ab} = V_T/2 = V_{ab}(oc)/2$, the load R_L required for MPT can be also found by adjusting its value until the load voltage V_L reaches the value $V_{ab}(oc)/2$.

PART (B) : EXPERIMENTAL:

The circuit theorems introduced above will be experimentally verified with reference to a five resistor network which will be built by the student on a breadboard. A modified Protoboard PB-60 breadboard system is available at each station along with sets of connecting (hook-up) wires and components. A photograph of the breadboard used is shown in Figure 2.3.

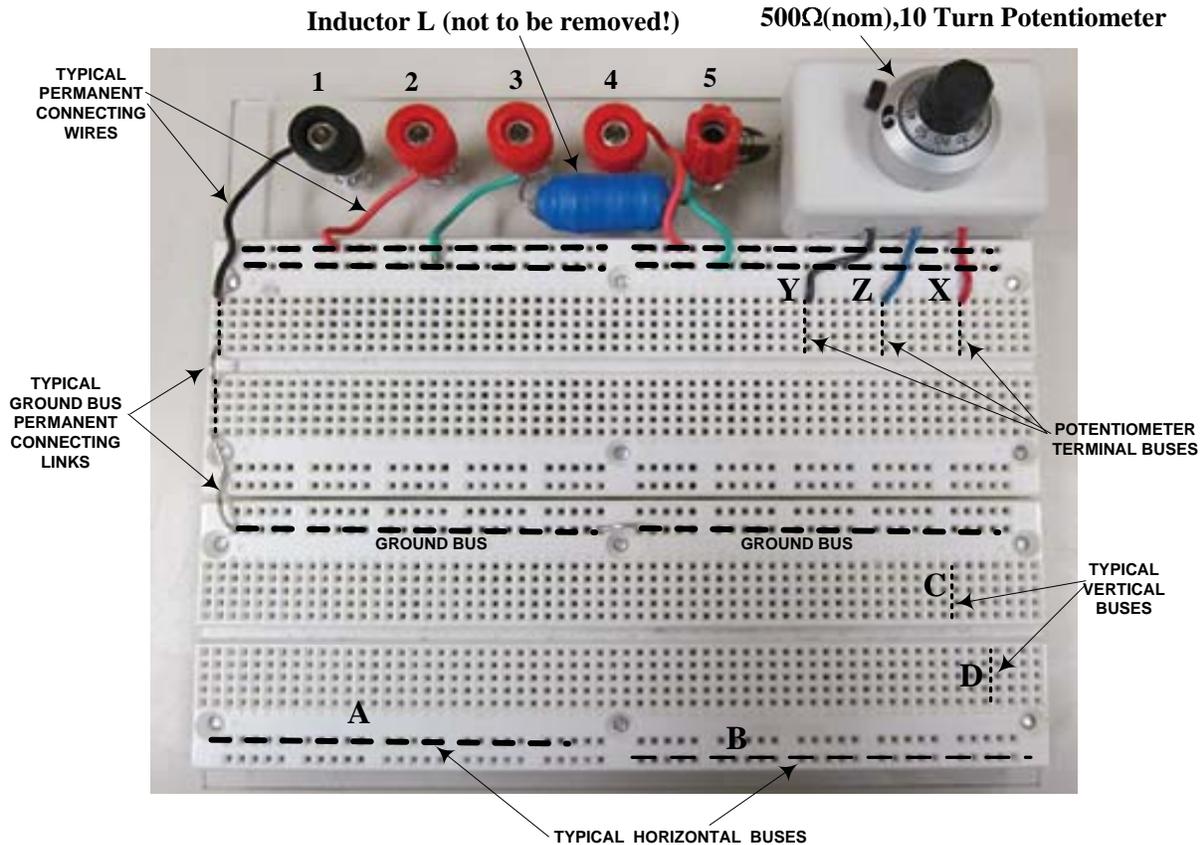


Figure 2.3 : The modified “Protoboard PB-60” breadboard system

The modification refers to the presence of an extra banana jack (#5) in addition to the four jacks (numbered #1 to #4 above) provided on the original board., as well as of an on-board 500Ω, 10-turn potentiometer. The five banana jacks are used to connect the board to power supplies and other instruments, by means of the banana-plug patch cords available at each station. The permanent connecting wires and links shown in the figure should not be disconnected. A 20 mH inductor (L), connected between jack terminals # 3 & # 5 , is also not to be disconnected. Before you start, make sure that the above permanent connections are in place and that the 5 banana-jack terminal tops are firmly screwed down.

The board contains a matrix of spring-loaded insertion-points at which ‘hook-up’ wire or the wire-end of a component may be inserted to construct a test circuit. The insertion-points are arranged as vertical and horizontal busses, a ‘bus’ being a series of internally connected points. Typical ‘horizontal’ busses (A,B) and ‘vertical’ busses(C,D) are shown in the lower part of the board in Figure 2.2.

On each horizontal bus, there are 25 internally-connected insertion-points (in 5 groups) and on the four fields of ‘vertical’ busses, each set of 5 insertion- points is also internally connected.

These buss structures can be also seen by looking at the underside of the breadboard. The terminals #2 to #4 are already connected, using the permanent connecting wires, to the group of four horizontal busses as shown in Figure 2.3. Terminal #1 is the designated Ground terminal is connected, using the black wire and the permanent links, to the Ground Bus as shown in the figure. On-board circuit connections are made to the above busses using components and wires stocked in a plastic box provided at each station. Proper insertion of a hook-up wire or of a component into the breadboard requires that the wire-end be physically **straight** to approximately an eighth of an inch (~ 3 mm).

The onboard potentiometer is a $500\Omega \pm 5\%$ (max), 10-turn wire-wound type with a positional linearity of 0.25%. It is attached to a turns-counting dial which can be set from 0.0 to 10.0. The **end terminals** of the pot are connected to the 6th(Red wire,**X**) and 18th (Black wire,**Y**) vertical busses starting from the right, top side of the breadboard [see Figure 2.3]. The Blue or Green wire (**Z**) is connected to the wiper-terminal of the pot **through a 100 Ω resistor.** The total resistance of $\approx 500\Omega$ appears between **X** and **Y** and the resistance between **Z** and **Y** increases from 100 Ω to 600 Ω as the dial is turned clockwise(CW) from zero to 10. Although R_{ZY} can be obtained approximately from the dial reading, the **exact** value should always be obtained by ohm-meter measurement.[This is only because R_{XY} may not be exactly 500 Ω . On the other hand, if R_{XY} is initially measured, then R_{ZY} can be obtained from the dial reading. For example, if $R_{XY} = 489.9\Omega$ and the dial is set at 5, then R_{ZY} will be $\approx (0.5)(489.9)+100$ or $\approx 345\Omega$].When connected as a potentiometer, **Y** should be used as the grounded terminal.

A photograph of the breadboard, with a resistance network constructed on it and with typical banana plug patchcords connected is shown in Figure 2.4. The banana jacks #3 & #5, across which the inductor L is connected, should be short-circuited as shown in the figure (the short orange-coloured patch cord) whenever L is not being used, in order to avoid sending DC currents through the inductor, which could result in its damage.

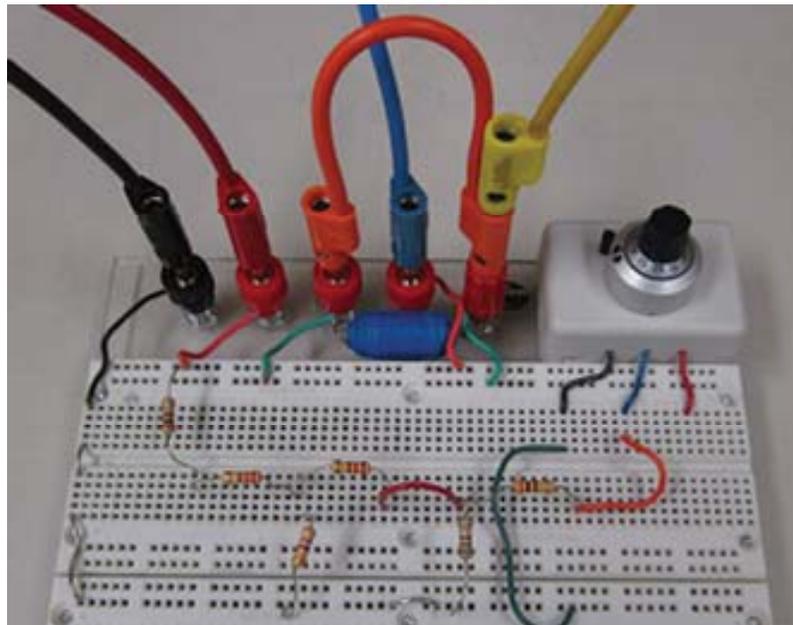


Figure 2.4

Procedure & Results

Figure 2.5 shows a collection of sample circuits to be built by the student on the breadboard using resistors having values ranging from 200 Ω to 510 Ω available in a plastic box at each station. Circuit A is to be used for the verification of the Superposition theorem only. Circuits B to E are sample circuits used for the verification of the TEC/NEC as well as for MPT. each containing five resistors [For circuits B to E, if resistance values other than those shown are used, make sure to ensure, by a preliminary calculation, that an R_T value will result such that it corresponds to a mid-point dial setting (ie $100\Omega + (1/2)(500)$ or $\approx 350\Omega$) on the on-board variable resistor (between Z & Y). This precaution is required for convenience in setting values above and below the R_T value for the MPT investigation.]

Resistive Circuit Collection for Expt # 2

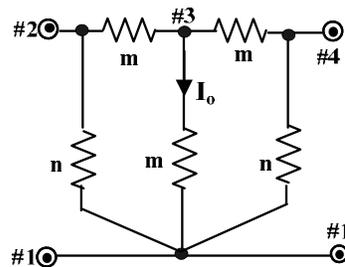
BASIC RESISTANCE COMBINATION RULES

For resistors connected in series $R_{eq} = R_a + R_b + R_c + \dots$ etc

For resistors connected in parallel $1/R_{eq} = (1/R_a) + (1/R_b) + (1/R_c) + \dots$ etc

For TWO resistors connected in parallel, $R_{eq} = R_a R_b / (R_a + R_b)$

Circuit A [to be used for the investigation of Superposition]

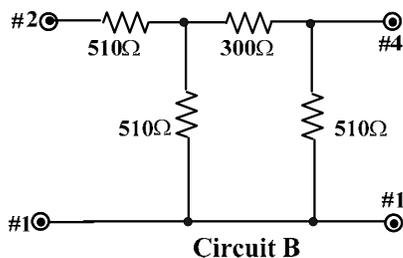


Note : The numbered terminals correspond to identically numbered terminals on the breadboard

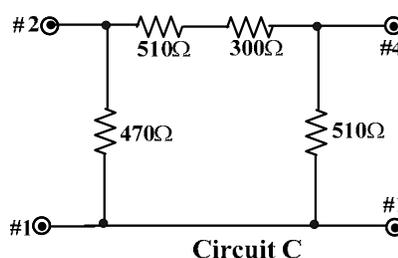
In Circuit A :
For resistances labelled 'm', use 200, 220, 300 or 330 Ω values
For resistances labelled 'n', use 470 or 510 Ω values

Sample Circuits B to E [to be used for the investigation of the TEC & MPT]

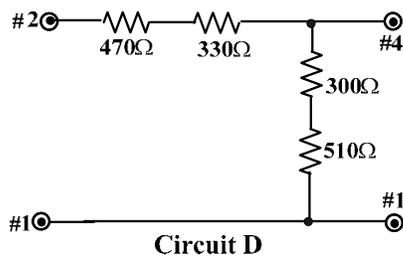
Note : Other circuits and available values may be used, but make sure that R_T will be in the approx 250 Ω to 400 Ω range



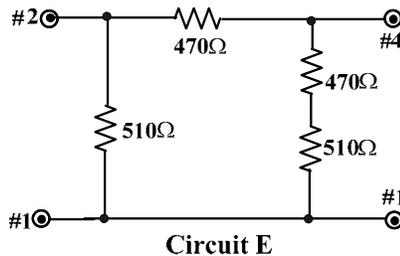
Circuit B



Circuit C



Circuit D



Circuit E

Figure 2.5

The superposition theorem will be verified only for the current I_o in the circuit "A" given in Figure 2.5. The TEC/NEC/MPT theorems will be verified using any one of the circuits "B"-to-"E" in Figure 2.5 to be chosen by the student. The terminals # 4 and #1 will refer to 'a' and 'b' (of Figure 2.2), respectively, in the TEC/NEC/MPT verification.

SUPERPOSITION THEOREM:

1. With the PSP still not turned on, connect the breadboard terminals #1, #2 and #4, respectively, to the ground, V_s and I_s supply outputs on the PSP as shown in Figure 2.6. On the PSP, set both output control knobs at 4-to-8 turns from the extreme CCW (minimum) position. This will result in a V_s output between 4 to 9 Volts and an I_s output in the range of 20 mA when the PSP is turned on. [The values of V_s and I_s obtained are not necessary for the Superposition theorem investigation, but may be calculated from node voltage measurements at points 2, 4 & S].

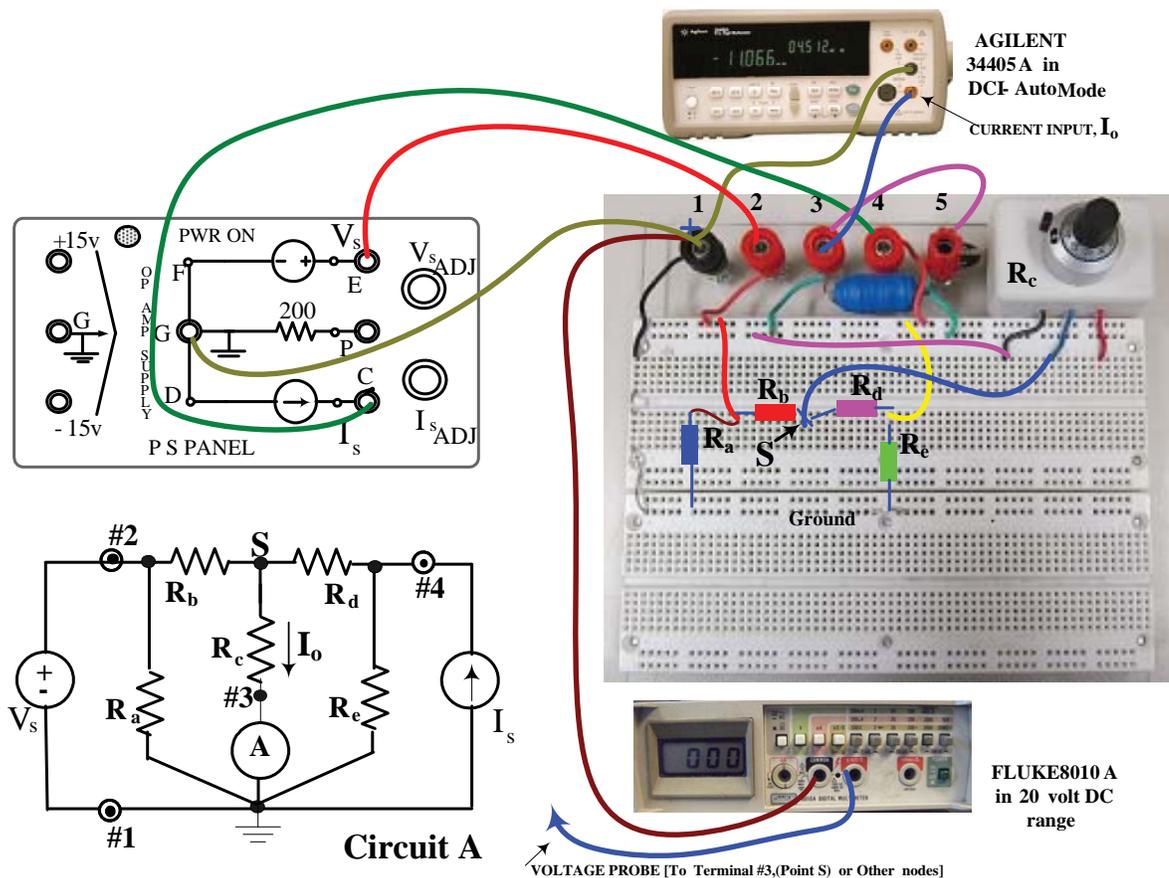


Figure 2.6: Setup for the verification of the Superposition Theorem

[The Fluke 8010A voltmeter shown may be also used to verify ‘superposition’ for any given node voltage and/or for general troubleshooting checks]

2. Before constructing the circuit, **measure and record the total resistance of the potentiometer** on your breadboard (between the Red and Black wire terminals (R_{XY})). Any other dial reading k ($k < 1$) can now be converted into a resistance value (between the Black & Blue wire terminals of $R_{ZY} = [100 + kR_{XY}]$, as explained on Page 27.. Then set R

($=R_{ZY}$) to a specific value between 200Ω and 300Ω . This value will be used for R_c in the test circuit of Figure 2.6.

- 3 . Construct the circuit “A” on the breadboard using the suggested available resistance values. for the other fixed resistors. A suggested layout is shown in Figure 2.6. The layout shown in Figure 2.6 need not be copied exactly by the student, but the variable resistor should be used for R_c .

Record the various R values used for your reference. When building the circuit, terminals # 1, # 2 and # 4 shown on the circuit must be arranged to correspond with the identically numbered buses (terminals) on the breadboard shown in Figure 2.6 for ease of reference. As shown, the point S is connected through R_c to terminal #3. The ammeter A (DMM) is connected between terminal #3 and Ground. Terminal #5 should be left unconnected, except for the inductor-shorting link..

4. Double check all your circuit connections Then turn on the master power switch to energize the PSP. Record the current I_o read by the ammeter under the three conditions (a),(b) and (c) listed below and enter the values in the data tables. With the ammeter leads connected with the proper reference directions shown, the indicated value will correspond to the designated current I_o .

Perform the following : **Record the current I_o**

- (a) With both V_s and I_s sources connected, turned on and adjusted as in Step 1 .
- (b) **With V_s acting alone.** This is achieved by disconnecting the lead between terminal # 4 of the breadboard and the current source terminal on the PSP.
- (c) **With I_s acting alone.** This is achieved by disconnecting the lead between terminal#2 of the breadboard and the V_s terminal on the PSP **and then** short-circuiting terminals #1 and #2 **on the breadboard.** [**Caution :Never short circuit the breadboard terminal #2 to ground when supply V_s is still connected !**]

Finally, turn the PSP off.

- 5 . **Results:** Verify the superposition theorem by showing that the current measured in Step 4(a) is the sum of the currents measured in Steps 4(b) and 4(c). **Simulate** the circuit “A” using Spice, with the elemental values that you used, both sources acting, and with I_o as the output.

THEVENIN, NORTON & MPT THEOREMS:

- 6 Choose a circuit from the set of “B” to “E” in Figure 2.5 and build it on the breadboard breadboard, again retaining the circuit-to-breadboard terminal number (terminals 1,2 & 4) correspondence as before. Once you have selected and built the circuit, draw the circuit in

the space provided in the data table section, making sure to record the resistor values used. On the breadboard, first set the potentiometer dial to its maximum ($R \approx 600\Omega$) and then connect the breadboard to the PSP (still **not** turned on) as shown in Figure 2.7.

- 7 The following list gives the theoretical value of the V_T/V_s ratio (α) as well as R_T value for the circuits of B to E in Figure 2.5. The value of V_T is then equal to αV_s where V_s is the source voltage set on the PSP. theoretical value of R_T is the resistance R_{41} when a short-circuit is assumed to be placed across terminals #2 & #1 instead of the voltage source V_s .

Circuit B : $\alpha \approx 0.24$ and $R_T \approx 266 \Omega$; Circuit C : $\alpha \approx 0.39$ and $R_T \approx 313 \Omega$

Circuit D : $\alpha \approx 0.5$ and $R_T \approx 403 \Omega$; Circuit E : $\alpha \approx 0.68$ and $R_T \approx 318 \Omega$

These are only provided as a guide; the actual values will depend upon the exact R values used .

Calculate V_T and R_T for your circuit.

- 8 Double check your circuit connections on the breadboard and then turn on the PSP. Temporarily disconnect the breadboard link between terminal #4 and the ‘top’ end of the variable resistor load [ie remove the **blue** wire link shown on the breadboard in Figure 2.7] . Adjust V_s on the PSP to obtain a node voltage V_4 in the range of 2 to 7 volts and record it in the data table. This corresponds to the ‘open-circuit’ voltage $V_{ab}(oc) = V_{41}(oc) = V_T$. Then reconnect the aforementioned **blue** wire link.

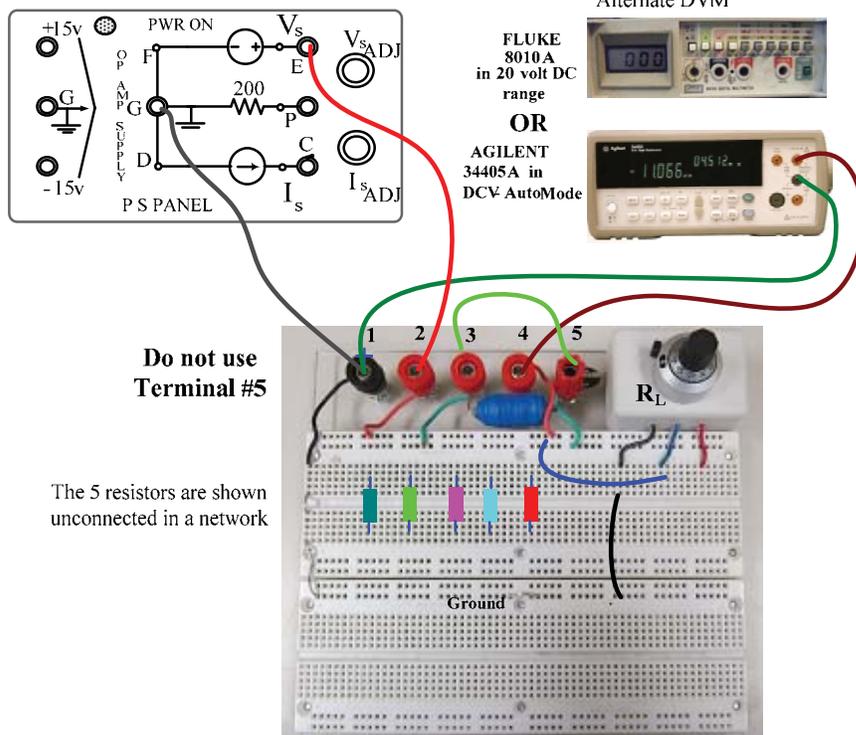


Figure 2.7: Setup for TEC/NEC/MPT verification

9. Temporarily disconnect the V-input lead at the DMM, configure the DMM as a DC ammeter (DCI, Auto) and then insert the lead into the I-input terminal .The DMM will then measure the ‘short-circuit’ current $I_{41}(sc)$. Record the value of $I_{41}(sc) = I_N$.**Then reconnect** the DMM as a voltmeter , as in Figure 2.7

10. Decrease the load resistance ($= R_L$) by turning the potentiometer dial CCW until the voltage V_{ab} ($= V_{41} = V_L$) falls to 50% of $V_{ab(oc)}$. [The pot resistance, calculated from the dial reading, should show close correspondence with the value calculated for R_T in Step 6]. Record this value as $R_{L3} = R_{mpt}$ in the middle row of Table A. Then vary the load resistance (by using the dial) above and below this value to two different positions such that distinct load voltage changes occur. Record the values as R_{L1}, R_{L2} (for $R_L < R_{mpt}$) and R_{L4}, R_{L5} (for $R_L > R_{mpt}$), as well as the associated load voltages V_{ab} ($= V_4 = V_L$) in the remaining rows of Table A.

11 **Results:**

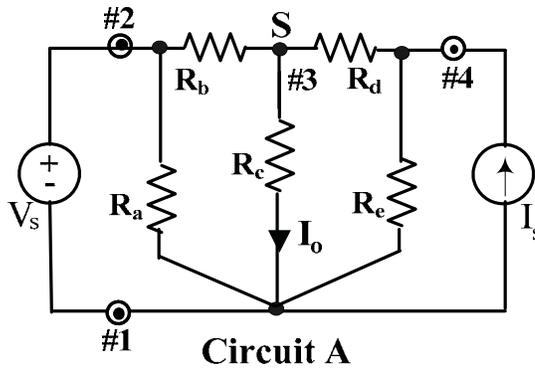
- (a) Determine the TEC and the NEC , with respect to terminals # 4 and #1 of your circuit, from the results of Steps 8 and 9 and compare the result with the theoretical values calculated in Step 7.
- (b) Compare the value of R_T obtained from Step 9 with the value calculated in Step 7.
- (c) From the data obtained in Step 10 and entered in the second column of Table A, calculate the load power $P_L = (V_L)^2/R_L$ and enter the values in the third column of the table. Plot P_L as a function of R_L to show that P_L reaches a maximum when $R_L = R_{mpt} = R_T$. Compare the value of $P_L(\max)$ from the plot with the theoretical value calculated from $(V_T)^2/4R_T$ where R_T and V_T are as obtained in Step 6 (with V_T found from the actual value of V_s used).

=====

DATA TABLES

[To be cut out & **pasted** into the Report booklet]

SUPERPOSITION



Steps 1, 2 & 3:

Resistor values used: $R_a = \dots\dots\dots$ Ohms, $R_b = \dots\dots\dots$ Ohms , $R_c = \dots\dots\dots$ Ohms

$R_d = \dots\dots\dots$, Ohms, $R_e = \dots\dots\dots$ Ohms

Source values used: $V_s = \dots\dots\dots$ volts , $I_s = \dots\dots\dots$,mA

Step 4:

(a) With both V_s and I_s sources connected, $I_o = \dots\dots\dots$, mA

(b) With V_s acting alone, $I_{o1} = \dots\dots\dots$, mA

(c) With I_s acting alone, $I_{o2} = \dots\dots\dots$, mA

TEC/NEC/MPT

Step 6 :

Draw the circuit selected in the space below and indicate the R values used.

TA Signature :

BLANK

Step 7: Calculate the theoretical values for your circuit (chosen from circuits B to E in Figure 2.5):

Calculate the value of V_T/V_s & of $R_T = R_{41}$ with terminals #1 & #2 short-circuited
 [after disconnecting V_s !]

Theoretical $V_T = \dots\dots\dots(V_s)$, volts Theoretical $R_T = \dots\dots\dots$ Ohms

Steps 8 to 9: (**Experimental** values)

Actual value of supply voltage $V_s = \dots\dots\dots$, volts

Open-circuit voltage $V_{ab}(oc) = V_{41}(oc) = V_T = \dots\dots\dots$, volts

Short-circuit current $I_{ab}(sc) = I_{41}(sc) = I_N = \dots\dots\dots$, mA

(Step 10) : $R_{L3} (= R_{mpt}) = \dots\dots\dots$ Ohms

TABLE A: Data for MPT verification
 [$R_{L1} < R_{L2} < R_{L3} < R_{L4} < R_{L5}$]

Load Resistance Value $R_L \Omega$ [Calculated from Pot Dial Setting]	Measured Load Voltage $V_{ab} = V_4 = V_L$ volts	P_L (expt) [from measured V_L] mW
$R_{L1} = \dots\dots\dots$		
$R_{L2} = \dots\dots\dots$		
R_{L3} (from Step10) = $\dots\dots\dots$		
$R_{L4} = \dots\dots\dots$		
$R_{L5} = \dots\dots\dots$		

TA Signature :

BLANK

EXPERIMENT 3

OPERATIONAL-AMPLIFIER CIRCUITS

OBJECTIVES : Investigation of Operational Amplifier Circuits with DC and AC Signals.

PRE-REQUISITES: 1. Familiarity with operational amplifier ('opamp') basics and with the the experimental equipment descriptions given in Appendix-B.
 2. Reading & making an attempt to understand the tutorial material given below, before coming to the lab session.

PART (A): TUTORIAL

OPERATIONAL AMPLIFIER CIRCUITS : The general objective of opamp circuit analysis is either to obtain the output voltage V_o for a given set of input voltages, or to find the voltage gain $A_v = V_o/V_{in}$. As mentioned in Appendix-B, the desired results are obtained by applying nodal analysis at the input terminal(s), together with the constraints given by the VOC & VSC principles, ie $I_+ = I_- = 0$ and $V_+ = V_-$ respectively. It is important to remember that when the ideal opamp is used, a nodal (ie KCL) equation can be written at all terminals, except at the output terminal, as explained in Appendix-B. For example, consider the two ideal-opamp circuits shown in Figure 3.1. Suppose that for the circuit of Figure 3.1(a), it is required to find V_o as a function of the two input voltages V_a and V_b , and for the circuit of Figure 3.1(b) it is required to find the voltage gain A_v . We shall now use nodal analysis, as indicated above, to obtain the desired results.

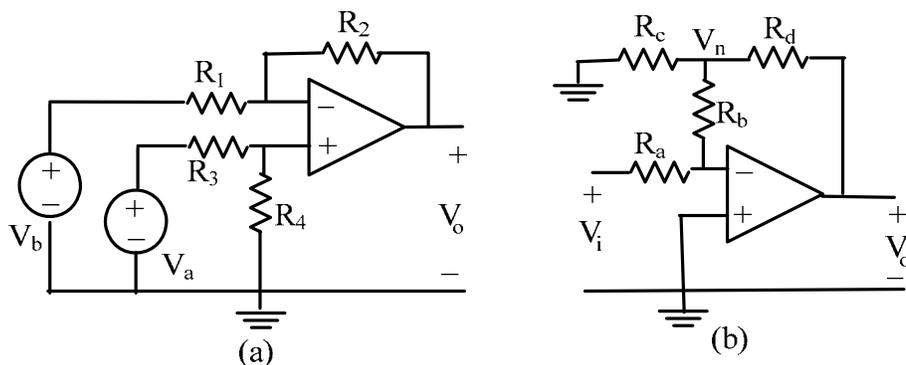


Figure 3.1

For the circuit of Figure 3.1(a), applying nodal analysis to the opamp input terminals, together with the VOC & VSC principles $I_+ = I_- = 0$ and $V = V_+ = V_-$:

$$\frac{V - V_b}{R_1} + \frac{V - V_o}{R_2} = 0 \quad \text{and} \quad \frac{V - V_a}{R_3} + \frac{V}{R_4} = 0$$

Elimination of V between the two equations, and re-arrangement, gives the desired result:

$$V_o = \frac{R_4(R_1 + R_2)}{R_1(R_3 + R_4)} V_a - \frac{R_2}{R_1} V_b$$

[If all resistors are chosen to be of the same value, the above relation is $V_o = V_a - V_b$ which indicates that the circuit can perform the operation of subtraction of V_b from V_a .]

For the circuit of Figure 3.1(b), writing the nodal equation at the inverting input terminal and at the node 'n' (with $V = V_+ = V_- = 0$) yields, respectively :

$$\frac{0 - V_i}{R_a} + \frac{0 - V_n}{R_b} = 0 \quad \text{and} \quad \frac{V_n - 0}{R_b} + \frac{V_n - 0}{R_c} + \frac{V_n - V_o}{R_d} = 0$$

Elimination of V between the above two equations, and re-arranging the result to obtain the voltage-gain V_o/V_i we have :

$$\frac{V_o}{V_i} = -\frac{R_b R_d}{R_a} \left[\frac{1}{R_b} + \frac{1}{R_c} + \frac{1}{R_d} \right]$$

[If all resistors are chosen to be of the same value, the above relation results in a voltage gain of -3 volt/volt.]

The nodal analysis procedure used in the two examples above can be applied similarly to any opamp circuit (DC or AC) to obtain the output voltage or to find the voltage gain.

AC opamp circuits : The presence of a linear operating range (see Appendix-B) in opamp circuits is particularly evident when an AC (sinusoidal) input is used : As the input signal amplitude is increased, 'limiting' of the output voltage waveform can be clearly observed when displayed on an oscilloscope. An important AC application is in the area of 'electrical filters', which are circuits designed to obtain frequency-dependent gain characteristics which allow the transmission or rejection of certain frequency bands.. Opamp filter circuits generally incorporate RC networks as seen in the circuits shown in Figure 3.2 (a) & (b). The circuits shown are those of basic first-order 'low-pass' (LPF) and 'high-pass'(HPF) filters, which are used to 'filter out' high-frequencies and low frequencies, respectively.

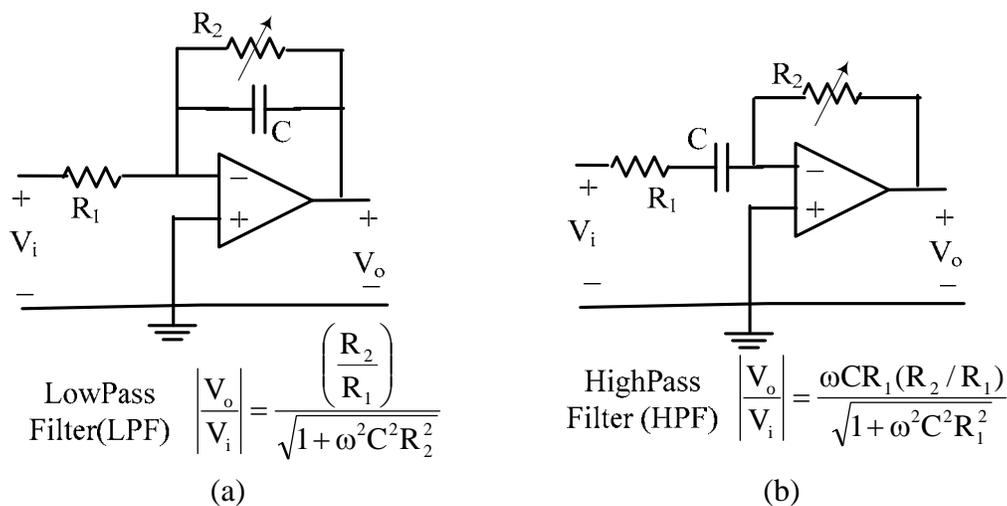


Figure 3.2

The expressions for the voltage gain A_v for both circuits are also given in the above figure. For the LPF, $A_v=(R_2/R_1)$ at zero frequency (DC) but falls off as the frequency increases. For $\omega CR_2 = 1$ (ie when $f = f_c = 1/2\pi CR_2$), the gain falls to $(1/\sqrt{2}) (R_2/R_1)$ or approximately 0.707 of the DC gain. The frequency f_c is called the ‘cutoff’ frequency of the LPF. Similarly, for the HPF, the DC gain is zero, but increases as the frequency increases and approaches (R_2/R_1) for frequencies for which $\omega \gg 1/CR_1$. For $\omega = 1/CR_1$ (ie when $f = f_c = 1/2\pi CR_1$), the gain increases to $(1/\sqrt{2}) (R_2/R_1)$ or approximately 0.707 of the ‘high frequency’ gain value and f_c is called the ‘cutoff’ frequency of the HPF.[It can be shown that for $f \gg f_c$, the LPF can perform the mathematical operation of integration with respect to time, and for $f \ll f_c$, the HPF can perform differentiation]

The AC response of the ‘low-pass’ circuit will be investigated in this experiment.

PART (B) : EXPERIMENTAL:

Procedure & Results

In all the experimental circuits, the power to the opamp is provided by the opamp supply terminals on the PSP. In every case, the PSP should be turned on only after double-checking your connections!

DC OPERATIONAL AMPLIFIER CIRCUITS :

1. With the PSP still not turned on, make the general connections shown in Figure 3.3 between the ‘3140’ opamp box, PSP and the DMMs. Set the Fluke 8010A to the 20 volt DC range. and the Agilent 34405A to DCV Auto. Then turn on the master supply to energize the PSP so that the $\pm 15v$ supplies are applied to the opamp. [Use the + lead from the Fluke 8010A as a probe to check the input and supply voltages]
2. First, check the opamp IC : With no other inputs applied, temporarily short-circuit one of the inverting input terminals to ground, for example such as the dark blue connection to 100k . The output voltage should be zero volts; if it is not zero, the IC should be replaced.

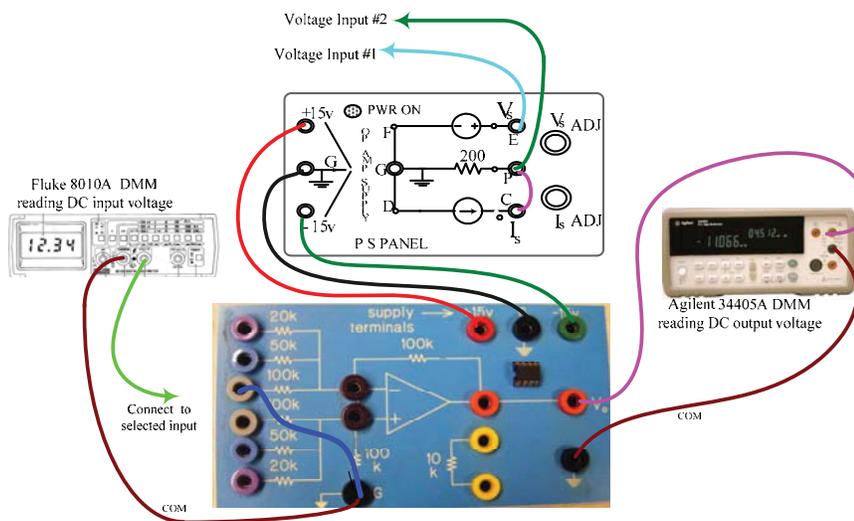
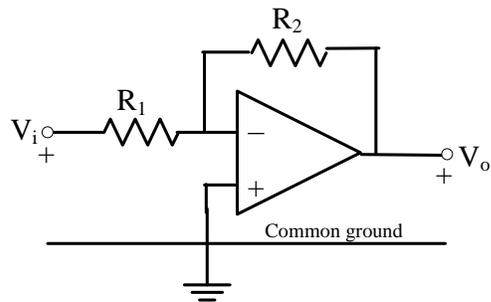
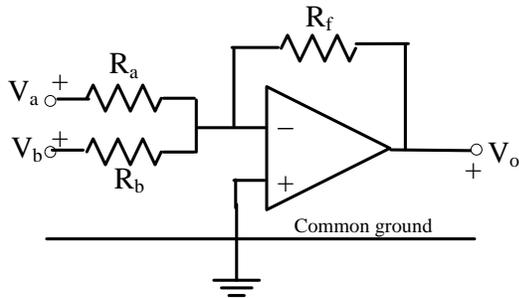


Figure 3.3: DC Opamp Circuit Test Setup

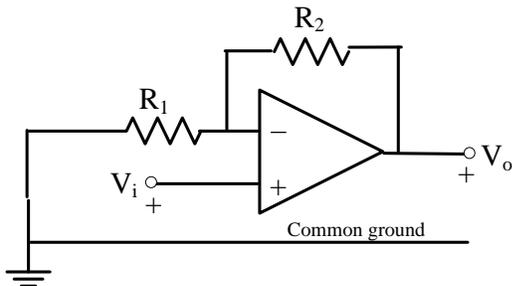
3. Successively build the circuits shown in Figure 3.4, using the short banana-plug patch cords to provide the required interconnections. Make sure to record the effective values used for the resistors R_1 to R_4 in each case. Various feedback resistance values and input resistance values may be effectively obtained by using the available onboard resistors in parallel with one another. An isolated 10 k Ω resistor is also available. For each circuit, apply an appropriate input voltage (or voltages) from the variable supplies on the PSP [Remember to SC the terminals P & C on the PSP to convert the current source into a second voltage source]. Use the data tables to record the R values and voltages. Make sure to use gain and input values such that the output voltage remains within the linear range of approximately ± 12 volts. For each of the circuits given, do the following :
- Inverter [Figure 3.4(a)] : Short circuit the (+) input terminal to ground. Construct inverters with at least two different gain values. Measure V_o for different values of V_i to verify the $V_o/V_i = - (R_2/R_1)$ relationship .
 - Summing Inverter [Figure 3.4(b)]: Short circuit the (+) input terminal to ground. Using the pre-wired 100 k Ω feedback resistor, construct a summing inverter circuit with two different weighting coefficients (R_f/R_x). Measure V_o for two different sets of input values to verify the relation $V_o = - [(R_f/R_a)V_a + [(R_f/R_b)V_b]$
 - Non-inverting Amplifier [Figure 3.4(c)] The inverter circuit of Figure 3.4(a) can be easily converted to a non-inverter by interchanging the input connections as follows: Remove the short circuit between the (+) input terminal & ground Then, ground the input end of resistor R_1 (previously connected to V_i) and apply the input V_i directly to the non-inverting (+) input. Measure V_o for two different input values to verify the relation $V_o = [1 + (R_2/R_1)]V_i$. Obtain results for a gain values of **2** and for one other value of your choice..
 - Voltage Follower [Figure 3.4(d)]: The voltage follower can be treated as a special case of the non- inverter, with $R_2 = 0$ and $R_1 \rightarrow \infty$. Construct a voltage-follower and measure V_o for different values of V_i to verify the $V_o = V_i$ relation. {Voltage followers are used to provide ‘isolation’ between two adjacent networks in a chain (or ‘cascade’). A typical case is the ‘loading’ of a potential-divider (‘potentiometer’) by a load resistor R. The load R will be in parallel with the lower section of the potentiometer thereby changing the voltage-divider ratio. If a voltage-follower is inserted between the load and the potentiometer, then no current will be drawn from the latter, thus preserving the voltage-divider ratio. The current drawn by R will now come from the opamp. }
 - Differential Amplifier [Figure 3.4(e)]: Connect the opamp as a differential amplifier with a gain of one(ie with all resistors $R = 100$ k Ω). Measure V_o for two different sets of input values to verify the relation $V_o = V_a - V_b$.
4. **Simulate** the summing inverter and differential amplifier circuits that you investigated experimentally [Use an ‘ideal’ opamp and the resistor & input voltage values that you selected in the experiment.]



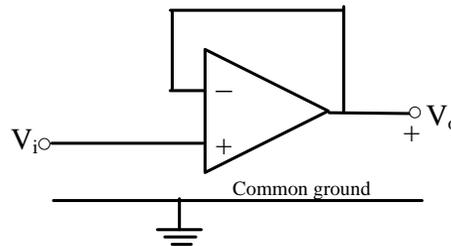
(a) Inverter $V_o = -[R_2/R_1]V_i$



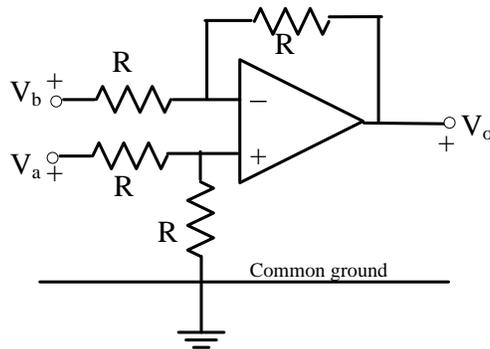
(b) Summing Inverter $V_o = -[(R_f/R_a)V_a + (R_f/R_b)V_b]$



(c) Non- Inverter $V_o = [1 + (R_2/R_1)] V_i$



(d) Voltage Follower $V_o = V_i$



(e) Differential Amplifier $V_o = V_a - V_b$

Figure 3.4 : Basic opamp circuits

AC OPERATIONAL AMPLIFIER CIRCUITS

5. With the PSP still not turned on, make the general connections shown in Figure 3.5 between the '741' opamp box , PSP , FG and DSO. Set the FG to the sinewave 10kHz range and the DSO to DC coupling, both channels to 1 volt/div sensitivity, and 2.5 msec/div timebase. Also set the DSO to display the Ch1 Frequency, and the RMS amplitudes of both channels. {First check the opamp: With the connections shown in Figure 3.5 in place , temporarily disconnect the input from the FG at terminal N and connect N to ground. Then turn on the PSP and observe the output trace on the DSO. If the output voltage is zero, the opamp IC is functioning properly.}

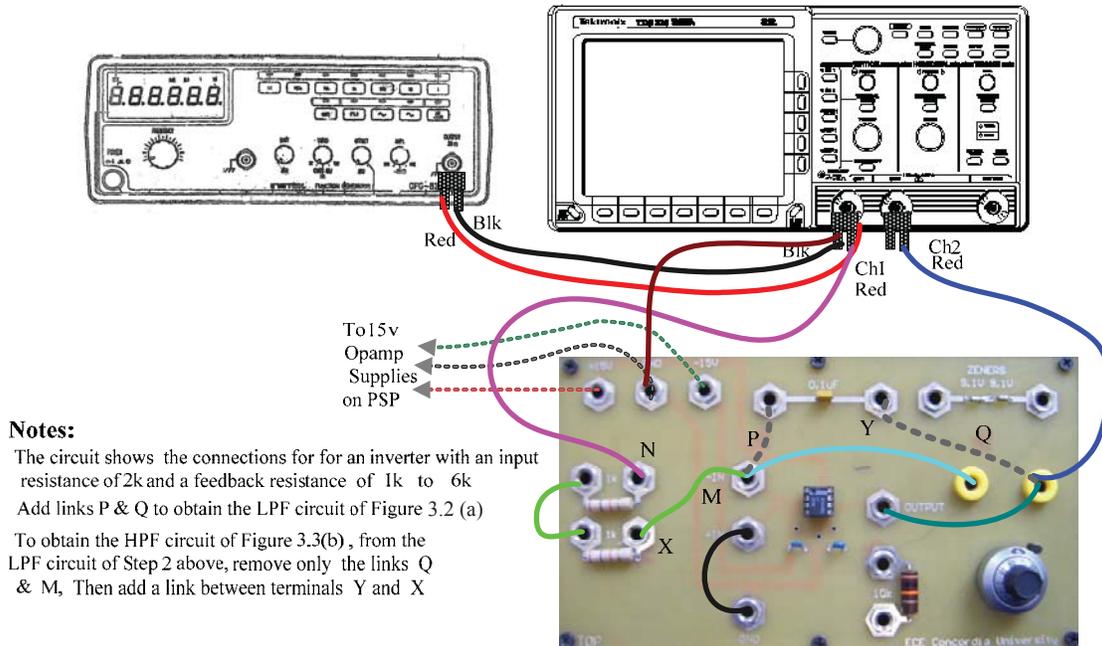


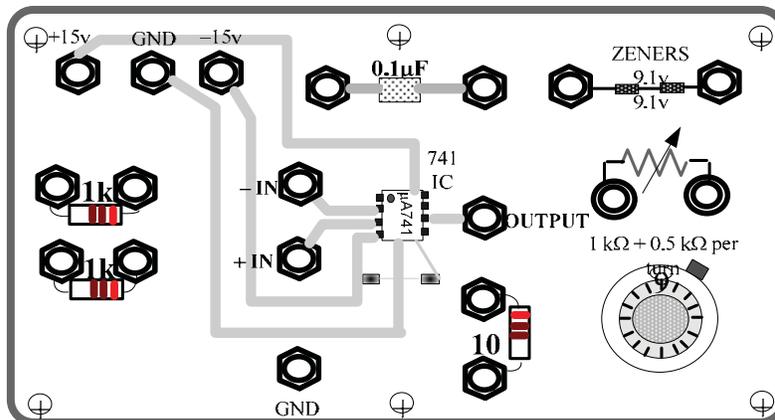
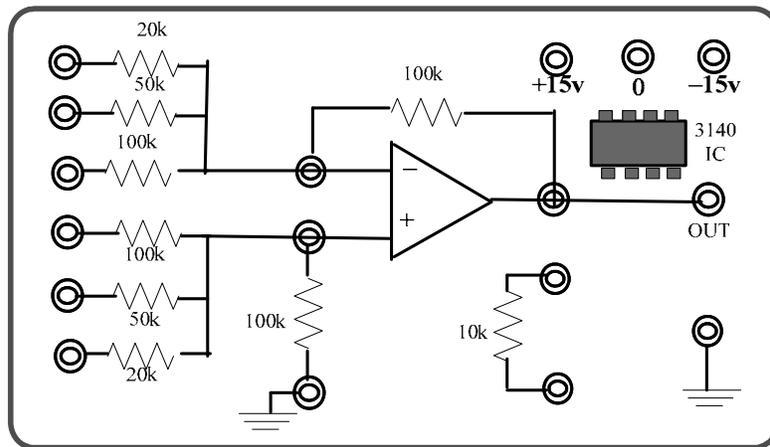
Figure 3.5: AC Opamp Circuit Test Setup

6. The basic connections (solid line connections) shown in Figure 3.5 are for an inverter with variable gain: The effective input resistance is 2 k Ω (ie 1k Ω +1k Ω) and the feedback resistance is provided by the onboard 6 k Ω variable resistance (1k Ω +5k Ω of the 10-turn potentiometer) resulting in a gain which is variable from -0.5 to -3. Ch1 and Ch2 as wired in Figure 3.5 correspond to the input and output respectively. Adjust the FG controls to obtain a 2 volt RMS, 1 kHz sinewave as measured on the DSO. [Note: On the DSO, the frequency measurement will only appear when the timebase is adjusted so that 3 or 4 cycles of the waveform are displayed. This can be ensured if the 'setup' button on the DSO is pressed after every change of frequency]. Adjust the gain magnitude to exactly unity using the 10-turn knob. Obtain a printout showing the inverting action and voltage gain of the circuit.
7. Observation of linear range (output voltage limiting): Temporarily increase the gain to maximum by turning the 10-turn control knob. Then turn the Ch2 sensitivity to 5 volt/div, increase the input magnitude to maximum and observe the effect of 'saturation' (ie V_o limited by power supply voltages) as the output voltage magnitude approaches $\sim \pm 12$ volts. [The output voltage can also be limited to selected values ($< V_s$) by using a non-linear element known as a 'Zener diode'. The '741' box has a pair of Zener diodes which can limit the output voltage to 9.7 volts. This 'limiting' can be observed on the DSO if the Zener diode terminals (at the top right hand corner of the box) are temporarily connected to the feedback resistor terminals when the output is already at $\sim \pm 12$ volts. It will be seen that the limiting now occurs at ± 9.7 volts]

8. Build the LPF circuit (with unity DC gain) shown in Figure 3.2 using the basic setup of Figure 3.5. The LPF circuit will result if the $0.1\mu\text{F}$ onboard capacitor is connected across the feedback resistor by simply adding the two links P and Q shown as dotted lines in Figure 3.5. Keeping the input to the LPF constant at 2 volt RMS, vary the frequency and record the RMS output voltage at 100, 200, 500, 1000, 5000 and 10,000 Hz. Use the data table to record the frequencies and voltages. [Use the 'RMS' & 'Frequency' measurement functions on the DSO].
 { To observe the 'integrator' effect of the circuit (for $f \gg f_c$), temporarily switch the input waveform at 10 kHz, to a square-wave. The output will be a triangular-wave, which is the result of integrating the square-wave. Obtain a printout for your report. }
9. Compare the experimental results with theoretical values for the voltage gain A_v , at two different frequencies of your choice, one below the 'cutoff' value $f_c = 1/2\pi CR_2$ and the other above it. Calculate the % error in each case, relative to the theoretical value and comment .

=====

OUTLINE DRAWINGS OF THE TWO OPAMP BOXES



BLANK

DATA TABLES

[To be cut out & **pasted** into the Report booklet]

DC CIRCUITS

INVERTER DATA

Inverter #1			Inverter #2		
$R_1 = \dots\dots\dots k\Omega$ $R_2 = 100 k\Omega$ Gain = -			$R_1 = \dots\dots\dots k\Omega$ $R_2 = \dots\dots\dots k\Omega$ Gain = -		
V_i volts	V_o volts	Gain	V_i volts	V_o volts	Gain

SUMMING-INVERTER DATA

#1			#2		
$R_a = \dots\dots\dots k\Omega$ $R_b = \dots\dots\dots k\Omega$			$R_a = \dots\dots\dots k\Omega$ $R_b = \dots\dots\dots k\Omega$		
V_a volts	V_b volts	V_o volts	V_a volts	V_b volts	V_o volts

NON-INVERTER DATA

Non-Inverter #1			Non-Inverter #2		
$R_1 = 100 k\Omega$ $R_2 = 100 k\Omega$ Gain = + 2			$R_1 = \dots\dots\dots \Omega$ $R_2 = \dots\dots\dots \Omega$ Gain = +		
V_i volts	V_o volts	Gain	V_i volts	V_o volts	Gain

VOLTAGE-FOLLOWER DATA

Input V_i volts			
Output V_o volts			

DIFFERENTIAL AMPLIFIER DATA

All $100k\Omega$ Resistors, Gain = 1

V_a volts	V_b volts	V_o volts

TA Signature :

BLANK

AC CIRCUITS

[Note: For the LPF, 'cutoff frequency' f_c is the frequency at which $A_v = \frac{1}{\sqrt{2}} = 0.707$, ie when $\omega CR = 1$. With $R_1 = R_2 = 2 \text{ k}\Omega$ and $C = 0.1 \text{ }\mu\text{F}$, the 'cutoff

frequency' $f_c = \frac{1}{2\pi CR} \approx 796 \text{ Hz}$. [Note that at 100 Hz, $A_v = 0.992 \approx 1$]

LOWPASS FILTER

Input $V_i = 2 \text{ volt RMS}$

f Hz	ω Radians/sec	Output V_o Volts RMS	$A_v = V_o/V_i$
100	628		
200	1257		
500	3142		
1000	6283		
2000	12566		
5000	31415		
10,000	62830		

TA Signature:

BLANK

EXPERIMENT 4

DYNAMIC RESPONSE OF RC, RL and RLC CIRCUITS

- OBJECTIVES** : (A) Investigation of the transient response of RC, RL and RLC circuits
(B) Determination of the frequency response RC, RL and RLC circuits
(C) Resonance & Impedance topics

- PRE-REQUISITES**: 1. Complete familiarity with the breadboard system used in Expt#2.
2. Reading & making an attempt to understand the tutorial materials given below, as well as in Appendix C before coming to the lab session.

PART (A): TUTORIAL

TRANSIENT RESPONSE: When a circuit containing only resistance elements is subjected to a sudden change, either in circuit structure or in excitation magnitude, the response of the various voltages (and currents) is instantaneous. In contrast, a circuit containing storage-elements (L and/or C) in addition to R elements, such a change results in a ‘transient’ response during which the various voltages (and currents) change from old to new equilibrium values. Transient response is an important consideration in circuit design, since it indicates the speed at which the output can respond to a sudden change in input or in other circuit conditions. In AC circuits, it is desired that the transient period is as short as possible since only the steady state AC response is of interest. In DC-excited circuits containing storage-elements, however, the transient response is of considerable interest since it can be incorporated in design requirements such as a desired ‘time delay in the output. In general, for circuits containing R, L and C elements, the theoretical transient response is obtained as a solution to a linear differential equation (LDE) which results from the analysis of the circuit.

In the case of “first-order” circuits [ie RC or RL circuits, which contain only one equivalent storage element (either C or L)] the LDE is of the general form :

$$\tau \frac{dy(t)}{dt} + y(t) = f(t)$$

where $y(t)$ is the variable of interest, $f(t)$ is called the ‘mathematical input’ or ‘forcing function’ which is a constant for DC-excited circuits. The parameter τ is called the ‘time-constant’ and it is given by $\tau = RC$ or by $\tau = L/R$, where R is the Thevenin resistance of the TEC connected to C or to L. The transient response is commonly obtained as a ‘step response’, which is the circuit response to a ‘step input’ function defined by $f(t) = Fu(t)$, where $f(t) = 0$ for $t < 0$ and $f(t) = F$, a constant, for $t > 0$. For

$t > 0$, which is the range of interest, the general solution of the above LDE, with $f(t) = F$, is given by

$$y(t) = A e^{-(t/\tau)} + F \quad \text{where } A = \text{a constant}$$

The DC steady state (DCSS) solution F in the above equation is obtained when $t \rightarrow \infty$ or $y(\infty) = F$ and the constant A can be found by using a known value of $y(t)$, eg: $y(0+)$. Thus, at $t = 0+$, $A = y(0+) - F = y(0+) - y(\infty)$ and the above response equation can be also written as :

$$y(t) = [y(0+) - y(\infty)] e^{-(t/\tau)} + y(\infty)$$

The general response has two particular (or classic) cases, which depend upon the initial value $y(0+)$ (or ‘initial condition’) and the final (DCSS) value $y(\infty)$:

(a) a ‘rising exponential’ type of response

$$y(t) = y(\infty) [1 - e^{-(t/\tau)}] = F [1 - e^{-(t/\tau)}], \quad \text{which occurs when } y(0+) = 0,$$

and

(b) a ‘falling exponential’ type of response

$$y(t) = y(0+) e^{-(t/\tau)} \quad \text{which occurs when } y(\infty) = F = 0$$

These two classic responses (which are also called the ‘forced-response’ and the ‘natural response’ respectively) are shown in Figure 4.1. Examples of a ‘rising’ response are (i) the voltage across C in a series-RC circuit subjected to a positive step voltage input, and (ii) the voltage across R in a series RL-circuit subjected to a positive step voltage input. Examples of a ‘falling’ response are (i) the voltage across R in a series-RC circuit subjected to a positive step voltage input, (ii) the voltage across L in a series RL-circuit subjected to a positive step voltage input.

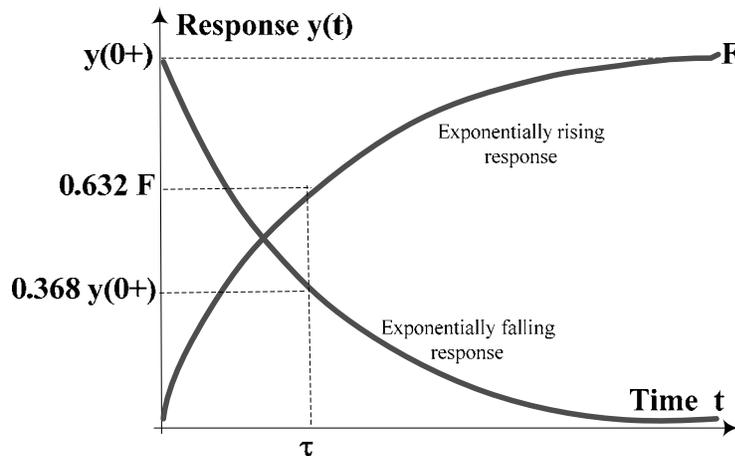


Figure 4.1: First-order transient responses

For either type of response, the DCSS value is the quantity on the right hand side of the LDE, which is either F or zero. The DCSS value may be also found from the circuit after replacing the capacitor C with an open-circuit or the inductor L with a short-circuit. Although the curves in Figure 4.1 are shown between specific limits, it should be noted that these responses can occur between any two limits which are determined by the ‘boundary’ values of $y(0+)$ and $y(\infty)$.

Time-constant determination: Substitution of $t = \tau$ in the above classic response equations leads to a graphical procedure of obtaining τ from the response plots: For the ‘rising’ response, $y(\tau) = F[1 - e^{-1}] = 0.632F$, or τ is the time taken for $y(t)$ to rise from zero to 63.2% of its final value F. For the ‘falling’ response, $y(\tau) = y(0+)e^{-1} = 0.368 y(0+)$ or τ is the time taken for $y(t)$ to fall to 36.8% of its initial value $y(0+)$. In the lab experiments, τ will be found from oscilloscope traces of the response using cursor measurements with the above reference points. The time constant τ is the major parameter of interest in first order circuits.

In the case of “second-order” circuits [ie circuits which contain R, L and C, or R and two storage elements of the same type, but which cannot be combined], the LDE is of the general form:

$$\frac{d^2 y(t)}{dt^2} + 2\alpha \frac{dy(t)}{dt} + \omega_n^2 y(t) = f(t)$$

where α and ω_n , which are called the ‘damping coefficient’ and ‘undamped natural frequency’ respectively, are important transient response parameters. In the above equation, α is alternatively expressed as $\zeta\omega_n$ where ζ (pronounced ‘zeta’) is called the ‘damping ratio’ $\zeta = \alpha / \omega_n$. Second-order responses can also be of the ‘rising’ or the ‘falling’ type.

The ‘step response’ of the second-order LDE with $f(t) = Fu(t)$ can be of three distinct forms depending upon the value of ζ :

- (i) ‘over-damped’ (OD), which occurs when $\alpha > \omega_n$, or $\zeta > 1$
- (ii) ‘critically-damped’ (CD), which occurs when $\alpha = \omega_n$, or $\zeta = 1$
- (iii) ‘under-damped’ (UD), which occurs when $\alpha < \omega_n$ or $\zeta < 1$

The three response types are shown in Figure 4.2 for a second-order circuit which has a zero initial condition and a positive DC steady state value. The OD and CD responses shown are similar to an ‘exponentially rising’ first-order response. The UD response, however, is characterized by an ‘overshoot’ ; it is a response which oscillates around the steady state value before settling down. The UD response, which is the fastest of the three response types, has important engineering applications.

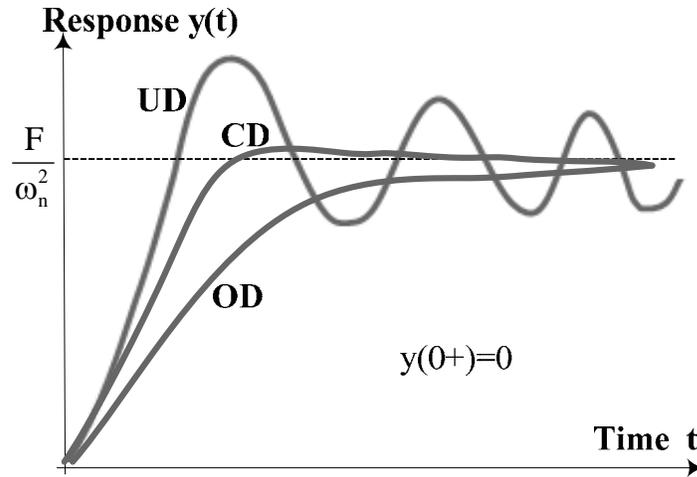


Figure 4.2: Second-order transient (step) responses

Transient response tests : The ‘step’ responses described and shown in Figures 4.1 and 4.2 above can be experimentally found by using a switch to suddenly apply a DC input and then observing the change in output on an oscilloscope. However, in such a ‘switched’ circuit, observation of the resulting response will require elaborate time-synchronization arrangements which can be complex to implement, especially when the duration of the transient response is short. A common way of avoiding such synchronization problems is to use a square-wave input, which may be considered as a cyclically-repetitive step input (positive-going and negative-going steps), as shown in Figure 4.3 for a rising exponential RC circuit response.

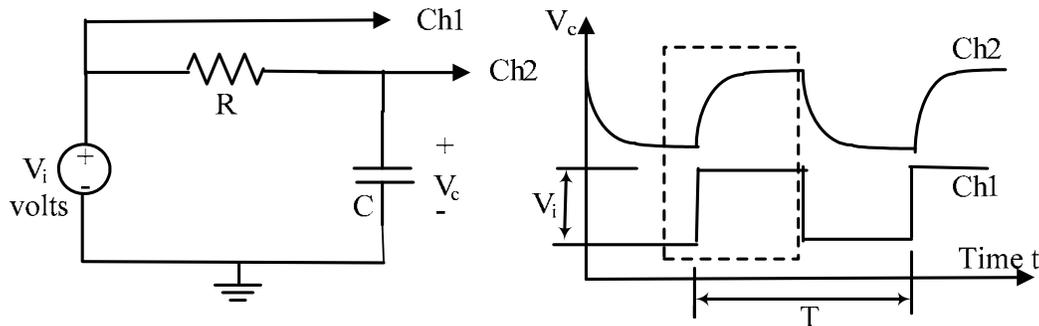
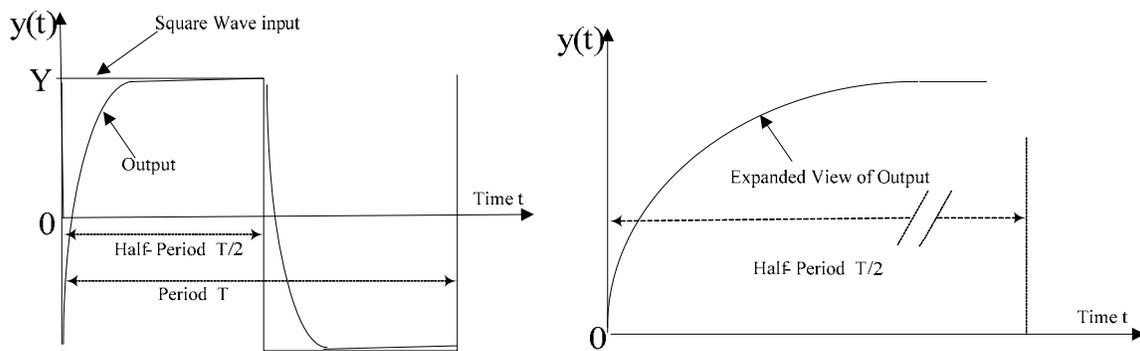


Figure 4.3

If the duration of the half-period of the square-wave is made large in comparison with the time over which the transient response occurs, the display will appear as the transient response to a step input, as can be seen from Figure 4.4.



One cycle of the square-wave response

Expanded view for $t < T/2$

Figure 4.4

If the oscilloscope time base ('time/div') is then adjusted to display only the half-cycle, the associated response will appear as a 'step response' similar to that seen in Figures 4.1 and 4.2. For example, we can make the half-period $T/2$ of the square-wave to be equal to 20τ , i.e. $T = 40\tau$ or $f = 1/40\tau$. Thus, if a $\tau = 0.5$ ms is to be observed, a square-wave frequency of $1/(0.02)$ Hz or 50 Hz will ensure that the response within the half-cycle will appear as a 'step response'. This technique will be used in the laboratory for observing both first-order (RC, RL) and second-order (RLC) step responses.

SINUSOIDAL STEADY STATE & IMPEDANCE :

In Appendix-C, it is shown that AC circuit analysis is performed using 'phasor circuits', in which the sinusoidal steady state voltages and currents are represented by their 'phasor' equivalents. **Phasors** [see Appendix C] are complex numbers which represent the various voltages and currents in AC circuits. Thus, phasors have magnitudes and relative phases with respect to a reference zero-phase voltage or current. **Impedance** Z is defined as the phasor ratio of voltage to current, with respect to any given terminal pair. The reciprocal of Z is called the Admittance, Y . All circuit elements such as R , L , and C are represented as impedances [$Z_R = R$, $Z_L = j\omega L = \omega L \angle 90^\circ$, and $Z_C = 1/j\omega C = (1/\omega C) \angle -90^\circ$] in a phasor circuit. Since Z is defined by a voltage to current ratio as in Ohm's law, all the results and techniques applied to DC resistive circuits such as KCL, KVL, series-parallel combinations, voltage and current division rules, mesh and nodal analysis etc. are applicable to impedances (phasor circuits), the only difference being that all quantities are now complex numbers.

FREQUENCY RESPONSE

Phasor circuits are frequency-dependent because the impedances in the circuit are frequency-dependent. The term 'frequency response' refers to the variation, with frequency, of the magnitude and phase of a specified voltage or current. A 'frequency-response test' usually consists of keeping the magnitude of the input voltage V_i of a circuit constant while changing the frequency and recording the resulting variation in output voltage magnitude V_o and its phase shift ϕ , with respect

to the input. Frequency response data is conventionally presented as a ‘magnitude ratio’ $M(\omega) = V_o(\omega)/V_i(\omega)$ and the ‘phase-shift’ $\phi(\omega) = \theta_o - \theta_i$. In the interests of covering a large response range, it is a convention to plot ω (or f) along a semi-logarithmic x-axis and a decibel (db) value for M along a linear y-axis. The decibel value of M is given by

$$M(\text{db}) = 20 \log_{10} M$$

Examples of M and ϕ curves for a first-order circuit are shown in Figure 4.5(a). Examples of logarithmic M plots for first and second order low-pass type circuits are shown in Figure 4.5(b)

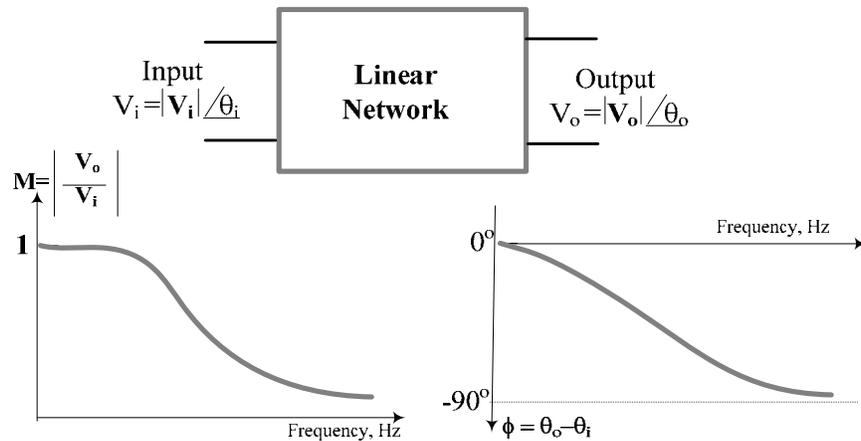


Figure 4.5(a): Frequency response curves of a First-order circuit

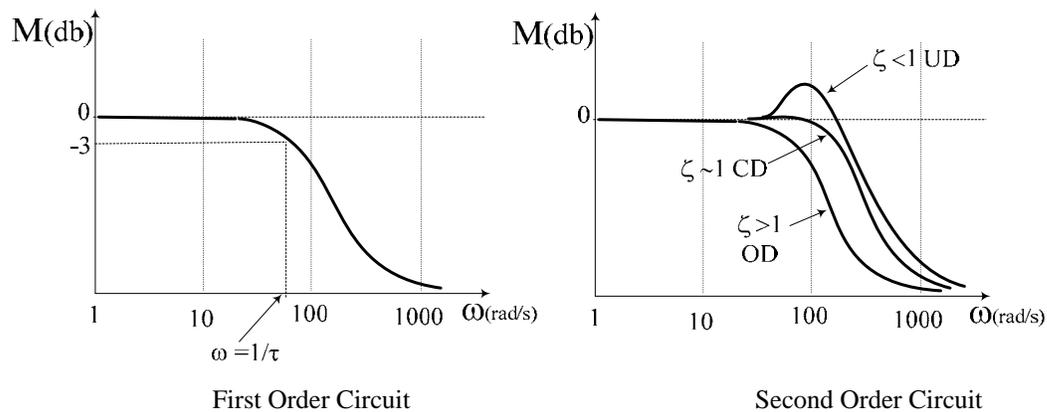


Figure 4.5(b) : M curves for First-order & Second-order circuits

The complex quantity $M(j\omega) \angle \phi(j\omega)$ is called the Transfer Function $T(j\omega)$, which is used as a mathematical model of the ‘system’ in advanced analysis] The frequency response curves for first-order circuits and a second-order circuit will be obtained in this experiment.

RESONANCE:

Resonance is said to occur whenever the reactive part of a branch impedance (or admittance) is reduced to zero by cancellation (see Appendix-C) of complementary reactances, caused either by change in frequency or by change in elemental values. For example, the impedance of a series-RLC combination, given by

$$Z = R + j\omega L - 1/j\omega C = R + j[\omega L - 1/\omega C]$$

will be ‘resonant’ at the frequency f_r for which $\text{Im}(Z) = 0$, ie when $[\omega L - 1/\omega C] = 0$. Thus $f_r = 1/(2\pi\sqrt{LC})$ is called the ‘resonant frequency’. Since such reactance cancellations will result in either maximization or minimization of impedances (or admittances), resonances are characterized by positive or negative ‘peaks’ in the magnitude response curve. The phenomenon of resonance in a series RLC circuit will be observed in this experiment.

IMPEDANCE MEASUREMENT: The measurement of impedance Z is frequently required in practice. Although $Z (= V_z/I_z)$ can be measured using various techniques, a direct method using measurements of the magnitude and phase of the associated voltage and current phasors is instructive. Figure 4.6 shows a circuit in which an unknown impedance Z is to be found. The circuit consists of a sinusoidal function generator connected to the series combination of the impedance Z with a known resistance R . The voltage across the series combination of Z and R is V_A and the voltage across R is V_B . The voltage across Z is then given by $V_z = V_A - V_B$.

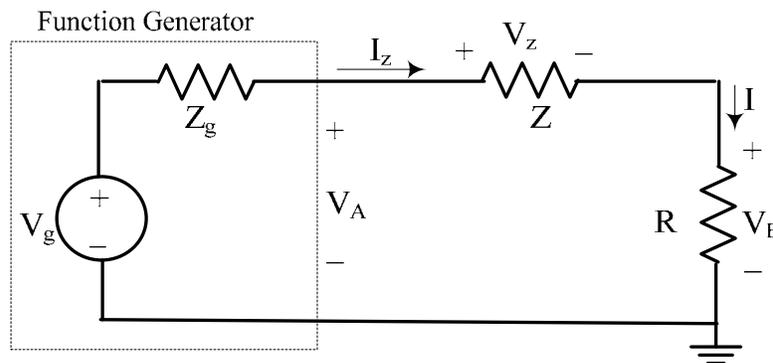


Figure. 4.6

Since R is a resistance, the current $I (= I_z = V_B/R)$ is in phase with the voltage V_B , ie the waveform of the voltage V_B represents a scaled version of the waveform of the current I_z and the phase-shift θ between V_z and I_z (ie between the waveforms of V_z and V_B) will correspond to the angle of the impedance Z . Thus, if the waveforms of

V_Z and V_B are displayed, the phase-shift between the two waveforms will correspond to the angle of Z . The magnitude of Z is given by :

$$Z = V_Z/I_Z = V_Z/(V_B/R) = R(V_Z/V_B)$$

The method outlined above will be used in this experiment to demonstrate Z-measurement.

PART (B) : EXPERIMENTAL

All the experimental circuits are constructed by the student on the same modified breadboard system that was used in Expt#2 (see Figure 2.3). The general setup for all three parts of the experiment is shown in Figure 4.7, which shows the breadboard without any circuit constructed on it.

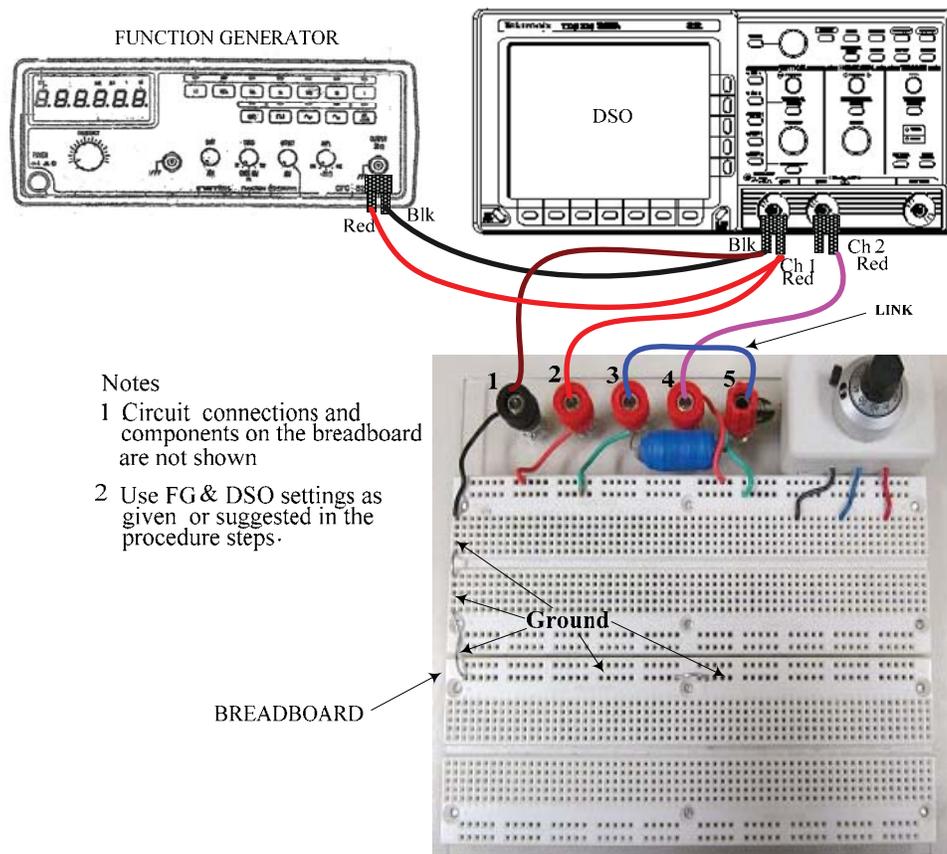


Figure 4.7: General experimental setup

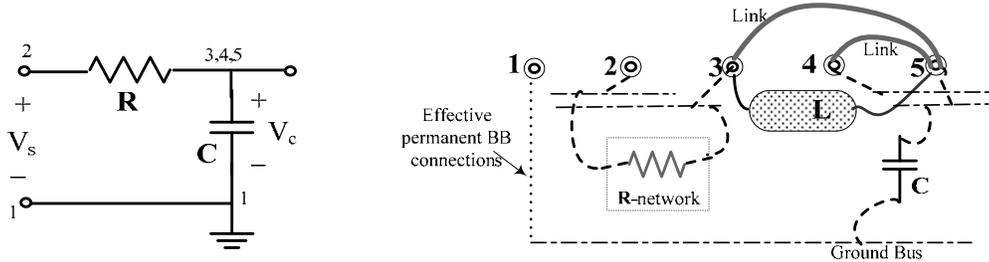
In the setup shown, the FG is first connected to Ch1 of the DSO and the input leads to the test circuit on the breadboard are routed from those terminals, since such an arrangement reduces wiring-clutter. Breadboard terminals #1, #2 and #4 are designated as ground, input and output respectively. [Reminder: A shunting link should be placed between terminals #3 & #5 as shown, when the onboard inductor L is not being used. With such a link in place,

these two terminals can be treated as a single terminal. Also, the inductor and the other permanent connections on the breadboard are not to be removed

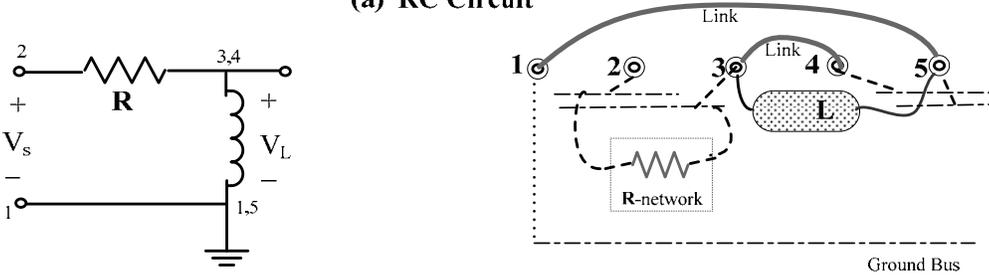
The experimental circuits to be used are given in Figure 4.8, with each test circuit shown on the left side and the corresponding breadboard terminal and pictorial wiring diagram shown at right [the banana-plug patch cord links are shown as solid lines while the BB wiring is indicated by the dotted lines].

Procedure & Results

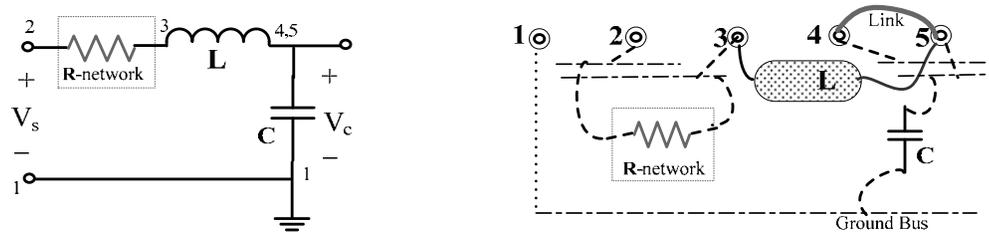
Make the general connections shown in Figure 4.7 and then successively construct the various circuits shown in Figures 4.8 on the breadboard, as required. [The terminal correspondences shown should be maintained for ease of reference]



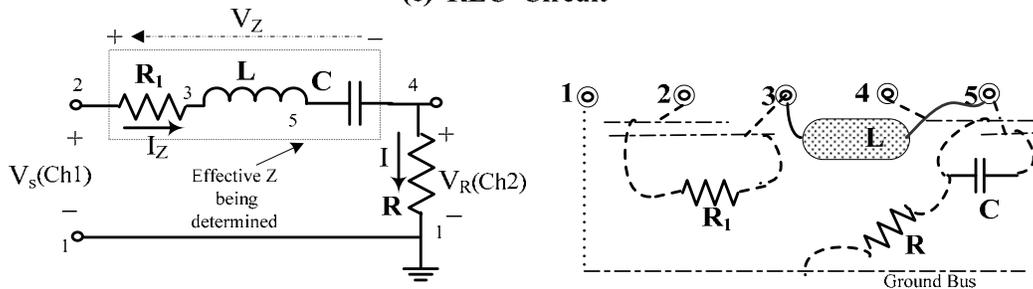
(a) RC Circuit



(b) RL Circuit



(c) RLC Circuit



(d) RLC circuit for demonstration of resonance & Z measurement

Figure 4.8

TRANSIENT RESPONSE: The following settings for the FG & DSO are to be used for the transient response tests:

FG: Waveform: square-wave
Amplitude: ~ 4 volts peak-peak
DC Offset : OFF
Frequency Range: as required

DSO: Ch1 & Ch2 sensitivities: 2 Volts/div or 1 Volt/div
Input Coupling (both channels) : DC
Bandwidth (both channels) : 20 MHz
MEASURE: Display Ch1 & Ch2 Peak-to-Peak amplitudes & Ch1 Frequency
CURSORS: Select 'Paired', and use on Ch2 waveform.

RC Circuit: Figure 4.8(a) shows an RC circuit, in which the step response of V_c is of the typical 'rising exponential' type, $V_c(t) = V_s[1 - e^{-t/\tau}]$, $\tau = RC$

1. Make the connections shown in Figure 4.8 (a) on the breadboard. The effective R used can either be a single resistor or a resistor network configured to obtain some specific value. Use $C = 0.1 \mu\text{F}$ (nominal value; actual C value of caps marked '0.1 μF ' could be in the 80~120 nF range!) and equivalent R-network values in 300Ω to 1500Ω range to obtain a theoretical time constant in the range of 20 μs to 200 μs . [For example (500 Ohms)(88 nF)= 44 μs].

Next, as outlined in the tutorial section, adjust the square-wave frequency on the FG to an appropriate value ($\sim 1/40\tau$) and the time/div switch on the DSO, so that a 'step response' display is seen. [eg: If the 'theoretical' τ is 95 μs , then try a square-wave frequency of $1/40(95\mu)$ or ~ 260 Hz. Note that the DSO will read the frequency of the square-wave only when a few cycles are displayed.]. Adjust the Ch2 sensitivity to obtain a good-sized display for cursor measurement. Position the 'paired' cursors **on the Ch2 waveform** to measure τ : If the H-bar cursors are positioned so that they are at 'zero' & at $\sim 63.2\%$ of the 'final' value of the 'step', then V-bar cursor readout will correspond to τ . A typical display is shown in Figure 4.9(a). Obtain a printout for the report. Compare the measured value of τ with the theoretical value and comment on any differences between the two. Simulate the circuit (in the report) using V_c as output, remembering to use a 4 volt pulsed voltage source for V_s and to perform a 'transient response' analysis. {See Appendix-E}. Compare the result with the experimental result.

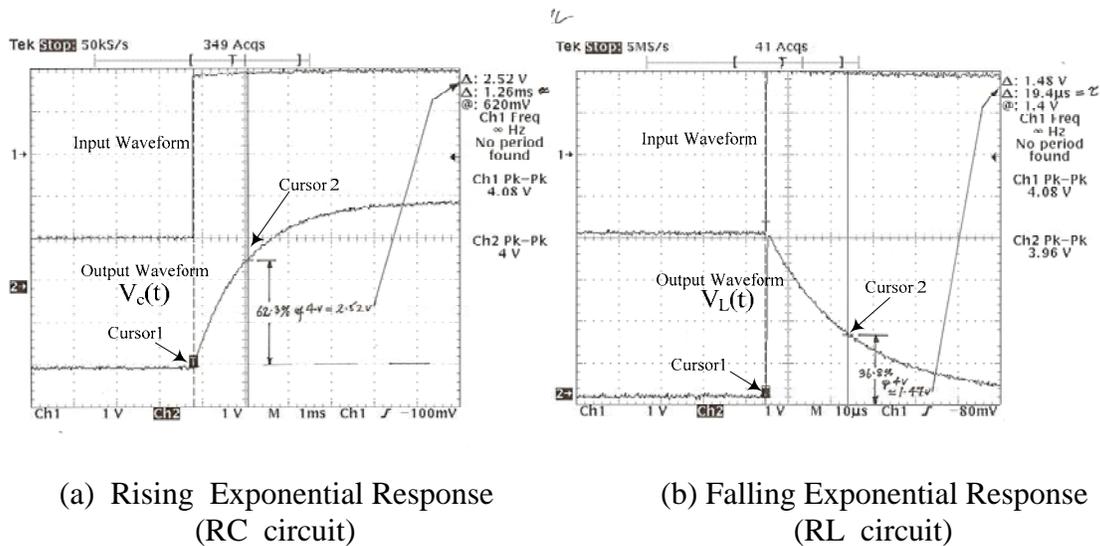


Figure 4.9

RL Circuit: Figure 4.8(b) shows an RL circuit, in which the step response of V_L is of the typical ‘falling exponential’ type $V_L(t) = V_s e^{-t/\tau}$, $\tau = L/R$. The nominal value of the inductor is 20 mH. (The inductor has an ‘internal’ resistance of about 14 Ω .)

- Make the connections shown in Figure 4.8 (b) on the breadboard. The effective R used can either be a single resistor or a resistor network designed to obtain some specific value. Use available R values to obtain a theoretical time constant in the range of 20 μ s to 200 μ s [For example $(0.02)/(1020) = 19.6 \mu$ s or $(0.02)/(110) = 182 \mu$ s]. Repeat the procedure of Step 2 to measure the time-constant of the ‘falling’ response. If the H-bar cursors are positioned so that they are at the ‘initial’ value & $\sim 36.8\%$ of the ‘initial’ value of the ‘step’, then V-bar cursor readout will correspond to τ . A typical display is shown in Figure 4.9 (b). Obtain a printout for the report. Compare the measured value of τ with the theoretical value and comment on any differences between the two. **Simulate** the circuit (in the report) using V_L as output, remembering to use a 4 volt pulsed voltage source for V_s and to perform a ‘transient response’ analysis. {See Appendix-E} Compare the result with the experimental result.

RLC Circuit: Figure 4.8(c) shows a series-RLC circuit, in which the step response of V_c is of the type shown in Figure 4.2. For the circuit $\alpha = R/2L$ and $\omega_n = 1/\sqrt{LC}$, so that for critical damping (CD), $\alpha = \omega_n$ gives the result $R = R_c = 2\sqrt{L/C}$, and R_c is called the Critical Resistance. For $R < R_c$, the circuit response will be under-damped (UD) and for $R > R_c$ the response will be over-damped (OD)

3. Make the connections shown in Figure 4.8 (c) on the breadboard. Use $C = 0.1 \mu\text{F}$, so that the nominal value of R_c is $\sim 894 \Omega$. As before, the effective R used can either be a single resistor or a resistor network designed to obtain some specific value. Use the value of R_c as a guide to configure R-networks to obtain typical OD,CD and UD responses. Apply a square-wave signal (~ 4 volt pk-pk) of the appropriate frequency adjust the DSO to obtain a 'step response' display. Obtain typical printouts for the report, showing the three response types. For each case, indicate the value of R used and the corresponding nominal value of the damping ratio.
4. **Simulate** the circuit (only for the UD condition that you used experimentally) using V_c as output, remembering to use a 4 volt pulsed voltage source for V_s and to perform a 'transient response' analysis. {See Appendix-E}. Compare the result with the experimental result. [eg: compare the magnitude of the 'overshoot' of the output above the 'DCSS' value, for the experimental and simulation curves]

FREQUENCY RESPONSE: The following settings for the FG & DSO are to be used for the frequency response, resonance and impedance tests:

FG: Waveform: Sinusoidal
 Amplitude: ~ 2 volts RMS (~ 3 volts Peak)
 DC Offset : OFF
 Frequency Range: 10 kHz (Max 100 KHz)

DSO: Ch1 & Ch2 sensitivities: 1 Volt/div
 Input Coupling (both channels) : AC
 Bandwidth(both channels) : 20 MHz
 MEASURE: Display Ch1 & Ch2 RMS amplitudes, Ch1 Frequency
 CURSORS: Select 'V-bar' cursors [for Δt (phase) measurement]

Theoretical expressions for the frequency response are easily obtained by analysis of the phasor circuits. For example, for the RC circuit of Figure 4.8(a), applying the voltage-divider rule to the phasor circuit yields;

$$V_c/V_s = \frac{(1/j\omega C)}{R + (1/j\omega C)} = \frac{1}{1 + j\omega CR}$$

from which the magnitude ratio is: $M(\omega) = \frac{1}{\sqrt{1 + (\omega CR)^2}}$

and the phase shift is : $\phi(\omega) = -\tan^{-1}(\omega CR)$.

Expressions for $M = V_c/V_s$ and for ϕ can be similarly obtained for the RLC circuit of Figure 4.8(c):

$$V_c/V_s = \frac{(1/j\omega C)}{R + j\omega L + 1/j\omega C} = \frac{1}{(1 - \omega^2 LC) + j\omega CR}$$

ie
$$M = \frac{1}{\sqrt{(1 - \omega^2 LC)^2 + (\omega CR)^2}} \quad \text{and} \quad \phi = -\tan^{-1} \frac{\omega CR}{(1 - \omega^2 LC)}$$

5. Use the circuits of Figures 4.8(a) and (c) successively (with the corresponding values of R and C used for the transient analysis ; for the RLC circuit, use the UD case values) to obtain the **frequency response** data for V_c of both circuits. . Use the data tables provided..

Both these responses are of the ‘low-pass’ type, with the RLC circuit showing a ‘peak’ near the resonant frequency $\omega_r = 1/\sqrt{LC}$. It is not necessary to obtain printouts for frequency response tests, except perhaps one sample printout. Also, it is not necessary to obtain data in frequency ranges where there is no appreciable change in output. The SETUP button should be pressed each time the frequency is changed, to obtain a frequency readout on the DSO. The RUN/STOP button should be used to ‘freeze’ the amplitude and frequency readings for recording the values in the data tables. *It is efficient to use the ‘frequency’ control knob on the FG to preliminarily ‘sweep’ the frequency in the 100Hz – 10 kHz range and observe the region where the output is changing appreciably and to subsequently record readings only in those regions.*

6. Conventionally, the magnitude ratio M is plotted as a ‘decibel’(db)value where $M(\text{db}) = 20 \log_{10}(M)$ against ω (or f) on semi-log graph paper. Equivalent computer plots using software such as Excel™ may be used, but the results can be more instructive if hand-plotting on commercial graph paper is done. Using the data from Step 5 (and the table entries), plot the magnitude-frequency response (M) versus ω (or f) curves for the RC circuit and the RLC circuit, on 2-cycle [100Hz–1kHz–10kHz] semi-log graph paper. (available from technical stationery suppliers or downloadable from the internet).

RESONANCE : As indicated in the tutorial, a resonance condition is indicated by a ‘peak’ (positive or negative) in the output. The frequency response of the circuit of Figure 4.8(d) is of the ‘band-pass’ type, which has a positive peak at the resonant frequency f_r , with the response falling off on either side of f_r . The sharpness of the resonant peak (or ‘selectivity’) is characterized by a parameter known as the ‘Quality Factor’ Q which is a measure of the ratio of stored energy to dissipated energy in the circuit. For the series-RLC circuit, $Q = \omega_r L/R$. Q can be graphically found from the ‘band-pass’ characteristic: If f_{c1} and f_{c2} ($> f_{c1}$) are the two frequencies on each side of f_r , where the magnitude response has fallen to 0.707 of the maximum, it can be shown that $Q = f_r/\Delta f$ where $\Delta f = f_{c2} - f_{c1}$ is called the ‘half-power bandwidth’. Thus, a high Q value is associated with a narrow bandwidth or high selectivity. For second-order circuits, the damping ratio ζ and Q are related by the equation $Q = 1/2\zeta$.

For the circuit of Figure 4.8(d), the impedance of the breadboard circuit (at terminals 2,1) is

$$Z = Z_{21} = R_1 + j [\omega L - 1/\omega C] + R$$

At the resonant frequency $f = f_0 = 1/2\pi\sqrt{LC}$ the circuit appears as a pure resistance with $Z = (R_1 + R)$. The resonance frequency can be identified by varying the frequency until the V_s and V_R waveforms are in-phase, since the V_R waveform is a scaled version of the $I (=I_z)$ waveform. A convenient way to monitor the resonance condition (V_s & V_R in phase) is to display the two signals in the XY-mode (signals plotted against each other) on an oscilloscope. The DSO used in the lab offers this mode under the DISPLAY menu. An XY display results in the so-called ‘Lissajous’ figure, which is a pattern from which frequency and phase relations between the two signals can be obtained. It can be shown that a straight line with a 45° slope, as seen in Figure 4.10, occurs when the two signals of the same frequency are in phase. The corresponding YT-mode display, indicating the in phase condition, is also shown for reference.

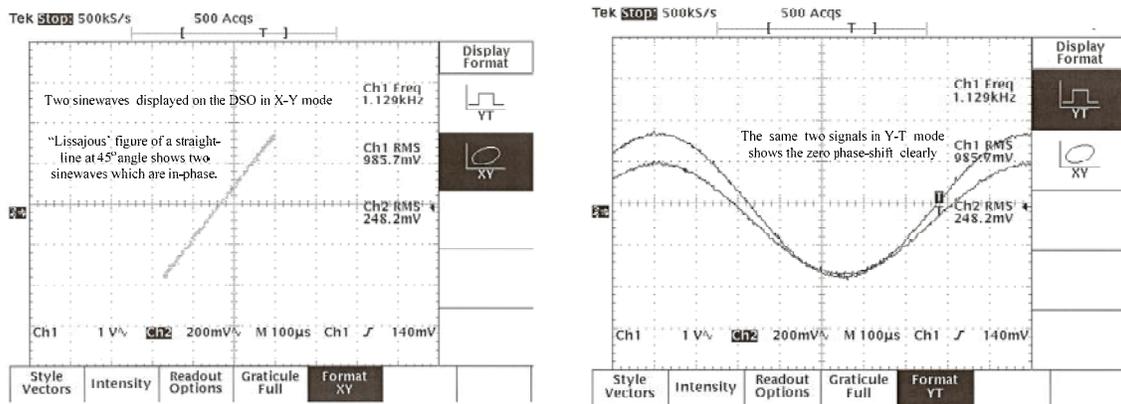


Figure 4.10

7. Make the connections shown in Figure 4.8 (d). Use $R_1 = 0 \Omega$, $R = 100 \Omega$, and $C \approx 0.1 \mu F$ so that, with $L = 0.02 H$, the series-resonant frequency f_r is approximately 3.6 kHz. The impedance Z , with these component values, is

$$Z = 100 + j [0.02\omega - 10^7/\omega] = 100 + j [0.12566 f - 1.592(10)^6/f]$$

Using the component values given above, slowly vary the frequency around 3.5 kHz to find the resonant frequency, i.e. the frequency at which the output reaches a maximum. Then obtain output readings on both sides of f_r , over the range of 1 kHz to 6 kHz. Use the data tables provided.

8. Using the data of Step 7, plot the band pass characteristics of the circuit, obtain the Quality Factor Q . From the plot and compare it with the theoretical value: With the nominal L and C values,

$$R_c = 2\sqrt{L/C} = 894.43, \quad \text{hence} \quad \zeta = R/R_c = 100/894.43 = 0.11$$

and $Q_{\text{Theoretical}} = 1/2\zeta = 1/0.22 = 4.47.$

MEASUREMENT OF IMPEDANCE is an important requirement in practice. As indicated in Appendix-C, impedance Z can be ‘inductive’ (general form $Z = R_e + jX_e$) or ‘capacitive’ (general form $Z = R_e - jX_e$). For a series RLC circuit, it can be shown that for frequencies $f < f_r$, Z is capacitive, whereas for $f > f_r$, it is inductive. The method explained in the tutorial section together with the circuit of Figure 4.7(d) will be used to demonstrate impedance measurement.

9. Use the same circuit as in Step 7 above, but use $R_1 \approx 500 \Omega$ [$R = 100 \Omega$ & $C \approx 0.1 \mu\text{F}$ as before]. Since $f_r \approx 3.56 \text{ kHz}$, capacitive and inductive impedances can be obtained using an excitation frequency below and above this value, respectively. Use the same DSO settings used for “Frequency Response”, but additionally switch on the ‘Math Channel’ $M = \text{Ch1} - \text{Ch2}$ and display its RMS amplitude. The M channel waveform corresponds to the voltage V_z across Z .
10. Obtain a printout at a frequency $f \approx 2 \text{ kHz}$ and another at a frequency $f \approx 5 \text{ kHz}$, each showing the three waveforms, their RMS magnitudes and the V-bar cursors positioned to indicate the time-shift Δt between the M and Ch2 waveforms. A typical DSO display (with $R_1 = 504 \Omega$ and $f = 2083 \text{ Hz}$) is shown in Figure 4.11. (Make sure to obtain your TA’s signature on the printouts)

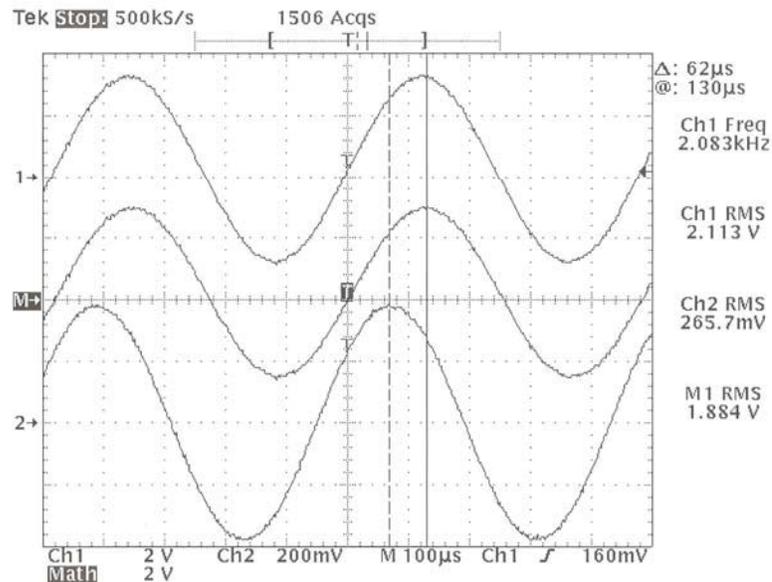


Figure 4.11

In the example shown in Figure 4.11, Z is capacitive since $f < f_r$ and its magnitude is $100(1.884/0.2657)$ or 709Ω and its angle is $-360(2083)(62\mu)$ degrees or -47° .

Using the nominal known component values, the corresponding 'expected' value of Z is

$$\begin{aligned} Z &= (504+14)+ j[0.02\omega-10^7/\omega] \\ &= 518 + j(261.75-764.28) \\ &= 518-j502.53 = 721.7 \angle -44.1^\circ \Omega. \end{aligned}$$

The errors in the measured values, relative to the 'theoretical values' are approximately -1.8% and $+6.6\%$, for the magnitude and angle, respectively. The V_z (M) waveform is seen to lag the I_z (Ch2) waveform, as it should in a capacitive impedance.

For the two printouts at 2kHz & 5 kHz that you obtained in Step 10, find the corresponding experimental capacitive and inductive impedances and compare them with the 'theoretical' values calculated using the nominal elemental values and the appropriate frequency.

DATA TABLES

TRANSIENT RESPONSE DATA

Steps 1 to 3 :

(a) Values used for the series-RC circuit :

Nominal Value* of $R = \dots\dots\dots$ Ohms

Nominal Value of $C = 0.1 \mu\text{F}$

‘Theoretical’ Time Constant $\tau = RC = \dots\dots\dots$

*Neatly draw the circuit of the R –network which you used , to obtain the R value, in the space below.

(b) Values used for the series-RL circuit :

Nominal Value* of $R = \dots\dots\dots$ Ohms

Nominal Value of $L = 0.02 \text{ H}$

‘Theoretical’ Time Constant $\tau = L/R = \dots\dots\dots$

* Neatly draw the circuit of the R –network which you used , to obtain the R value, in the space below.

BLANK

Step 4 : Values used for the series-RLC circuit :

(c)

$$R_c(\text{nominal}) = 2\sqrt{0.02/0.1\mu} = 894.43 \Omega$$

Value of R used for 'OD' response, $R_{od} = \dots\dots\dots$ Ohms

Value of R used for 'CD' response, $R_{cd} = \dots\dots\dots$ Ohms

Value of R used for 'UD' response, $R_{ud} = \dots\dots\dots$ Ohms

TA Signature :

[Make sure to also obtain your TA's signature on the various time- and frequency- response printouts]

BLANK

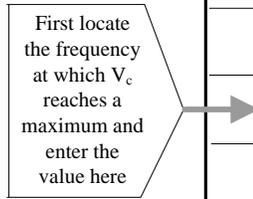
BLANK

DATA FOR Series-RLC RESONANT CIRCUIT, (Step 7)

[First locate the frequency at which V_c is a maximum and then ‘descend’ on both sides of the peak frequency. This will ensure proper plotting of the ‘bandpass’ curve]
 Use a V_s magnitude in the 1 to 3 Volts RMS range

Attempt to maintain V_s at a constant level

First locate the frequency at which V_c reaches a maximum and enter the value here



f kHz	Input V_s Volts RMS	Output V_R Volts RMS	$M(\text{db}) = 20 \log \frac{V_c}{V_s}$
1			
1.5			
2			
2.5			
3			
4			
4.5			
5			
5.5			
6			

TA Signature :

BLANK

EXPERIMENT 5

AC POWER MEASUREMENTS

OBJECTIVES : Part A: AC Power Measurements on Resistive, Inductive and Capacitive Loads

Part B: Power-Factor-Correction of an Inductive load

PRE-REQUISITES: 1. Familiarity with the Power equipment descriptions given in Appendix-D
2. Reading and making an attempt to understand the tutorial material given below, before coming to the lab session.

TUTORIAL

The measurement of AC power is an important part of electrical engineering practice. In this experiment, the student will become familiar with basic AC power measurements at the commercial AC line frequency of ~60 Hz.. The procedure known as “Power Factor Correction”(PFC) which is used to improve power efficiency in industrial consumption, will also be studied.

Consider a general load shown in both time and phasor domains in Figure 5.1.

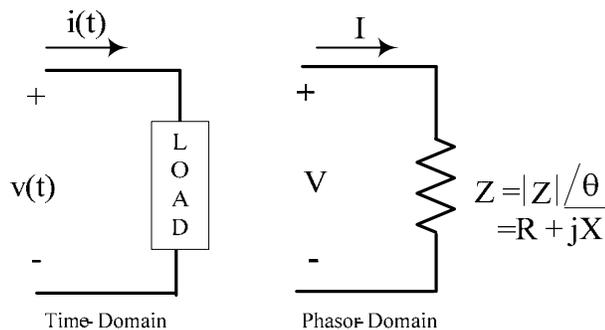


Figure 5.1

In general, the load will contain both dissipative and reactive(storage)elements and can be represented in the phasor domain by an impedance $Z (= R + jX) = |Z| \angle \theta^\circ$ as shown on the right hand side in the figure. Power dissipation is associated with the resistive component R of the load, whereas the interchange of stored energy in the reactance X will account for a ‘reactive’ component in the overall power. The instantaneous power will therefore contain an average (or ‘steady’) component reflecting real power dissipation and a sinusoidally fluctuating component reflecting energy interchanges in the reactive elements. These components can be quantitatively expressed by the following definitions.

AC Power definitions: There are several definitions associated with AC power which are derived from the basic expression for the instantaneous power $p(t) = v(t)i(t)$, which is the power absorbed by a load impedance Z at any given instant. If the time-domain voltage and current associated with the load are expressed as :

$$v(t) = V_p \cos(\omega t + \phi_v) \quad \text{and} \quad i(t) = I_p \cos(\omega t + \phi_i)$$

where V_p , I_p are peak values, and ϕ_v , ϕ_i are the phase angles (with respect to an arbitrary zero-phase reference) of the voltage and current phasors respectively, the instantaneous power $p(t)$ can be written as

$$\begin{aligned} p(t) &= V_p I_p \cos(\omega t + \phi_v) \cos(\omega t + \phi_i) \\ &= (V_p I_p / 2) \cos(\phi_v - \phi_i) + (V_p I_p / 2) \cos(2\omega t + \phi_v + \phi_i) \end{aligned}$$

The difference between the voltage and current phase angles ($\phi_v - \phi_i$) is actually the angle θ of the impedance Z . The expression for $p(t)$ can be simplified by expressing the voltage and current in their RMS values $V = V_p / \sqrt{2}$ and $I = I_p / \sqrt{2}$ (which is the convention in phasor circuits) and by choosing the current phasor as the arbitrary reference (i.e. $\phi_i = 0$) so that $\theta = \phi_v$. After making these substitutions, we obtain

$$p(t) = VI \cos \theta + VI \cos(2\omega t + \theta) = P + VI \cos(2\omega t + \theta)$$

It is seen that $p(t)$ has an average value $P = VI \cos \theta$, which is independent of the frequency ω , and further, $p(t)$ fluctuates about this average at twice the frequency of $v(t)$ & $i(t)$. P is measured in 'watts' and is called the Average Power (or Real Power) and it represents the power dissipated in the load. P is measured using Wattmeters (see Appendix-D). The factor ' $\cos \theta$ ' in P is called the Power Factor (commonly abbreviated as PF or pf). The product VI , called the Apparent Power P_{app} (or RMS Power, P_{RMS}), is simply the product of the load voltage and RMS load current as measured using rms-reading meters, i.e. $P_{app} = VI$. Hence $PF = \cos \theta = P/P_{app}$. Further expansion of the above expression for $p(t)$ yields:

$$\begin{aligned} p(t) &= P + VI \cos(2\omega t + \theta) = P + VI \cos 2\omega t \cos \theta - VI \sin 2\omega t \sin \theta \\ &= P [1 + \cos 2\omega t] - Q \sin 2\omega t \end{aligned}$$

where $Q = VI \sin \theta$ is called the Reactive Power measured in the unit 'var', which stands for 'volt-amperes-reactive'. Q represents the (non-dissipative) rate of change of energy associated with the reactive component of the load impedance. Q is positive for an inductive load (θ positive, between 0° and 90° , or I lagging V) and negative for a capacitive load (θ negative between 0° and -90° , or I leading V). The complex quantity $S = P + jQ$ is called the Complex Power (unit: Watt). From the defining equations for P and Q , we have $Q = P \tan \theta$, and hence Q can be calculated from measured values of P and θ , the latter being obtained from $\theta = \cos^{-1}(P/P_{app})$.

The following alternative equations for the calculation of P & Q are often found to be convenient: Since the real part of Z , $\text{Re}Z = |Z| \cos \theta = (V/I) \cos \theta$, we have $V \cos \theta = I \text{Re}Z$ and hence $P = VI \cos \theta = I^2 \text{Re}Z$. For any given load impedance $Z = R + jX$, $\text{Re}|Z| = R$ and the average power $P = I^2 \text{Re}Z = I^2 R$, Watts. Similarly, using the imaginary part of Z , we obtain

$Q = VI \sin\theta = I^2 \text{Im}Z = I^2 X$. P and Q will have finite values when the load impedance is neither purely resistive nor purely reactive .

Power Factor Correction (PFC): The PF is a very important parameter in AC power systems since it determines the line current magnitude for any given set of values of line voltage V and average power P ie: since $P = VI \text{ (PF)}$, the current $I = \frac{P}{V(\text{PF})}$. In

industrial situations , it is of interest to utility companies, for the sake of efficiency, to maximize the PF of a customer's load in order to minimize the line current I, since this will reduce ohmic losses in the supply lines. Most industrial loads are inductive due to the fact that AC motors are the major 'workhorses'. The addition of a capacitor in parallel with an RL load results in a change in PF (towards unity PF) . Hence the PFC procedure requires the connection of a 'PFC-correcting' capacitor C, in parallel with the inductive load. The PF change must only be due to a change in the overall reactive power without affecting the average power P. The value of C required can be found using the change in Q required for obtaining the desired PF. This is demonstrated by the following example:

Problem: A 500 kW, 240 VRMS, 60 Hz industrial load is operating at a PF of 0.6, lagging*. Determine the value of a PFC capacitor to be connected in parallel in order to correct the PF to 0.78, lagging.[*The added term 'lagging' indicates that the current lags the voltage, thus implying an inductive load].

Solution: $Q_{\text{old}} = P \tan \{\cos^{-1}\text{PF}_{\text{old}}\} = 500 \tan \{\cos^{-1}0.6\} = 666.67 \text{ kvar}$

$$Q_{\text{new}} = P \tan \{\cos^{-1}\text{PF}_{\text{new}}\} = 500 \tan \{\cos^{-1}0.78\} = 401.14 \text{ kvar}$$

$\Delta Q = Q_{\text{new}} - Q_{\text{old}} = 401.14 - 666.67 = -265.53 \text{ kvar}$.This ΔQ change is only due to the added capacitor impedance $Z_c = 1/j\omega C = -j/\omega C$ and since $\Delta Q = I^2 \text{Im} Z_c$, we have

$$\Delta Q = I^2 \text{Im} Z_c = [V / (1/\omega C)]^2 (-1/\omega C) = -\omega C V^2$$

Equating $-\omega C V^2 = \Delta Q = -265.53 \text{ var}$, we obtain

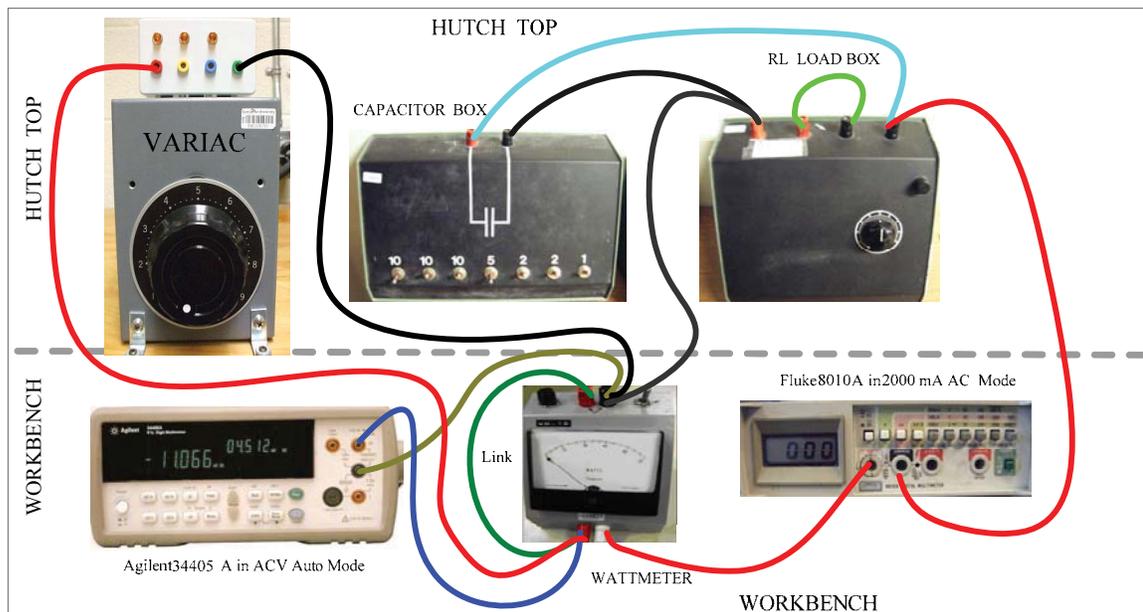
or $C = \Delta Q / \omega V^2 = 265.53 / 2\pi(60) (240)^2 = \underline{12.23} \mu\text{F}$

CAUTION: THIS EXPERIMENT INVOLVES CURRENTS & VOLTAGES WHICH CAN CAUSE ELECTRICAL SHOCK, IF IMPROPER PROCEDURES ARE USED. IN ORDER TO AVOID SUCH HAZARDS, PLEASE OBSERVE THE FOLLOWING PRECAUTIONS:

- ◆ ALWAYS return the Variac knob to its zero position after taking each set of readings and/or before making any circuit change.
- ◆ DO NOT daisy-chain patchcords ; Use proper patchcord lengths and colour codes.
- ◆ DOUBLE-CHECK your circuit connections before applying power, every time.
- ◆ NEVER leave the R-control in the load unit in its minimum position.
- ◆ The RL load & the C-box should not be moved from their positions on top of the hutch [see Figure 5.2].

EXPERIMENTAL

Use the general wiring diagram shown in Figure 5.2 below, taking care to use patchcords of appropriate lengths and colours. The RL load & the C-box should not be moved from their positions on the hutch. Please return the wattmeter (& the Fluke DMM if applicable) to their original locations after use.



CIRCUIT SHOWN IS FOR A LOAD OF A SERIES RL IN PARALLEL WITH C [PFC Circuit]

Figure 5.2: General Setup for Expt #5

Procedure & Results

POWER MEASUREMENTS: In this part of the experiment, measurements are made on three specific loads: (1) a ‘resistive’ load $Z = R$, (2) an ‘inductive’ load, with the two elements of the RL-box connected in series, $Z = R + j\omega L$, and (3) a ‘capacitive’ load, with R from the RL-box and C from the C-box connected in series, $Z = R + (1/j\omega C)$. The general circuit shown in Figure 5.2 is used **with appropriate modifications to the manner in which the RL-box and C-box are connected.** The circuit for power measurements is shown in Figure 5.3. The circuit is the schematic equivalent of that shown in Figure 5.2, but the load is shown as a general impedance Z .

Before starting the experiment it is advisable to check the fuses on the RL-box and on the wattmeter !

1. **Calibrate** the rheostat in the RL-unit by using either DMM as an ohm-meter to find the actual value of R corresponding to each dial setting of 10 to 90 in steps of 10. Use the table provided. [An LCR-Meter is available at the Printer Bench. It can be used to measure L & C if required. However, the nominal value of $L = 0.2$ H may be used for calculations. [The value of L marked on the RL-box may not be accurate. The capacitance box values need not be measured.]

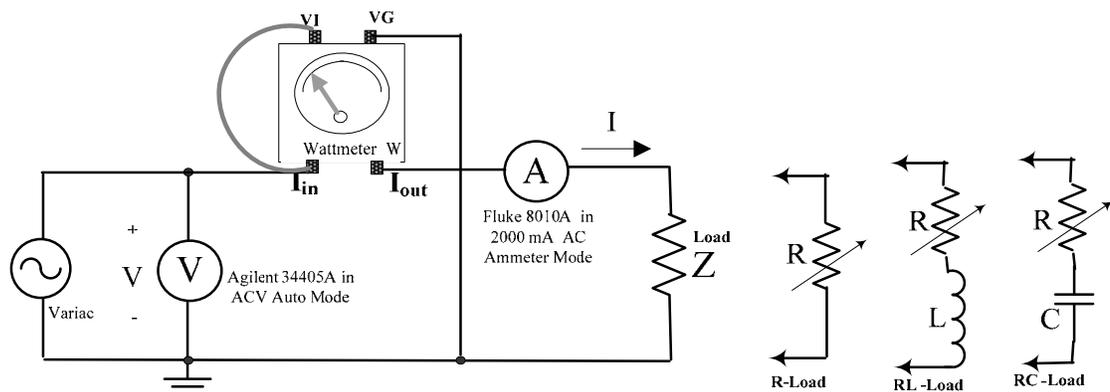


Figure 5.3 : Circuit for power measurements

2. **RESISTIVE LOAD:** Set up the circuit of Figure 5.3 with the R load. {This can be achieved in Figure 5.2 by disconnecting the C-box at the RL-load end, and then altering the RL-load connections so that only R is connected }
- (a) Set the variac control to zero and the R-dial to 40. Slowly increase the variac setting until the Fluke 8010A ammeter reading is about 1 Amp [**Do not exceed 1A !**] Note this current value together with the corresponding power P read by the wattmeter and the corresponding input voltage indicated by the Agilent 34405A voltmeter and enter them in the appropriate data table .
 - (b) Next, slowly increase R to two additional values and tabulate the corresponding I and P values [The voltage V should not change appreciably]

{Note:The above Steps 2(a) & 2(b) will be basically repeated for the remaining experimental loads , with a specific load modification being made in each case }

3. **INDUCTIVE LOAD:** Set up the circuit of Figure 5.3 with the RL load [R and L in series , so that $Z = R + j\omega L \approx R + j 75.4$, Ohms.] {This can be achieved in Figure 5.2 by simply disconnecting the C-box at the RL-load end }

Repeat steps 2(a) & 2(b) above **but with R set at 10 as the starting value.** Use the data table.

4. **CAPACITIVE LOAD:** Set up the circuit of Figure 5.3 with an RC load [R from the RL-box and C from the C-box in series , with all switches toggled upwards so that so that $C = 40\mu\text{F}$, $Z = R - j / \omega C \approx R - j 66.3$, Ohms]. {This can be achieved in Figure 5.2 by simply removing the central (green wire)link between the R & L element on the RL-load box and moving the (blue wire) lead from the capacitor to the other R terminal }

Repeat steps (a) & (b) above **but with R set at 10 as the starting value.** Use the data table.

Results [Power Measurement] : For each load case, calculate the experimental PF from the tabulated values of V,I & P and compare the results with the expected value , which is 1 for $Z=R$, and $\cos [\tan^{-1}(75.4/R)]$ and $\cos [\tan^{-1}(-66.3/R)]$ for the inductive and capacitive cases, respectively. Choose any one set of data from the inductive or capacitive load cases and calculate the corresponding value of the complex power S.

POWER FACTOR CORRECTION: Set up the PFC circuit of Figure 5.4 with the variac at its minimum, the R-dial set to **30** and with all switches on the C-box toggled down so that $C = 0$. This circuit corresponds to the setup of Figure 5.2, without any modifications.

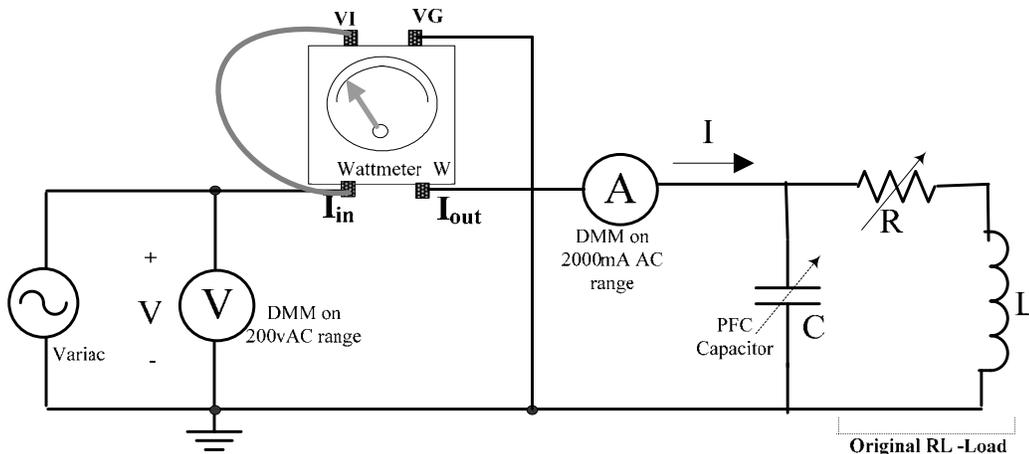


Figure 5.4

5. Slowly increase the variac voltage until the ammeter reading is about 1A **[Do not exceed 1A !]**. Record the I , P and V values in the first row of the PFC data table Next, successively switch in C values from $1\mu\text{F}$ to about $20\mu\text{F}$ **and observe that**

the current I decreases while the average power P indicated by the wattmeter remains unchanged. Measure and record I & P values corresponding to at least three different (and well-spaced out) C values in the range of 4 μ F to 20 μ F , for example 5 , 9, 12 ,15,19, μ F etc. Enter the data in subsequent rows of the PFC table. Determine the value of $C = C_{\min}$ which results in a minimum value of current.

6. Calculate ($PF = P/VI$) and enter all the experimental PF value in column 3 of the PFC table. [Verify that the PF value for $C = 0$ {ie original RL load) is close to the expected value : ie If the R dial setting of 30 corresponds to 33 Ω , for example, the or PF of the original RL-load will be $\sim \cos(\tan^{-1} 75.4/33)$ or 0.4]

Results [PFC] :

- (a) From the “PFC- corrected” data of the PFC table, select three different representative cases PF corresponding to three “corrected” values of PF.
- (b) Assume the three “corrected” values in (a) above to be “desired” PFs .Then calculate the values of C that would be required to obtain these “corrected” PFs and compare the calculated values of C with the corresponding practical values recorded in the first column of the PFC table
- (c) **Simulate** the circuit of Figure 5.4 (without the wattmeter!) using $R= 30$ Ohms, $L = 0.2$ H and $C = C_{\min}$ found in Step 5 above. Use AC analysis to obtain the plot of current versus frequency (ie a frequency response plot) .If the parameters are set properly, the simulation should show that the current is a minimum at the line frequency of $f \approx 60$ Hz.

BLANK

DATA TABLES

Rheostat Calibration

DIAL SETTING	10	20	30	40	50	60	70	80	90
ACTUAL RESISTANCE R									

Power Measurements

Resistive Load
Voltage V = volts

R DIAL SETTING	I(A)	P(W)	PF=P/VI
40			

Inductive Load L = H
Voltage V = volts

R DIAL SETTING	I(A)	P(W)	PF=P/VI
10			

Capacitive Load C = F
Voltage V = volts

R DIAL SETTING	I(A)	P(W)	PF=P/VI
10			

PFC Correction Data: Original Load R = , L = H
Voltage V = volts , P = Watts

C (μF)	I(A)	PF=P/VI	C (F)	I(A)	PF=P/VI
Orig C = 0			22		
2					
5					
10					
20			40		

TA Signature

BLANK

APPENDIX – A

DESCRIPTIONS OF MEASURING EQUIPMENT

The Fluke 8010A Digital Multimeter (DMM)

A *multimeter* is an instrument which can be configured to measure circuit variables such as current and voltage, as well as other parameters such as resistance. *Ideal* meters are fictitious instruments which will not disturb or change the circuit in which they are being used to measure a given quantity. Thus, an ideal ammeter must behave like a short-circuit and an ideal voltmeter must behave like an open-circuit. The characteristics of a DMM closely approach those of an *ideal* meter, thereby improving the accuracy of measurements. DMMs are available with very advanced measurement capabilities, but the basic DMM is usually designed to measure current, voltage and resistance. A typical meter of the latter kind, the **Fluke Model 8010A**, is used in the lab. A short description of the front panel items of this DMM follows.

Front panel features of the Fluke 8010A DMM: The front panel of the Model 8010A used in the lab is shown in Figure A1. Descriptions of the various controls are given below Figure A1.

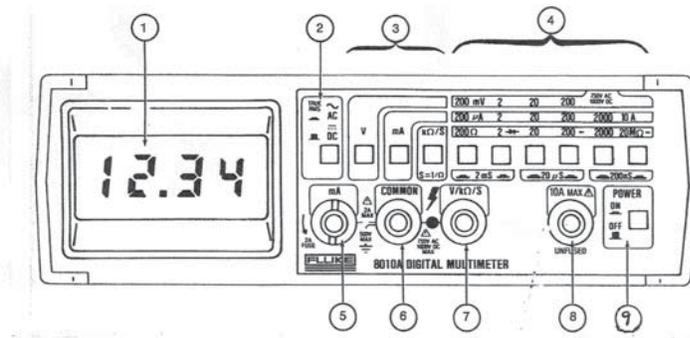


Figure A1 : Front panel of the DMM

Description of front panel items (according to the indicated item number in Figure A4)

1. Numerical LCD Display
2. DC/AC Switch : Push to 'out' position for DC measurements, 'in' for AC)
3. MODE buttons : Push 'in' : 'V' for voltmeter mode, 'mA' for ammeter mode, 'kΩ/S' for resistance or ohmmeter mode (For Conductance S, push in 'kΩ/S' **and pairs** of range buttons as indicated, eg 200Ω & 2kΩ pressed together gives the 2mS range)
4. RANGE buttons : Push 'in' to select the range indicated above the button
5. **Current Input Terminal** with a 'built-in' 2A Fuse: DC current flowing into this terminal and out of the COMMON terminal(6) , will produce a 'positive' indication on the display. The positive indication has no '+' sign displayed. If the current enters the COMMON terminal and leaves terminal 5, then a negative sign precedes the displayed value of the current . **There is a 2A fuseholder built into this terminal.** It can be accessed by turning the terminal by 90° counterclockwise and then lifting out the terminal. The fuse in the fuseholder must only be replaced with a 2A (AGX-2) fuse if the ammeter mode is not working. Replacement of the fused-terminal is in the reverse order of the removal steps.
6. COMMON (or Reference Ground) terminal (for ALL modes)
7. **Voltage (and Ohmmeter) Terminal** : A positive voltage (with respect to the COMMON terminal) applied to this terminal will produce a 'positive' indication on the display. The positive indication has no '+' sign displayed. If the voltage applied to this terminal is

negative with respect to the COMMON terminal , then a negative sign precedes the value displayed .

If an **isolated resistance** R is connected between this terminal and the COMMON terminal, and the appropriate resistance range is selected, the value of R is displayed.

8. Special **10A (high current)** current input terminal, for currents > 2A. The ‘mA’ and ‘10A’ buttons must **both** be pressed. **THIS TERMINAL IS NOT FUSED AND CURRENTS FLOWING INTO IT MUST NOT EXCEED 10A !!**
9. ON/OFF or POWER button : Push ‘in’ to turn ON , push ‘out’ to turn OFF.

SPECIFICATIONS of the Fluke 8010A DMM

Voltmeter Mode: [DC or effective-value(RMS-value) on AC]

Input Impedance* : DC :10 MΩ ; AC : 10 MΩ in parallel with < 100pF, all ranges

Accuracy : ± 1 % or better, DC to 20 kHz (max . frequency 200kHz)

Response time : < 1 sec

Ammeter Mode: [DC or effective-value(RMS-value) on AC]

Accuracy : ± 2 % or better, DC to 20 kHz (max . frequency 200kHz)

Maximum Burden Voltage * : 0.3v upto 200mA range, 0.9v on 2000mA range

Response time : < 2 sec

Ohmmeter Mode: Accuracy : ± 0.5 % or better, all ranges.

- These specifications are indicative of the closeness of the DMM to the “ideal meter” conditions.

CAUTION : . **The proper mode must be selected on the DMM, to make it function as an ammeter, voltmeter or Ohm-meter. Always set the DMM in the required mode BEFORE connecting it! In particular never connect the ammeter terminals directly to a voltage output.**

The Agilent 34405A Digital Multimeter (DMM) is a more recent DMM model , which has advanced capabilities which include Frequency ,Capacitance and Temperature measurement.This DMM has **auto-ranging** capability as well.

The front panel features of the Agilent 34405A together with identification of the various controls are given in Figure A2.

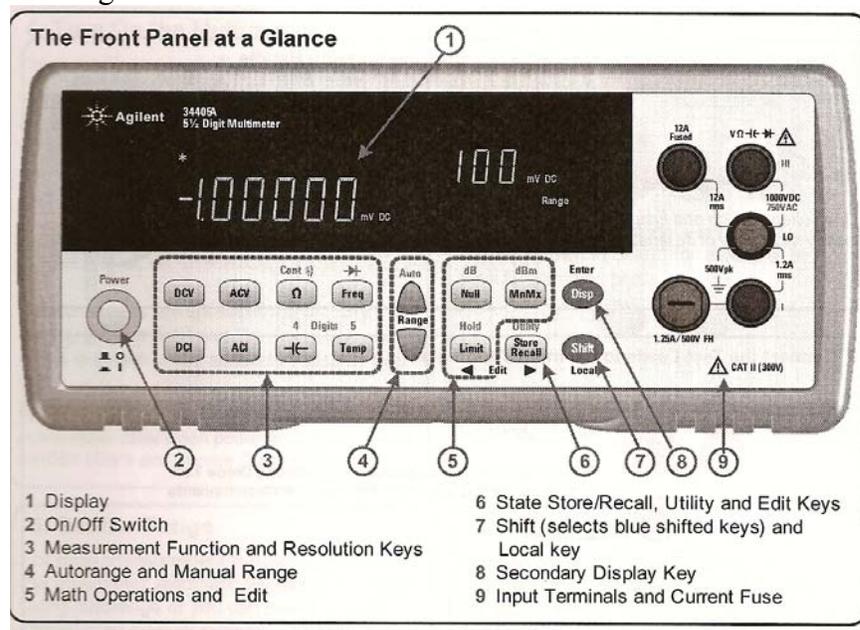


Figure A2

The detail of the input terminals and the basic setup steps are given in Figures A3(a) & A3(b)

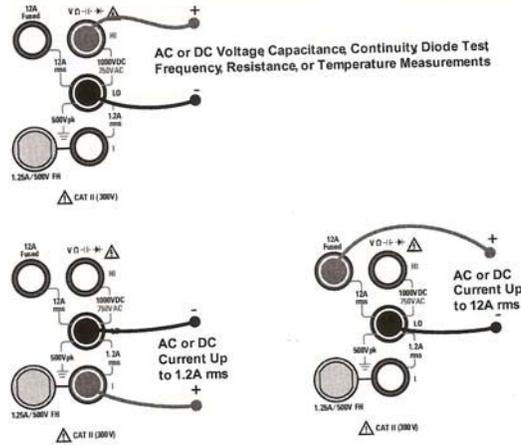


Figure A3(a)

4. Turn On the Multimeter

Press the Power (on/off) switch:
The front-panel display illuminates while the multimeter performs its power-on self-test. Following self-test, the multimeter defaults to DC voltage measurements.* Typical display:

Measurements are shown in the primary display (bottom left) and the measurement range is shown in the secondary display (top right).

**You can set the multimeter to return to the last power-down state when power is applied. See the 34405A User's and Service Guide for details.*

5. Select a Function

Press one of these keys to select a measurement function:

DCV	ACV	Ω	Freq
DCI	ACI	←	Temp

The continuity and diode test functions are shifted (shown in blue). For example, to select continuity, press:

Shift	Ω
-------	---

Press **DCV** to return to the DC voltage function.

6. Set the Range

For most measurement functions, you can let the multimeter automatically select the range using autorange or you can select a fixed range using manual ranging. The **ManRng** annunciator is displayed when manual ranging.

Selects a higher range and disables autoranging

Selects a lower range and disables autoranging

To return to autoranging and disable manual ranging, press:

7. Set the Resolution

You can select either 4½ or 5½ digit resolution for most measurement functions. 5½ digit readings have the best accuracy and noise rejection. 4½ digit readings provide for faster readings.

4 Digits

Selects 4½ digit mode

Digits 5

Selects 5½ digit mode

Note: The continuity and diode test functions have a fixed, 4½ digit display. Capacitance and temperature have a fixed, 3½ digit display.

Figure A3(b)

Specifications of the Agilent 34405A are given below;

<p>DC VOLTAGE Measuring Method: Sigma Delta A-to-D converter Input Resistance: 10 MΩ \pm 2% range (typical) Input Protection: 1000 V on all ranges</p>	<p>AC VOLTAGE Measurement Method: AC coupled true-RMS - measure the ac component with up to 400 Vdc bias any range Crest Factor: Maximum 5:1 at full scale Input Impedance: 1 MΩ \pm 2% in parallel with < 100 pF of all ranges Input Protection: 750 Vrms on all ranges</p>				
<p>RESISTANCE Measuring Method: 2-wire Ohms Open-circuit voltage: Limited to < 5V Input Protection: 1000 V on all ranges</p>	<p>AC CURRENT Measuring Method: DC coupled to the fuse and current shunt, AC coupled true-RMS measurement (measure the AC component only) Shunt Resistance: 0.1 Ω to 10 Ω for 10 mA to 1.2 A range 0.1 Ω for 12 A range Input Protection: Externally accessible 1.25 A, 500 V fuse for I terminal Internally replaceable 15 A, 600 V fuse for 12 A terminal</p>				
<p>DC CURRENT Shunt Resistance: 0.1 Ω to 10 Ω for 10mA to 1.2 A ranges 0.01 Ω for 12 A range Input Protection: Front Panel 1.25 A, 500 V fuse for I terminal Internal 15 A, 600 V fuse for 12 A terminal</p>	<p>FREQUENCY Measurement Method: Reciprocal counting technique. AC coupled input using AC voltage function. Signal Level: 10% of range to full scale input on all ranges Auto or manual range selection Gate Time: 0.1 second or 1 period of the input signal, whichever is longer. Input Protection: 750 Vrms on all ranges</p>				
<p>CONTINUITY/DIODE TEST Measuring Method: Uses 0.83 mA \pm 0.2% constant current source, < 5 V open circuit voltage Response Time: 70 samples/second with audible tone Continuity Threshold: 10 Ω fixed Input Protection: 1000 V</p>	<p>MATH FUNCTIONS Null, dBm, dB, Min/Max/Avg, Hold, Limit Test</p>				
<p>TEMPERATURE Measurement Method: 2-wire Ohms measurement of 5 kΩ thermistor sensor (E2308A) with computer conversion Auto-ranging measurement, no manual range selection Input Protection: 1000 V</p>	<p>TRIGGER and MEMORY Single trigger, 1 reading memory</p>				
<p>MEASUREMENT NOISE REJECTION CMRR (Common Mode Rejection) for 1 kΩ unbalance LO lead</p> <table border="1"> <tbody> <tr> <td>DC</td> <td>120 dB</td> </tr> <tr> <td>AC</td> <td>70 dB</td> </tr> </tbody> </table>	DC	120 dB	AC	70 dB	<p>REMOTE INTERFACE USB 2.0 full speed, USBTMC-USB488 device class</p>
DC	120 dB				
AC	70 dB				
	<p>PROGRAMMING LANGUAGE SCPI, IEEE-488.1, IEEE-488.2</p>				

The Instek GFG-8216A FUNCTION GENERATOR : The front panel of the GFG8216A function generator is shown in Figure A4, followed by brief descriptions of various controls. **The student should become familiar with all the features of the FG.** The FG also has two other input connectors on the rear panel, which are not used for this lab. [The front panel button labelled “Gate” operates only when the ‘counter’ input on the rear panel is used, and is not described in the explanation of terms following Figure A4]

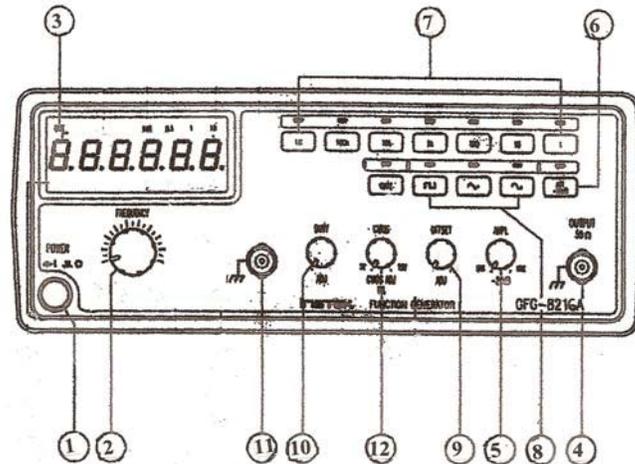


Figure A4 : Front panel of Function Generator

Description of front panel controls

1. Power Switch : Push 'in' to turn on and to 'out' to turn off.
2. Frequency Control knob: Turn clockwise to increase the frequency (~10:1ratio) Adjusting this control in conjunction with the ' Frequency-Range' control buttons(7), changes the output frequency. A small 'prefix'(m, k or M , standing for milli- , kilo- and mega-) appears at the bottom of the display(3), as an indicator of the frequency
3. Six-digit LED Display of the frequency (Hz, kHz or MHz)
4. Output Connector('BNC'): A BNC-to-Banana Jack Adapter is permanently affixed to this connector. The output impedance is 50Ω , resistive.
5. Output Amplitude Control knob : Adjusts voltage output to a maximum of $10V_{pp}$ [Pull 'out' for variable attenuation upto -20dB]
6. Attenuator Button : Provides an additional - 20 dB attenuation.
7. Frequency Range Buttons : Labelled 1, 10,100,1k, 10k, 100k, 1M , these buttons allow the selection of the frequency-range controlled by the frequency-control knob (2) [Total **Range is 0.3Hz to 3 MHz**]
8. Waveform Selector Buttons: Square, Triangular or Sine-wave output.
9. DC Offset Control knob: If pulled 'out', this knob can be adjusted to center the AC output over a DC 'bias' voltage (± 10 volts ,variable)Push the knob 'in' for **zero offset** , which is the normal position.
10. Duty Cycle/Symmetry Control: Adjusts the duty-cycle (or 'symmetry') of the Output waveform , when pulled 'out'.
- [11 & 12: TTL/CMOS output control and Output Connector: These are digital logic outputs and associated controls, not used in these labs]

Note: The BNC-to-Banana Jack adapter fixed to the FG output connector allows the standard banana-plug patch cords available in the lab to be used for making input and output connections.

OSCILLOSCOPE BASICS : The oscilloscope is an instrument which allows the display and measurement of time-varying voltage waveforms. Early oscilloscopes used the cathode-ray-tubes (CRTs) which were similar to television picture-tubes and utilized the deflection of an electron beam by the voltage of the signal to be displayed. The beam was controlled by X- deflection and Y-deflection circuits (which deflected the beam in the horizontal and vertical directions) and made to strike a phosphor-coated glass screen to produce a visible trace. The visible line-traces are seen on the screen due to the effect of phosphor coating as well as ‘vision persistence’ effects of the human eye. The basic ‘mechanics’ of waveform display may be simplistically explained as follows: In the absence of the applied voltage-input (applied to the Y-deflection), the electron beam is *repeatedly swept* (deflected) horizontally across the screen by a *sawtooth-shaped* voltage applied to the X-deflection circuit. This ‘sweep voltage’ ensures that the beam-spot (on the screen) traverses the screen, from left to right, in a *precise time period* but returns to the starting point on the screen in a negligibly short time. The precise and repeated X-deflections of the beam allow the *calibration* of the horizontal screen-trace in terms of a specific time, known as the time-base (or ‘Time per Division’) of the instrument. With the X-sweep operating, if the time-varying voltage signal to be displayed is simultaneously applied to the Y-deflection circuits, then the resulting trace on the screen will be that of the Y-signal, *scaled in accordance with both the operating time-base and the deflection-characteristics of the Y-deflection circuit*. If the voltage to be displayed is a repetitive signal (such as a sine-wave), a ‘stable’ (stationary) display can be produced by ‘synchronizing’ the start of the time-base(X-sweep) with the signal, a process known as ‘*triggering*’. With proper horizontal and vertical *calibrations* and *triggering* of the trace, the signal under consideration can be clearly displayed. Waveform measurements could be then made either using the directly-viewed trace or from photographs of it. The term ‘storage oscilloscope’ originally referred to a ‘scope in which a ‘long-persistent phosphor’ was used for the screen coating, enabling a rapid event on the trace to be viewed for a relatively long time. With the development of digital circuits this ‘storage’ capability has been more comprehensively achieved . The oscilloscope is an instrument which has undergone tremendous development since its invention. Modern oscilloscopes use liquid-crystal display (LCD) panels and are almost exclusively digital machines which have extensive automated measurement capabilities. The Tektronix TDS 320 DSO used in the present lab is such an instrument .

The Tektronix TDS 320 Digital Storage Oscilloscope (DSO) : This oscilloscope is a 2- Channel, 100 Mhz waveform display and measuring instrument which is highly automated. The DSO is menu-driven and all settings are made by pushing the front panel buttons and turning a few rotary knobs. Many common electrical signal measurements such as peak-to-peak and RMS amplitudes, frequency and period can be automatically measured and displayed along with the observed waveforms. A very useful feature is the availability of vertical(‘V-Bar’), horizontal (‘H-Bar’) and ‘paired’ cursors, which when positioned on waveforms using a control knob, allows horizontal intercepts (time-differences),vertical intercepts(voltage-differences) or both (‘paired’ cursors) to be measured and displayed. Another useful feature is the availability of a ‘Math’ function which displays the result of performing mathematical operations on the Channel 1(Ch1) and Channel 2(Ch2) voltage signals, such as addition, subtraction and multiplication. The cursors and the ‘Math’ function are used frequently in the laboratory procedures.

In the laboratory, records of test waveforms are obtained by printing the displayed (and stored) waveform on a laser printer. The DSOs at all the lab stations are connected via a digital switch to a common laser printer. Pushing the ‘Hardcopy’ button on the DSO results in the displayed screen [which must first be ‘frozen’(stored) by using the Run/Stop button] being

printed with station identification. The printer is programmed to produce two copies (for use by the two students at each station) when ‘Hardcopy’ is pressed,

The front panel of the TDS320 is shown in Figure A5

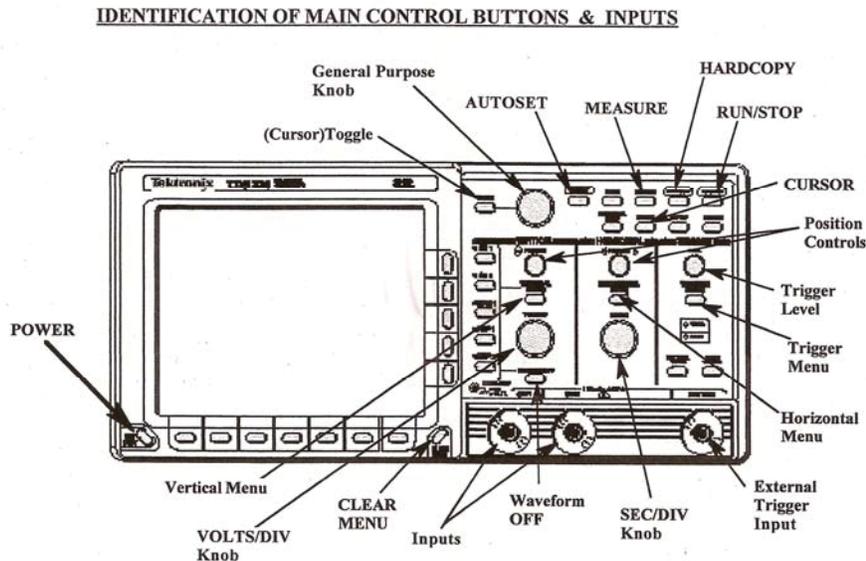


Figure A5: : Front panel of the TDS 320 DSO

Operation of the TDS 320 is mostly menu-driven. Three primary menu-buttons: VERTICAL MENU, HORIZONTAL MENU and TRIGGER MENU are located in the right half of the front panel. Pressing any of these, as well as the other buttons in the upper part of the panel, activates the 7 “Main Menu” buttons located at the bottom of the screen, and the 5 “Side Menu” buttons located immediately to the right of the screen. These two button-groups are then used for setting the DSO parameters.

INITIAL SET UP using AUTOSET:

1. Press ON/STDBY on the left of the screen. The DSO turns on and performs a self check and displays a message. Then press CLEAR MENU on the bottom-right corner of the screen. The graticule will now be displayed.
2. Press AUTOSET located at the top part of the panel. Pressing this button automatically selects the proper settings required for an *optimum* display of the signals applied to the input terminals. Pressing AUTOSET with no applied input, for example, displays the typical screen shown in Figure A4 which shows ‘noise’ voltages appearing across the BNC-to-Banana Jack adapter input terminals

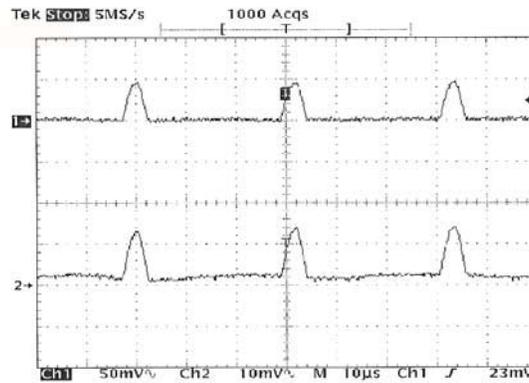


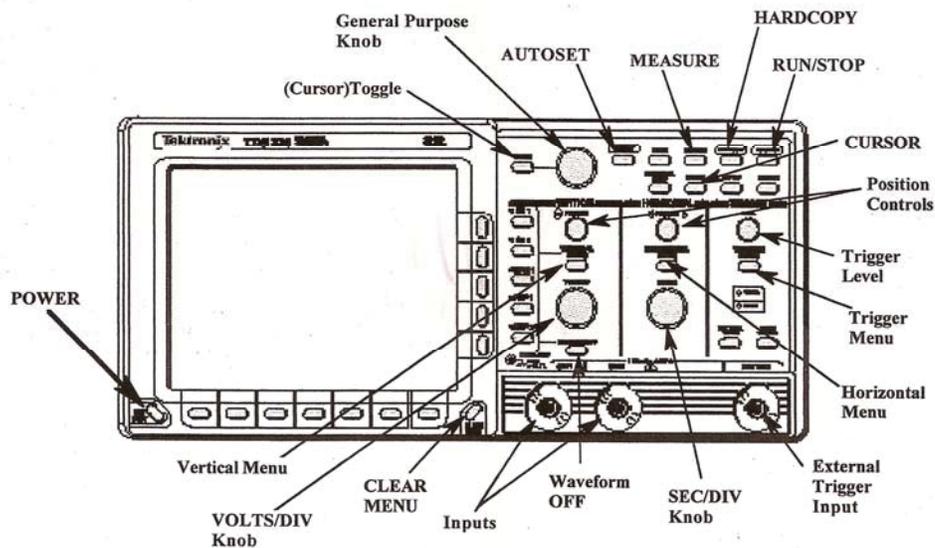
Figure A6: Typical screen display with no inputs applied

- If AUTOSET is pressed with a repetitive signal input (such as sinusoidal input) an optimum and stationary display consisting of several cycles of the waveform will result. The Channel 1(Ch1) and Channel 2(Ch2) sensitivities and the time- base (SEC/DIV) can then be changed using the rotary control knobs to satisfactorily display the input signal.

Note: The BNC-to-Banana Jack adapters fixed to the DSO input connectors (Ch1 & Ch2) allows the standard banana-plug patch cords available in the lab to be used for making input and output connections.

Brief descriptions of the front panel controls of the TDS320 digital storage oscilloscope follow.

IDENTIFICATION OF MAIN CONTROL BUTTONS & INPUTS



Front panel of the TDS 320 DSO [Reproduced for convenience]

FREQUENTLY USED OPERATIONAL BUTTONS & CONTROLS:

- CLEAR MENU :** Clears the menu from the screen when pressed. If used after selecting MEASUREMENT variables, pressing this button clears the menu but the measured values are moved to the right of the graticule
- Ch1,Ch2, Math, Ref1,Ref2 :** Pressing these buttons creates the appropriate trace on the screen. Ch1 & Ch2 are normally pressed to obtain the two channels. MATH creates a trace labelled 'M' which can be then configured

as Ch1+Ch2, Ch1-Ch2, Ch2-Ch1 or Ch1*Ch2 etc. The two Ref channels are intended for use in waveform comparisons

A small arrow indicator along the left edge of the graticule indicates the ZERO voltage position of each trace

WAVEFORM OFF : Pressing this button removes the **currently selected** waveform
POSITION (Vertical & Horizontal) : These controls located above the VOLT/DIV and SEC/DIV knobs respectively, are used to position the relevant traces.
VOLTS/DIV : Sets the vertical sensitivity of the selected channel (indicated at the bottom of the screen.)

SEC/DIV : Sets the time base (time per division) of the horizontal axis.

VERTICAL MENU : Pressing this button enables the setting of Input Coupling (DC, AC or Gnd), Bandwidth (Full or 20MHz), Channel Inversion, zero-ing of vertical position and vertical offset [The last named setting allows the *internal* addition of a variable DC signal to the input and is useful when observing small signals *with a large DC offset* (known as a “DC bias”).

For signals below 20 MHz, selecting the ‘20 MHz BW’ can yield a cleaner display in many cases since it ‘filters’ out ‘noise’ signals. ‘Fine Scale’ allows the vertical sensitivity to be set between standard pre-fixed values.

General Purpose (GP) Knob & Toggle Button : This knob, located at the top leftside and the associated TOGGLE button, is used to control many of the variable ‘side menu’ items. Together with the TOGGLE button it is also used for positioning the cursors for various voltage and time readouts [See ‘CURSOR’ next]

CURSOR : Pressing this button allows access to cursors, which are lines (or bars) which may be positioned using the abovementioned GP Knob and the TOGGLE button. The ‘active’ cursor is the ‘solid’ bar whose position can be set using the GP knob. The ‘dotted’ bar becomes the active cursor when the TOGGLE button is pressed. There are three types of cursors available : ‘H-Bars’, ‘V-Bars’ and ‘Paired’.

The V-bars are used to measure the time-interval (Δt) between cursors, while the H-bars are used for voltage-interval (ΔV) measurements. These two cursors can be used independent of the display. Paired cursors consist of both V and H cursors, acting simultaneously, with small H bars riding on the waveform. Thus, paired cursors are used to find the voltage-difference and the time-difference between any two points on a single waveform.

The general measurement procedure using cursors is as follows: Set the position of the active cursor using the GP knob, then press TOGGLE to activate the second cursor and then set its position. The intervals between the cursor positions appear on screen as small “ Δ ” values. Press the CLEAR button to move these readings to the righthand side of the graticule.

[Note: Cursor measurements can be visually recorded in the event of a Printer-system malfunction]

MEASURE : Pressing this button activates the main and side menus which are then used to make automated measurements such as Period, Frequency, Peak and RMS magnitudes, overshoots, etc. Measurements can be removed by using REMOVE MEASUREMENT from the main menu bottom row.

Buttons used for printout:

RUN/STOP & HARDCOPY : Pressing this button starts and stops signal acquisition .
The R/S button must always be pressed to ‘freeze’ the waveform before pressing the HARDCOPY button to obtain a printout.

OTHER BUTTONS (less frequently or not used normally) and their functions :

- HORIZONTAL MENU : Pressing this button subsequently allows setting of the timebase parameters (normally set to ‘Main Only’) and the Trigger position (normally set to 50% ,as can also be done by pressing the ‘SET LEVEL TO 50% under the TRIGGER MENU button)
- TRIGGER MENU : Pressing this button subsequently allows sweep or trigger parameters to be set. Normally, after pressing this button, the leftmost main-menu button is repeatedly pressed to get ‘Type Edge’ triggering [‘Video’ is not used in the lab].Under ‘Edge’ triggering, other parameters such as Trigger Source (Ch1,Ch2,EXT,AC line), Trigger Coupling (DC,AC) ,Slope and Level can be set.
- DISPLAY : Pressing this button subsequently allows screen such as screen Intensity, Graticule style (Full/Frame), Format (YT or XY) to be set
- ACQUIRE : Pressing this button allows setting of the signal acquisition mode (ie sample, peak-detect, single-sequence etc. Normally set to Mode >>Sample Stop after >>Run/Stop button only). To activate the Single Sequence mode, press ‘Stopafter’ >> ‘Single Acquisition Sequence’. The DSO then acquires a single sequence and displays it until the Run/Stop button is pressed. The R/S button then acts as a “single shot”button. The acquisition parameters (press ACQUIRE and set) must be set to Mode>> Sample, Stopafter >> Run/Stop button only.
- SAVE/RECALL SETUP : Pressing this button activates the Save/Recall setup menu under which , up to 10 front-panel stored setups (including Factory Setup) can be stored and recalled.
- UTILITY : THIS BUTTON IS TO BE USED FOR SYSTEM CONFIGURATION ONLY. Pressing this button brings up four pop-up menus: Config ,I/O, Cal and Diag., of which only I/O is of interest in normal operation, the others being used only for service procedures(Calibration & Diagnostics).Under I/O , the proper output port being used must be selected. (Serial,RS232 in the lab)

Proper settings for hardcopy in the lab should be as follows:

Hcp Port	set to >>> RS232C
Hcp Layout	set to >>> Landscape
Hcp Format	set to >>> Laserjet/Laser printer
GPIB	set to >>> Hardcopy(Talk only)
RS232C config.	set to >>> Baud Rate to 19200
Hard Flagging	set to "ON"
Soft Flagging	set to 'OFF'
EOL	set to LF
Parity	set to 'None'
Stop Bits	set to '1'
Delay	set to 0 sec , using the GP knob.

=====

APPENDIX – B

OPERATIONAL AMPLIFIER BASICS & DESCRIPTION OF THE OP-AMP MODULES USED IN THE LABORATORY

The Operational Amplifier (abbreviated ‘op-amp’ or OA) is a five-terminal electronic device which basically functions as a dependent source. The op-amp derives its name from the fact that it can be used in circuits that perform various mathematical operations such as addition & subtraction, multiplication & division, integration, differentiation etc. Schematic diagrams of the ‘industry-standard’ integrated circuit amplifiers $\mu A741$ and CA3140 , both of which are used in the lab, are shown in Figures B1(a) & B1(b)

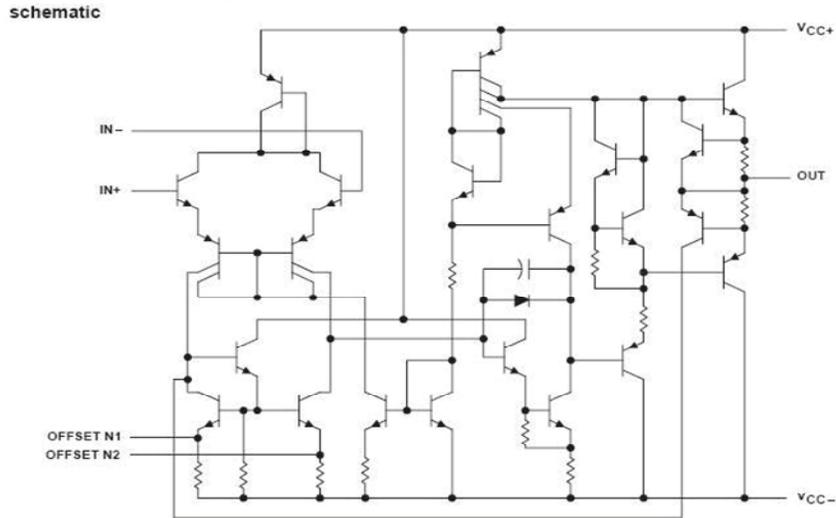


Figure B1(a) : Internal Circuit of ‘741’ op-amp

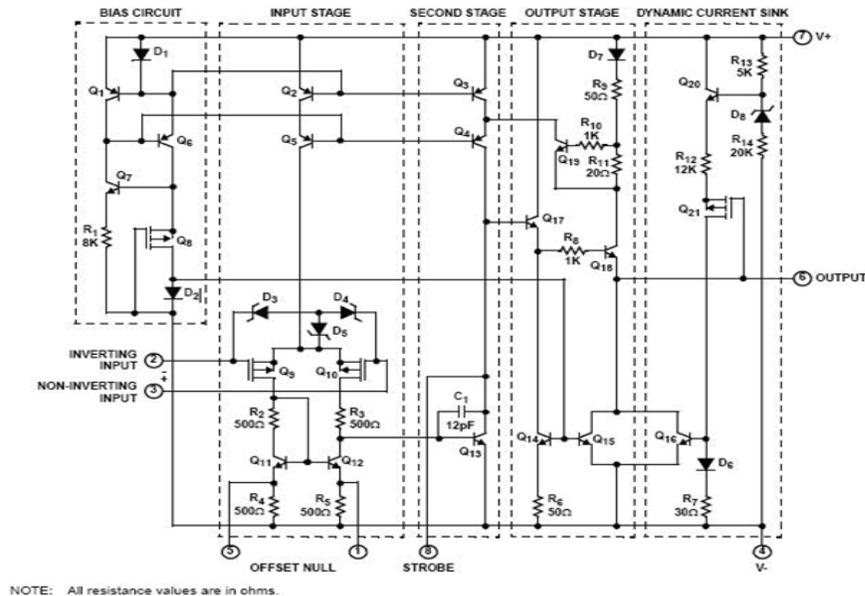


Figure B1(b) : Internal Circuit of ‘CA3140’ op-amp

The general equivalent circuit of an OA, much simplified, is shown in Figure B2(a).

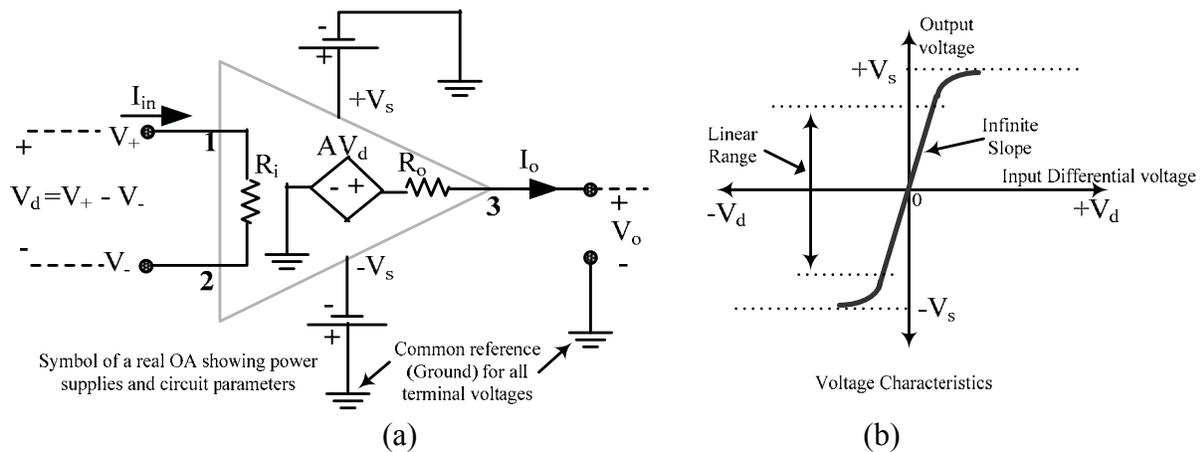


Figure B2

The three main terminals seen are named as follows: Terminal 1 (marked “+”) is called the ‘Non-Inverting’ input terminal, terminal 2 (marked “-”) is called the ‘Inverting’ input terminal and terminal 3 is called the ‘Output’ terminal. The remaining two terminals are power input terminals connected to balanced, bipolar power supplies $+V_s$ and $-V_s$, which share a common ground to which all the other terminal node voltages are referred. [The circuits in Figure B1 also show ‘offset null’ terminals which can be used to ‘balance’ the output voltage to zero, when there is no input applied.]

The output terminal voltage V_o is linearly related the input voltage differential $V_d = [V_+ - V_-]$ over a limited range which lies between the values of the power-supply voltages $+V_s$ and $-V_s$, as shown in Figure B1(b). The slope $A = \Delta V_o / \Delta V_d$ is called the ‘open-loop’ voltage gain A and its value in the linear range is extremely large, approaching infinity. The gain A appears as the controlling parameter of a voltage-controlled voltage source (VCVS) in the equivalent circuit shown within the triangular outline.

The resistance R_i is called the ‘input resistance’ and R_o is called the ‘output resistance’. The parameters A , R_i and R_o have typical values of $10^5 \sim 10^8$ volt/volt, $10^5 \sim 10^{13} \Omega$ and $10 \sim 100 \Omega$, respectively, and typical power supply voltages range from 3 to 24 volts. For an op-amp to represent an ideal VCVS, the conditions $R_o = 0$ and $R_i \rightarrow \infty$ are required and the second of these conditions requires that $A \rightarrow \infty$. Modern op-amps approach these ideal conditions, thereby making it possible to consider an ideal op-amp when analyzing circuits containing these amplifiers. [Typically, for the $\mu A741$ op-amp, $I_+ (I_-) \approx 500 \text{ nA}$, $A \approx 10^5 \text{ v/v}$, $R_i \approx 2 \text{ M}\Omega$, $R_o \approx 75 \Omega$. For the CA3140 op-amp, while A and R_o have values similar to those of the 741, the input currents are $\approx 10 \text{ pA}$, and $R_i \approx 1.5 \text{ T}\Omega$, showing characteristics that are closer to those of an ideal op-amp]

The symbol used for the ideal op-amp, shown in Figure B3, has three terminals associated with the node voltages V_- , V_+ , and V_o .

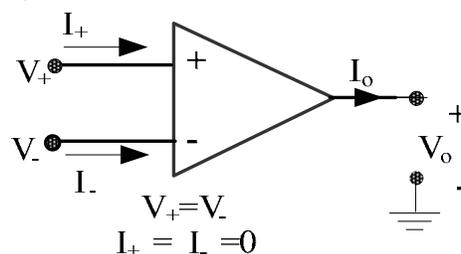


Figure B3

The ideal OA symbol does not show the two power supply terminals and it is understood that the output voltage range is limited by the supply voltages $\pm V_s$. The ideal OA is characterized by two important constraints : (1) the current drawn by the input terminals is zero, ie $I_+ = I_- = 0$ (which is equivalent to stating that the amplifier has an input resistance of infinity) and (2) the voltage difference between the two input terminals is zero, or $V_d = V_+ - V_- = 0$ (which is equivalent to stating that the node voltages at the two input terminals have the same value although the two input terminals are not physically connected to each other). The above two constraints are also called the ‘Virtual Open-Circuit’ (or VOC) and the ‘Virtual Short-Circuit’ (or VSC) principles, respectively. By the VSC principle, if one of the input terminals is connected to ground, the other input terminal is also at ‘zero node voltage’ and is called a ‘virtual ground’.

All linear OA circuits must contain an external “feedback” path **between the output terminal and the inverting input terminal** which provides a ‘negative feedback loop’, which is necessary for *stable* operation of the circuit. Normally, an OA circuit forms a ‘closed loop’, *negative-feedback system*, whose voltage-gain is finite. The external circuitry can be of various kinds, thus allowing a wide variety of “op-amp circuits” which perform different operations. In general, *all* OA circuits can be analyzed by the simple application of the VOC and VSC principles mentioned above together with the application of nodal analysis at the input and other terminals, but not at the output terminal. This is because the zero input currents can be accounted for, but the output current I_o contains the power supply currents which are not considered in the ideal OA and are hence unknown. However, once V_o is determined, the output current I_o may be found using using KCL.

In the laboratory, the OA is available along with associated components, in two types of experimental rigs (boxes) whose faceplates are shown in Figure B4. In both rigs, the op-amp IC is socketed to allow replacement of defective ICs. The rig at the top (with a blue-coloured panel) is used with the CA3140 op-amp, and for purposes of identification will be called the “3140 box”. This box contains resistors with nominal values of 20 k Ω , 50 k Ω and 100 k Ω wired as shown with one 100 k Ω pre-wired as a feedback resistance.

The “741 Box” (with a greenish beige-coloured panel), shown at the bottom of the figure uses the μ A741 op-amp and has the following components mounted on it as shown: Two 1k Ω resistors, one 10 k Ω resistor, one 0.1 μ F capacitor, two 9.1-volt Zener diodes connected back-to-back, and a variable resistance of 6 k Ω . The 6 k Ω resistance is provided (at the two terminals immediately above the dial) as a fixed 1 k Ω in series with a 5 k Ω , 10-turn potentiometer fitted with a turns-counting dial [ie: The potentiometer resistance is thus 0.5 k Ω /turn. Thus, with the dial set at zero, the resistance is 1 k Ω and with the dial set at 3 turns (for example), then $R = 1000 + 3(500) = 2500 \Omega$]. The “741 Box” has no pre-wired feedback resistor and one of the on-board resistors must be connected (using a short patch cord) to provide the negative feedback path required for stable operation. In both boxes, various OA circuits can be built by making the appropriate connections, using the short (4” or 8” length) banana plug patch cords available in the lab.

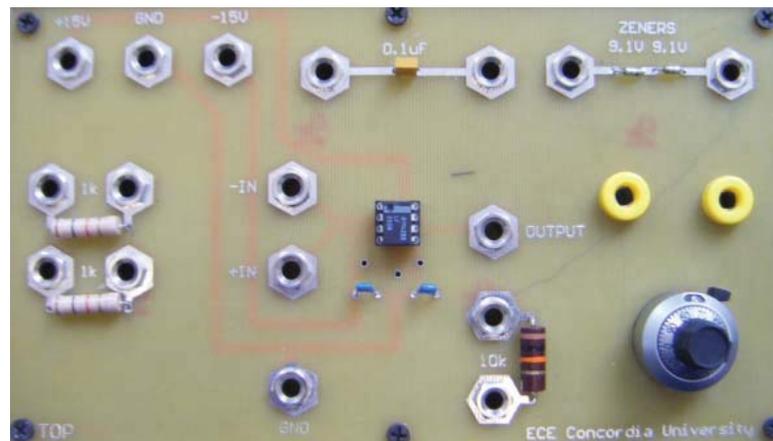
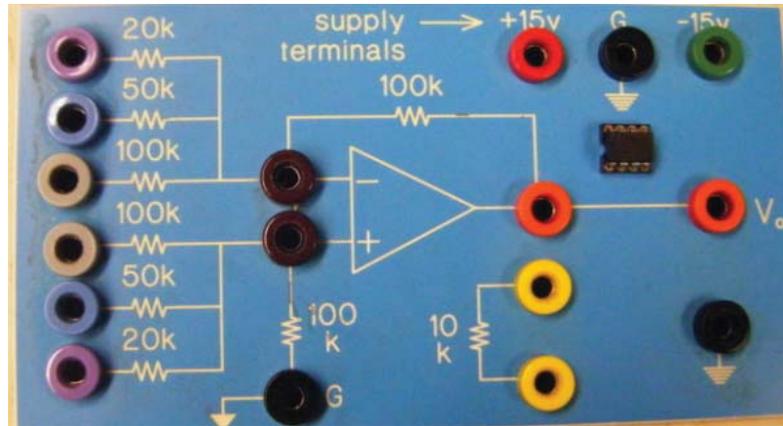


Figure B4

Both boxes have +15 / 0 /-15 volt supply voltage terminals which must be connected to the op-amp supply output terminals found at the extreme left on the PSP panel. In the op-amp experiments of Expt #3, the “3140 Box” and the “741 Box” will be used for DC and AC circuits, respectively.

Checking the opamp boxes before starting the experiments will be very helpful in case troubleshooting becomes necessary. With the PSP turned off, connect the +15/0/-15 supply terminals to the op-amp supply terminals on the PSP, **taking care to observe proper polarity**. To check the boxes, first connect the V_+ input to ground. Next provide a negative feedback path by connecting the output terminal to the V_- terminal directly thereby configuring the OA as a ‘voltage follower’ [see Figure 3.4(d)]. Then turn on the PSP. If the OA is properly functioning, the output voltage should be zero; if not, the IC must be replaced.



APPENDIX – C

INTRODUCTION TO COMPLEX NUMBERS, PHASORS & IMPEDANCE

Complex Numbers:

The quadratic equation $ax^2 + bx + c = 0$ is frequently encountered in engineering analysis.

The solution of this equation is : $x = -\frac{b}{2a} \pm \frac{\sqrt{b^2 - 4ac}}{2a}$. When $b^2 < 4ac$, the second term in

the solution is the square-root of a negative number, which is not defined in the system of real numbers. This situation is dealt with by introducing ‘imaginary numbers’ jN , where j is defined as $j = \sqrt{-1}$ and N is a real number. Thus, for example, a square-root such as $\sqrt{-36}$ can be expressed as the imaginary number $j6$. A complex number is defined by the quantity $(A + jB)$, where A and B are real numbers (positive or negative). The number $(A+jB)$ can be plotted as a point P in the *complex plane*, whose abscissa is the set of positive and negative real numbers (A) and whose ordinate is the set of of *imaginary* numbers (jB), as shown in Figure C1. If $P = A+jB$, then $P^* = A-jB$ is called the *complex-conjugate* of P

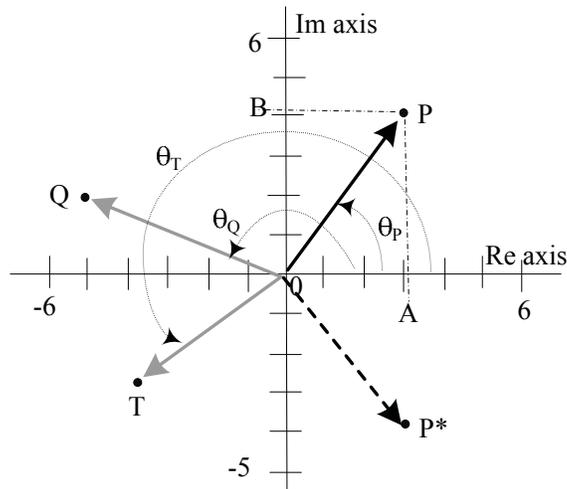


Figure C1

The number $P=A+jB$ can be also expressed in the ‘polar form’ $R \angle \theta$, where $R = \sqrt{A^2 + B^2}$ and $\theta = \tan^{-1}(B/A)$, are called the magnitude and phase respectively. [The relation $\theta = \tan^{-1}(B/A)$ is valid only when P is in the right-half of the complex plane (ie A positive, B positive or negative). When P is in the left-half of the complex plane, the relation must be changed to $[180^\circ + \theta]$ in keeping with the polar co-ordinate angle-measurement convention]. A third representation is the ‘complex exponential form’ $R e^{j\theta}$, which follows from Euler’s theorem:

$$R e^{\pm j\theta} = R \cos\theta \pm j R \sin\theta.$$

Complex numbers follow all algebraic rules such as addition, subtraction, multiplication, division etc. It is useful to remember the following facts about operations in the complex plane:

(a) Since $j = \sqrt{-1}$, $j^2 = -1$, $j^3 = jj^2 = -j$, $j^4 = j^2j^2 = +1$ and every multiplication by j is equivalent to a counter-clockwise rotation of the complex-number vector by 90° and every multiplication by $-j$ (or division by j) is equivalent to a clockwise rotation of the vector by 90°

(b) Additions & Subtractions are more easily done with the numbers in rectangular form:

Eg: $(3+j4) + (5+j6) + (-7-j8) = (3+5-7) + j(4+6-8) = 1+j2 = 2.24 \angle 63.4^\circ$.

(c) Multiplications & Divisions are more easily done with the numbers in polar form:

$$(R_1 \angle \theta_1) \cdot (R_2 \angle \theta_2) / R_3 \angle \theta_3 = R_1 e^{j\theta_1} R_2 e^{j\theta_2} / R_3 e^{j\theta_3} = (R_1 R_2 / R_3) e^{j(\theta_1 + \theta_2 - \theta_3)}$$

$$= (R_1 R_2 / R_3) \angle (\theta_1 + \theta_2 - \theta_3)$$

Eg: $[20 \angle 42^\circ] [5 \angle 30^\circ] / [40 \angle 27^\circ] = [(20)(5)/40] \angle (42+30-27)^\circ = 2.5 \angle 45^\circ = 1.77 + j1.77$

(d) 'Normalization': A complex number in 'reciprocal' form $1/(A+jB)$ can be converted to the non-reciprocal form $C + jD$ by multiplying and dividing by the complex-conjugate of the denominator, i.e. $\{1/(A+jB)\} \cdot [(A-jB)/(A-jB)] = (A-jB) / [A^2 + B^2]$ since $(A+jB)(A-jB) = [A^2 + B^2]$. The result can be written as $C + jD$ where $C = A / [A^2 + B^2]$ and $D = -B / [A^2 + B^2]$

Eg: $1/(3+j4) = (3-j4)/(9+16) = (3/25) - j(4/25) = 0.12 - j0.16 = 0.2 \angle -53.1^\circ$.

Knowledge of complex numbers and their manipulations is very important in AC circuit analysis because the quantities known as 'phasors' and 'impedances' which are used there, are complex numbers.

Phasors : The sinusoidal responses in AC steady state circuit analysis are characterized by the magnitudes and phase-angles (or simply, phase), which are also the quantities that define a complex number. The various sinusoidal voltages and currents in a circuit can be considered as vectors of the general form $F e^{j(\omega t + \theta)}$ rotating counterclockwise in the complex plane, having different magnitudes (F) and maintaining fixed phase (θ) differences between them. The vectors rotate at a constant angular velocity (ω) corresponding to the frequency (f) of alternation and $\omega (= 2\pi f)$ is called the *radian* (or *angular*) *frequency*. If the rotation is ignored, then the various voltage and current vectors only indicate their magnitudes and the phase-angle inter-relations and these *stationary* vectors $[F \angle \theta]$ are called phasors. If the various voltage and current phasors in a circuit are drawn in the complex plane, the resulting diagram, called a phasor diagram, shows the magnitudes and relative phase shifts of those voltages and currents in the sinusoidal steady state. Phasors are simply representations of sinusoidal signals on the complex plane.

The solution of the integro-differential equations which result from the analysis of circuits [containing resistances as well as the storage-elements L and C] are considerably simplified when the sinusoidal excitation and responses are expressed in complex exponential form [i.e. by using voltages such as $V e^{j(\omega t + \theta)}$ and currents such as $I e^{j(\omega t + \phi)}$]. This simplification occurs because using such an excitation results in the conversion of the integro-differential equations into linear algebraic equations whose variables are the aforementioned voltage or current phasors. Solution of these linear equations will yield the current and voltage phasors, which are directly related to the sinusoidal steady state responses in the time-domain. In AC circuit analysis, the zero-phase cosine function is taken as the reference by convention. Hence, if a voltage or current is given as a sine function $F \sin(\omega t + \phi_1)$, it is first changed into the equivalent cosine function $F \cos(\omega t + \phi_1 - 90^\circ)$ before analysis. By the same convention, a

phasor $F \angle \phi$ is converted back to the equivalent time-domain function $F \cos(\omega t + \phi)$, where ω is the specified radian frequency. The conversion of the integro-differential equations into linear algebraic equations referred to above, also leads to the definition of a generalized quantity called the 'impedance', which describes the current-voltage characteristics of the passive elements R, L and C in the phasor domain.

Impedance: If the voltage and current associated with a passive element (R, L or C) are expressed as complex exponentials, given by $v(t) = V e^{j(\omega t + \theta_v)}$ and $i(t) = I e^{j(\omega t + \theta_i)}$ then the impedance Z is defined as the voltage -to-current ratio $v(t)/i(t)$:

$$\begin{aligned} \text{ie } Z &= v(t) / i(t) = V e^{j(\omega t + \theta_v)} / I e^{j(\omega t + \theta_i)} = V e^{j\theta_v} / I e^{j\theta_i} \\ &= V \angle \theta_v / I \angle \theta_i = (V/I) \angle (\theta_v - \theta_i) = |Z| \angle \theta \end{aligned}$$

where the impedance magnitude $|Z|$ is the phasor-magnitude ratio V/I , and the impedance angle $\theta = (\theta_v - \theta_i)$, is the angle by which the voltage-phasor V leads the current-phasor I.

It is seen that the impedance Z of an element is a complex quantity given $|Z| \angle \theta$. The unit of Z is the Ohm and the symbol used for Z is the same as that used for R. The reciprocal of impedance $1/Z$ is called the admittance Y, and it is the phasor analogue of conductance G, just as Z is the phasor analogue of resistance R.

Impedance of R, L and C elements: Application of a complex exponential excitation to the respective current-voltage elemental relations, namely, $v(t) = R i(t)$, $v(t) = L di(t)/dt$ and $i(t) = C dv(t)/dt$, leads to expressions for the corresponding impedances of these elements, respectively given by

$$Z_R = R \qquad Z_L = j\omega L = \omega L \angle 90^\circ \qquad Z_C = 1/j\omega C = -j/\omega C = (1/\omega C) \angle -90^\circ$$

It is seen that while Z_R is a real number (=R), Z_L and Z_C are both frequency-dependent imaginary numbers, which are composed of reactances $X_L (= \omega L)$ and $X_C (= -1/\omega C)$. The phasor diagrams corresponding to the three elements are shown in the top half of Figure C2.

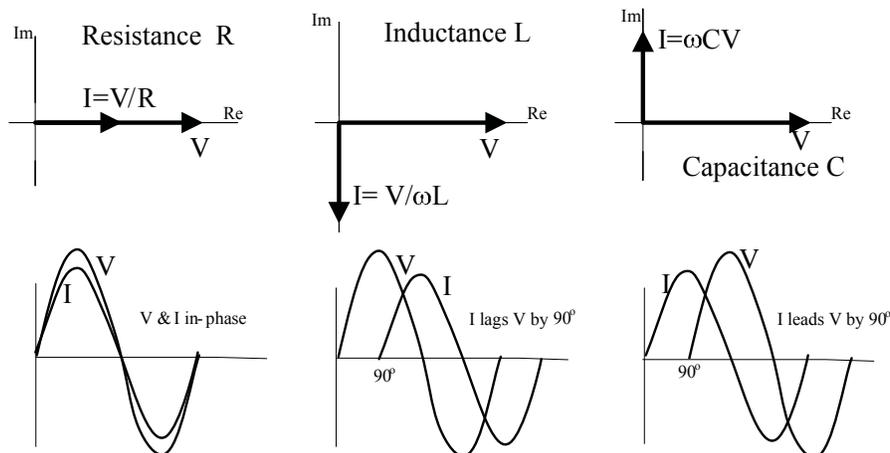


Figure C2: Phasor diagrams (& waveform) for R, L and C elements.

It can be seen that the voltage and current phasors are 'in-phase' with each other for the R element, whereas for L, I lags V by 90° and for the C element, I leads V by 90° . The corresponding sinusoidal waveforms are shown in the bottom half of Figure C2. Since impedances are complex quantities, they can be also shown in 'impedance diagrams' which

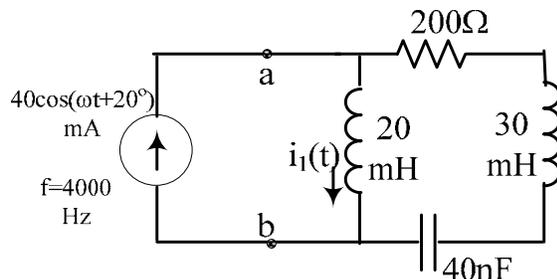
are similar to phasor diagrams, but illustrate the vector-relations between various impedances.

Since Z is defined by a V/I ratio (as in Ohm's law for R), impedances follow the same combination rules as resistors. The impedance of a two-terminal network can be defined as $Z = R \pm jX$ where $+jX = +j\omega L$ for an 'inductive' impedance and $-j/\omega C = -jX$ for a 'capacitive' impedance.

A **phasor circuit** [or *frequency-domain circuit*] is a steady state AC circuit in which all voltages and currents are represented by phasors and all passive circuit elements are replaced by their impedances. Since phasors are vectors, the addition (and subtraction) operations inherent in the application of KCL and KVL are **vector** additions (and subtractions) in phasor circuits. Further, since the definition of Z is analogous to that of R , all the laws and procedures used with resistive circuits [such as combinatory rules, voltage & current division rules & analysis techniques such as Nodal and Mesh analysis, Thevenin's theorem etc] are applicable to phasor circuits as well, the only difference being that the quantities involved are now complex numbers. In a phasor circuit, the various voltage and currents are frequency-dependent because the impedances in the circuit are frequency-dependent. The variation of any given voltage or current variable as a function of frequency is called the 'frequency response' of that variable.

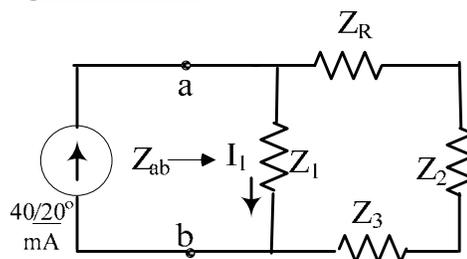
Resonance: Because of the possibility of obtaining positive and negative reactances, a combination of L and C elements may result in an inductive, capacitive or even a zero reactance at a given frequency. A 'resonance' condition is said to occur when the reactive part of an impedance becomes zero [ie $\text{Im } Z = 0$] due to reactance cancellation and the associated frequency is called a 'resonant frequency'. A problem example illustrating the phasor circuit and impedance concepts is given below.

Example : Consider the circuit shown below



- Find the equivalent impedance Z_{ab} 'seen' by the source .
- Determine the phasor voltage V_{ab}
- Determine the current $i_1(t)$

Solution: The equivalent phasor circuit is shown below.



At $f = 4000$ Hz, $\omega = 2\pi(4000) = 6.283(4000) = 25132$ rad/s.

(a) The impedances in the two parallel branches connected to the source are :

$$Z_1 = j\omega(0.02) = j 502.64 \text{ Ohms} = 502.64 \angle 90^\circ$$

$$\begin{aligned} Z_{RLC} &= Z_R + Z_2 + Z_3 = 200 + j\omega(0.03) + 1/j\omega(40 \times 10^{-9}) \\ &= 200 + j753.96 - j 994.75 = 200 - j 240.79 = 313.02 \angle -50.3^\circ \end{aligned}$$

The impedances Z_{ab} seen by the source is given by $(Z_1) \parallel (Z_{RLC})$

ie

$$\begin{aligned} Z_{ab} &= [j 502.64] [200 - j 240.79] / \{ j 502.64 + 200 - j 240.79 \} \\ &= [j 502.64] [200 - j 240.79] / \{ 200 + j 261.85 \} \\ &= [502.64 \angle 90^\circ] [313.02 \angle -50.3^\circ] / \{ 329.49 \angle 52.63^\circ \} \\ &= 477.52 \angle -12.93^\circ \text{ Ohms} (= 465.41 - j 106.85) \end{aligned}$$

(b) Voltage $V_{ab} = I_{\text{source}} Z_{ab} = 0.04 \angle 20^\circ (477.52 \angle -12.93^\circ) = \underline{19.1} \angle \underline{7.07^\circ}$ volts

(c) By current division, $I_1 = I_{\text{source}} [Z_{RLC} / (Z_1 + Z_{RLC})]$

$$= 0.04 \angle 20^\circ (313.02 \angle -50.3^\circ) / \{ 329.49 \angle 52.63^\circ \}$$

$$I_1 = 0.038 \angle -82.93^\circ \text{ A}$$

Hence, $i_1(t) = \underline{38 \cos(25132 t - 82.93^\circ)}$, mA

BLANK

APPENDIX – D

DESCRIPTIONS OF EQUIPMENT USED IN THE AC POWER EXPERIMENT

The equipment used in the ‘Power’ experiment (Expt#5) consists of the following units :

1. The Power Variac (0 to 120 volt Variable AC Power source)
2. The RL-load unit .
3. The Power-Capacitor unit
4. Two Analog Wattmeters. [Simpson Model 1379]
5. The Fluke 8010A & Agilent 34405A DMMs (described in Appendix A)

This appendix gives brief descriptions of items # 1 to # 4 above. The instruments in Item #5 are described in Appendix-A.

The Power Variac [Gray metallic unit with a large black knob, bolted on top of the left end of the hutch at each workstation.] is a “3-phase” AC power unit consisting of three single-phase, variable auto-transformers (called **variacs**), which are internally connected in a “Y- configuration” with a common ground terminal.).The movable arms of the autotransformers are mechanically coupled so that the wiper positions are identical on all three auto-transformers for any given control-knob position. The variac is plugged into a **3-phase** AC supply available in the lab.

The 3-phase variac is protected by three 3A time-delay fuses (which are not the standard AGX fuses used in the other lab modules). The unit is fitted with a Fuse-Status-Indicator (FSI) box which allows the fuses to be checked quickly: With nothing connected to the terminals and with the control knob turned fully clockwise, all three amber lamps on the FSI box should glow when the three fuses are good. Photographs of the power variac without the FSI box and with the FSI box mounted are shown in Figure D1.



VARIAC , SHOWING
FUSES [WITHOUT FUSE-
STATUS INDICATOR BOX]



VARIAC, [WITH FUSE-STATUS
INDICATOR BOX MOUNTED]

Figure D1: The 3-Phase Power Variac

When the variac control knob is turned clockwise to its maximum, a nominal voltage of 120 volts RMS is available from each of the output terminals Red, Yellow & Blue, with respect to the Green terminal (Ground G). The three outputs have a sequentially symmetrical 120° phase-difference between them. Since only single-phase power is involved in Expt#5, any one of the outputs R, Y or B may be used, R being the preferred output terminal. The variac has no ON/OFF switch and its knob must always be returned to the zero position for safety.

The RL-load unit, shown in Figure D2 contains a 0.2 H power inductor and a 100Ω , 150W variable power resistor (rheostat) mounted in a metal cabinet. The R element is provided with a control knob and graduated scale. The resistance value increases as the knob is turned clockwise. The rheostat is protected by a 1 Amp fuse [Type AGX1] which is accessible from the front. The fuse status can be checked by measuring the resistance across the R terminals. For safety reasons, the R control should always be set to its midpoint when not in use. When set at its minimum and maximum, the wiper of the rheostat sometimes makes intermittent contact; hence it is advisable not to set the knob at its extreme positions. The actual resistance at the terminals should always be calibrated against the dial knob setting if exact R values must be known.

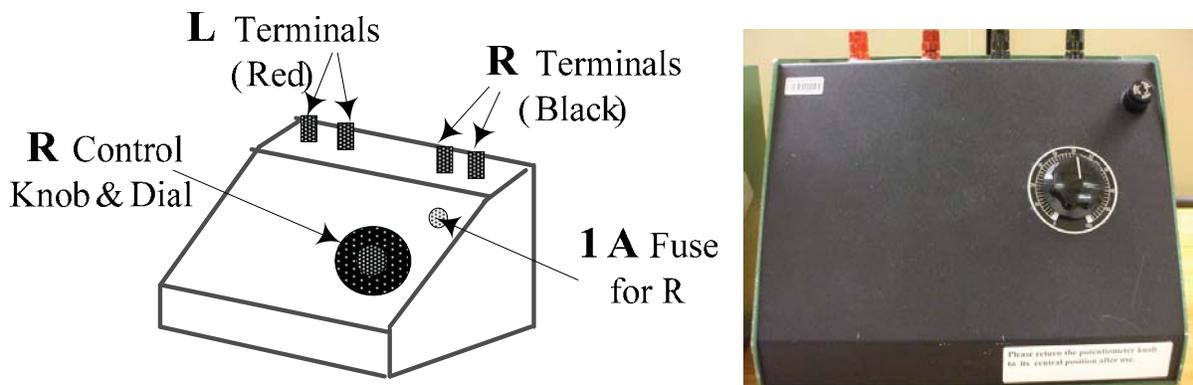


Figure D2: The RL load unit

The power capacitor (C) unit, shown in Figure D3, contains seven* power capacitors of values $1\mu\text{F}$, $2\mu\text{F}$, $2\mu\text{F}$, $5\mu\text{F}$, $10\mu\text{F}$, $10\mu\text{F}$ & $10\mu\text{F}$ which can be sequentially connected in parallel using switches, as shown. [*Some units may have 6 capacitors with values 1, 2, 2, 5, 10 & $20\mu\text{F}$]. Each capacitor is connected when a switch is toggled upwards. By sequentially toggling switches from right to left, terminal C values up to $40\mu\text{F}$ in $1\mu\text{F}$ increments can be obtained.

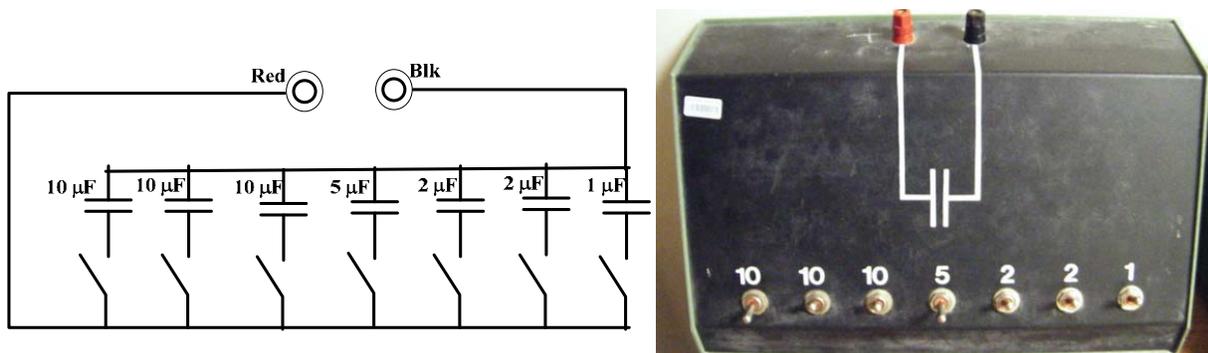


Figure D3: The Power Capacitor unit

The Wattmeter is a moving-coil meter which has two coils, a low resistance ‘current coil’ and a high resistance ‘voltage coil’ [see Figure D4]. Each coil is connected to a pair of terminals. To measure the average power P delivered to a load, the ‘current coil’ is connected in series with the load so that the load current $I(t)$ flows through it and the ‘voltage coil’ terminals are connected across the load so that the load voltage $V(t)$ appears across it. The meter produces a pointer-deflection which is calibrated in terms of average power P ($=VI \cos \theta$). Note that the ‘live’ end of the ‘voltage coil’ (Red terminal) may be connected either to the load-side of the ‘current coil’ or to the line-side (as in Figure D4). Either of these connections may be used assuming that the impedance of the current coil is negligible. At any instant, if the current I is entering the Red terminal of the ‘current coil’ while the voltage at the Red terminal of the ‘voltage coil’ is positive, then power is being absorbed by the load and this results in an upscale (positive) meter indication.

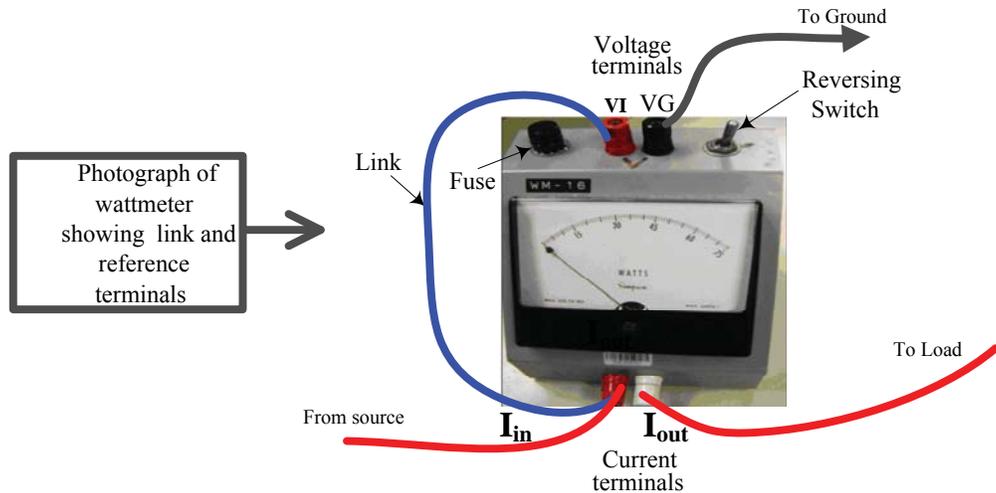
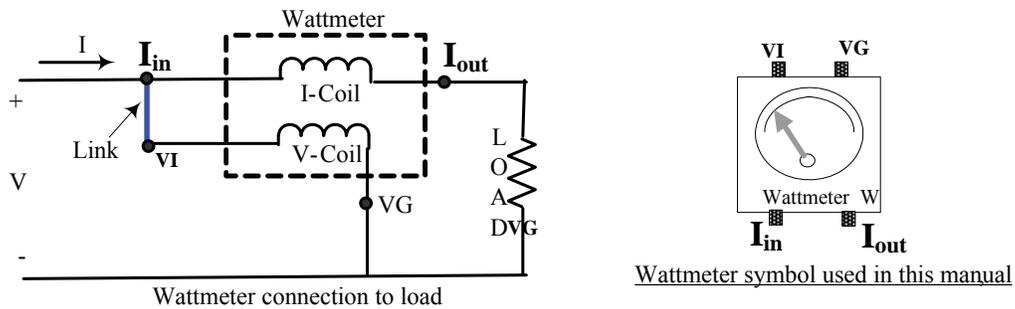


Figure D4: The Wattmeter and its connections

If the connections to either coil is reversed, or depending upon the phase relation between V and I , the meter pointer can swing downscale resulting in the pointer straining against the zero-stop. If allowed to continue, this will damage the pointer deflection mechanism. In order to avoid damage, the terminals connected to the ‘voltage coil’ are internally reversible by means of a ‘reversing switch’ which is provided in order to allow wattmeter readings which tend to go negative. The switch should be actuated immediately if it is observed that the pointer is straining against the zero-stop.

Ideally, the ‘current coil’ should have zero-impedance and no voltage across it and the ‘voltage coil’ should draw no current and hence the wattmeter itself should not draw any power. The wattmeter has a protective fuse which must be replaced if necessary. **The full scale (FS) range of the of the Simpson Model 1379 Wattmeter used in the lab is 75 Watts (max. current 1 Amp) and its accuracy is $\pm 2\%$ of FS.** The wattmeter unit is provided with a 1A fuse (see Figure D4) . [Use **AGX 1** fuses]

Simplified Wattmeter connections :

The connections to a wattmeter can be simplified by first ‘linking’ the Red terminal of the current coil to the Red terminal of the voltage coil and label this as the “current in”(I_{in}) terminal .with this link in place, only three connections [I_{in}, I_{out} and voltmeter ground VG] need be made to the rest of the circuit.

=====

APPENDIX – E

A BRIEF NOTE ABOUT SPICE SIMULATION

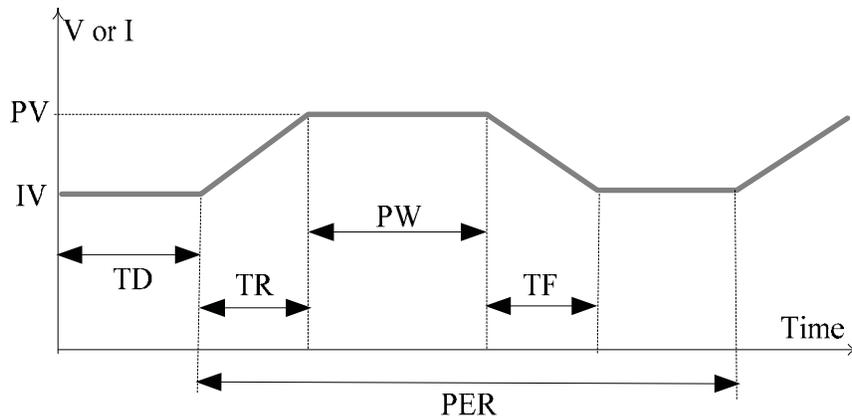
The following is a short guide to introduce the general methodology and terms used in ‘schematic capture’ Spice programs.

(a) Circuit construction: Various components are available under the ‘Devices’ menu or a device ‘library’. A circuit is usually constructed by first selecting and dragging the required device(s) to the workspace. Menu-driven operations allow orientation of the devices, setting of the device properties etc. This includes component name, value and, in the case of storage elements, initial conditions. In order to minimize the number of components, it is often preferable to use the ‘initial condition’ on a storage element rather than representing the initial condition by a source. Connections between the components are made by using a ‘connection tool’. The number of components is usually limited in demo or trial programs. Spice is based on nodal analysis and a circuit ‘Ground’ must be provided. The ground counts as a component. The output required is specified as a metering device such as a voltmeter or an ammeter or as a ‘probe’ tool.

(b) Simulation Types: The type of simulation is determined by the type of independent sources selected. Spice provides for DC, Pulse, Exponential, Piecewise-linear as well as Sinusoidal sources. Transient analyses, for example require pulse sources and frequency-response analyses require sinusoidal sources. Each of these sources is also associated with ‘set up’ instructions.

‘DC Sweep’ Analysis is used with DC sources and it allows the variation of the voltage or current of any one of those sources over a specified range. The source ID and the ‘sweep’ range [start, step and stop values] must be specified. The ‘step’ value is the voltage or current increment in the sweep. It should be short enough to define any sharp changes occurring in the response.

‘Transient’ or ‘Pulse’ analysis: The basic pulse waveform used is the repetitive trapezoidal waveform shown in Figure E1.



PV = Pulsed Value, IV = Initial Value

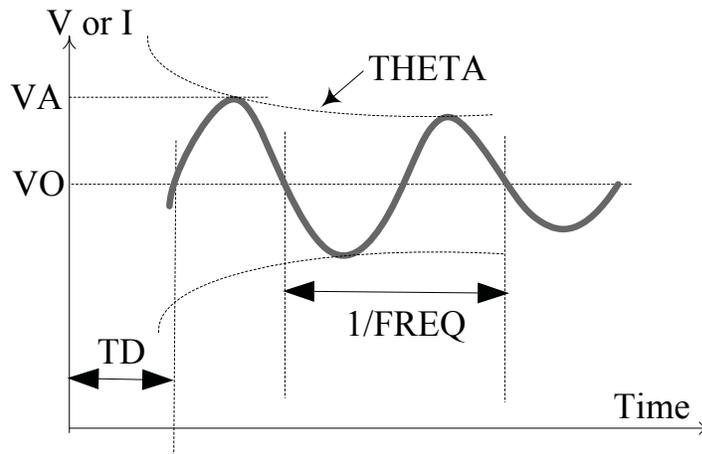
TD= Delay Time, TR=Rise Time, TF= Fall Time

PW =Pulse Width, PER= Pulse Period

Figure E1: Basic Pulse waveform used in Spice

By specifying zero and/or non-zero values for the variables shown, the waveform can be configured into repetitive 'pulse' functions of various types.

For example, setting $IV = TR = TF = TD = 0$, $PV = 5$ volts, $PER = 2PW = 100$ nanoseconds will result in a 0 to 5 volt, 10 MHz square-wave. Other pulse waveforms may similarly be generated. 'AC Sweep' analysis is used with AC sources and the swept parameter is the frequency. Thus the output data yields the frequency response, for magnitude (not the Magnitude Ratio !) and phase of the output. The basic sinusoidal source waveform is defined as an exponentially varying waveform shown in Figure E2. This waveform can also be modified as desired by specifying the variables shown.



VO = Offset, VA = Amplitude
 FREQ = Frequency, TD = Delay Time
 THETA = Damping Factor (default : Zero)

Figure E2 : Basic Sinusoidal waveform in Spice

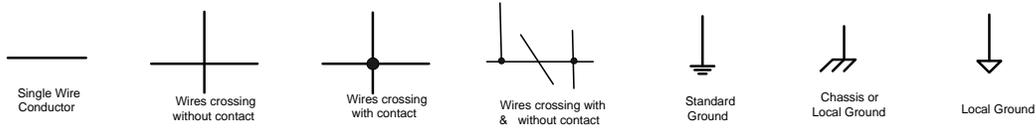
For example, with $THETA = 0$ (by default), $TD = VO = 0$, $VA = 10$ volts, and $FREQ = 1200$ Hz will produce a zero-biased, 10 volt Peak, 1.2 kHz waveform. A sine-wave 'riding' on a DC signal can be obtained by using a non-zero value for VO, etc.

=====

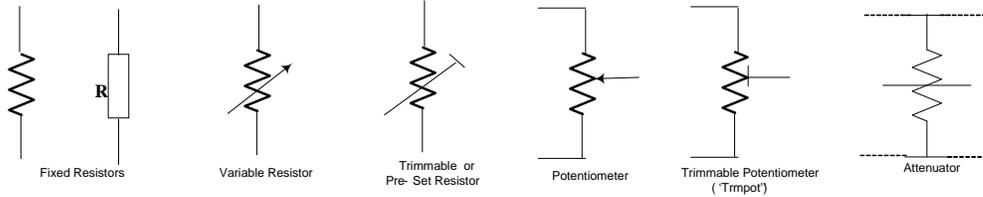
APPENDIX – F

SOME COMMON STANDARD ELECTRICAL GRAPHIC SYMBOLS

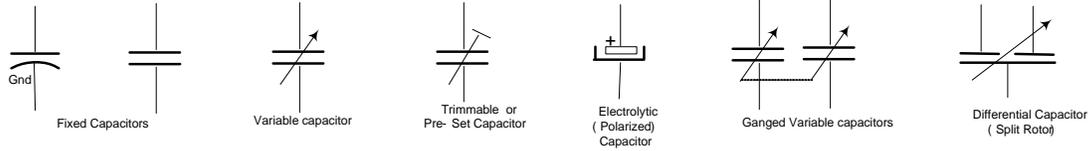
Wire connections & Grounds



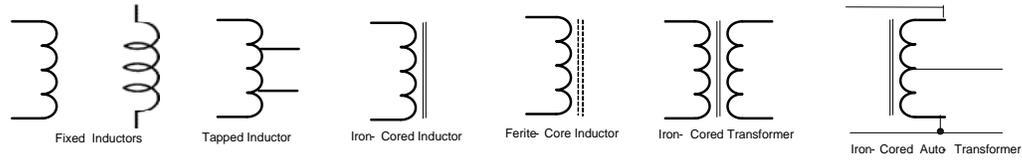
Resistors



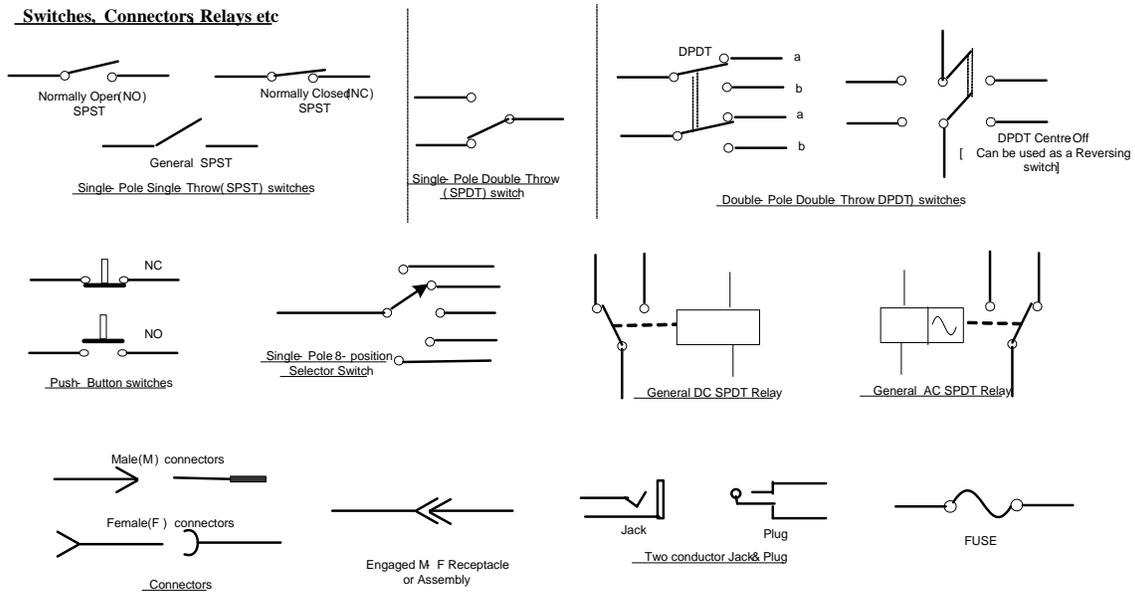
Capacitors



Inductors

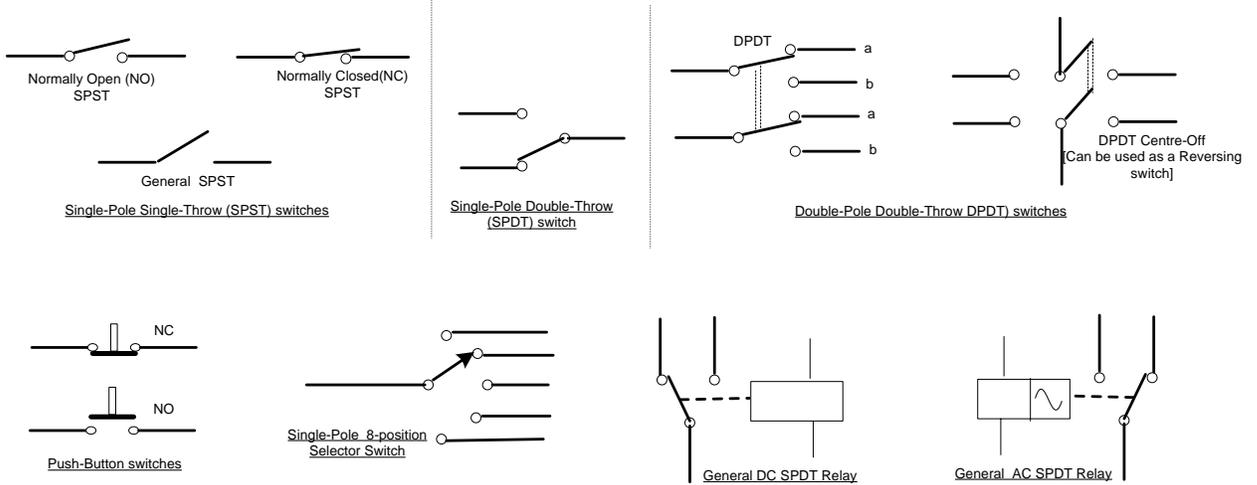


Switches, Connectors Relays etc

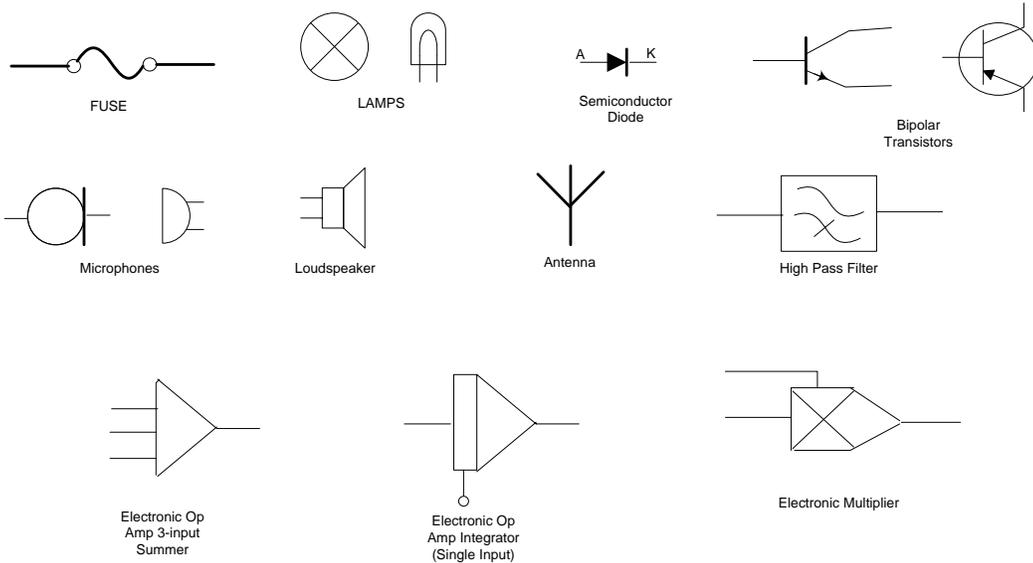


**SOME COMMON STANDARD ELECTRICAL GRAPHIC SYMBOLS,
(Cont'd)**

Misc. Switches & Relays



Miscellaneous devices



Faculty of Engineering and Computer Science Expectations of Originality

This form sets out the requirements for originality for work submitted by students in the Faculty of Engineering and Computer Science. Submissions such as assignments, lab reports, project reports, computer programs and take-home exams must conform to the requirements stated on this form and to the Academic Code of Conduct. The course outline may stipulate additional requirements for the course.

1. Your submissions must be your own original work. Group submissions must be the original work of the students in the group.
2. Direct quotations must not exceed 5% of the content of a report, must be enclosed in quotation marks, and must be attributed to the source by a numerical reference citation¹. Note that engineering reports rarely contain direct quotations.
3. Material paraphrased or taken from a source must be attributed to the source by a numerical reference citation.
4. Text that is inserted from a web site must be enclosed in quotation marks and attributed to the web site by numerical reference citation.
5. Drawings, diagrams, photos, maps or other visual material taken from a source must be attributed to that source by a numerical reference citation.
6. No part of any assignment, lab report or project report submitted for this course can be submitted for any other course.
7. In preparing your submissions, the work of other past or present students cannot be consulted, used, copied, paraphrased or relied upon in any manner whatsoever.
8. Your submissions must consist entirely of your own or your group's ideas, observations, calculations, information and conclusions, except for statements attributed to sources by numerical citation.
9. Your submissions cannot be edited or revised by any other student.
10. For lab reports, the data must be obtained from your own or your lab group's experimental work.
11. For software, the code must be composed by you or by the group submitting the work, except for code that is attributed to its sources by numerical reference.

You must write one of the following statements on each piece of work that you submit:
For individual work: **“I certify that this submission is my original work and meets the Faculty's Expectations of Originality”**, with your signature, I.D. #, and the date.
For group work: **“We certify that this submission is the original work of members of the group and meets the Faculty's Expectations of Originality”**, with the signatures and I.D. #s of all the team members and the date.

A signed copy of this form must be submitted to the instructor at the beginning of the semester in each course.

¹ Rules for reference citation can be found in “Form and Style” by Patrich MacDonagh and Jack Bordan, fourth edition, May, 2000, available at <http://www.encs.concordia.ca/scs/Forms/Form&Style.pdf>.
Approved by the ENCS Faculty Council February 10, 2012

I certify that I have read the requirements set out on this form, and that I am aware of these requirements. I certify that all the work I will submit for this course will comply with these requirements and with additional requirements stated in the course outline.

Course Number: _____

Instructor: _____

Name: _____

I.D. # _____

Signature: _____

Date: _____

IN CASE OF EMERGENCY REMAIN CALM AND FOLLOW THESE INSTRUCTIONS

Fire/Evacuation



- Fire:**
- If you see smoke or fire activate the nearest fire alarm.
- Evacuation:**
- Stay calm; do not rush or panic
 - Safely stop your work,
 - Gather your personal belongings; coat, purse, etc...
 - Close and lock your door and windows.
 - Use stairs only; do not use elevators or escalators,
 - Once outside, move away from the building.
 - Do not re-enter the building until instructed to do so by Security.

Shelter In Place



- Communication:**
- Shelter-in-Place will be announced by intercom P.A. voice communication, text messaging,
 - Fire alarms will not be sounded.

- Procedures:**
- Lock classroom, office and lab doors if possible, remain quiet and do not enter the hallway,
 - Should the fire alarm sound, DO NOT evacuate the building unless:
 1. You have first hand knowledge that there is a fire in the building,
 2. You are in imminent danger, or
 3. You have been advised by Security or Police to evacuate the building.
 - Crouch down in the areas that are out of sight from doors and windows,
 - Anyone in the hallways are to seek shelter in the nearest classroom,
 - Anyone outdoors on campus should immediately take cover,
 - If safe you can call 514-848-(8800) for more information on the situation.

Suspicious Person/Package



- Suspicious Person:**
- Do not physically confront the person,
 - Do not let anyone into a locked building/office,
 - Call Security @ 514-848-(3717),
 - Provide as much information as possible about the person and his or her direction of travel.

- Suspicious Package:**
- Do not touch or disturb object,
 - Call Security @ 514-848-(3717),
 - Notify your Supervisor.

Medical Emergencies



- In the event of a serious or life threatening injury or illness;
- From a safe location; call Security immediately at 514-848-(3717),
- Ensure your personal security before attempting first-aid,
- Provide the victim appropriate first-aid & comforting,
- Do not give the victim anything to drink or eat.

**If the injury is the result of a fall or significant trauma:
Do not move the victim unless absolutely necessary.*

Hazardous Materials



- If an emergency develops or if anyone is in danger, call 514-848-(3717),
- Move away from the site of the hazard to a safe location, Follow the instructions of Emergency Personnel,
- Alert others to stay clear of the area,
- Notify Emergency Personnel if you have been exposed to the hazard or have information about the release.

Power Failure



- Remain calm and move cautiously to a lighted area,
- Do not evacuate unless asked to by Emergency Personnel,
- Do not use candles!
- For localized outages, contact Security at 514-848-(3717).

BLANK