Faculty of Engineering and Technology

Electrical and Computer Engineering Department

Basic Electrical Engineering Lab
(ENEE2101)

Prepared By

Electrical and Computer Engineering Department

July 2023

Table of contents

Lab Safety Instructions and Rules

General Behavior

- During the laboratory session, it is required to have an experimental setup checked and approved by the instructor before starting data collection.
- Never hurry. Work deliberately and carefully.
- Smoking, eating, drinking and use of cell phones is not allowed during the laboratory session.

- Please don’t yell, scream, or make any sudden loud noises that could startle others who are concentrating on their work.
- When you are done with your experiment or project, all components must be dismantled and returned to proper locations.
- Dress properly during all laboratory activities. Long hair, dangling jewelry, and loose or baggy clothing are a hazard in the laboratory. Long hair must be tied back and dangling jewelry and loose or baggy clothing must be secured.

First Aid & fire

- First aid equipment and fire extinguisher are available in the lab, ask your instructor about the nearest kit.

Academic Instructions for Performing Experiments

- Each Student should prepare for the lab by reviewing the theoretical background for the circuits under test.
- Students will work in groups of 2-3 students maximum.
- Each group should prepare a report and submit it at the beginning of next lab session.
- **Data sheet should be signed by the instructor before you leave the laboratory, otherwise your report will not be accepted.**

Experiment 1
Introduction to PSPICE and report writing

➢ Objectives:

This experiment provides an introduction to the basics of circuits simulation using "PSPICE 9.1 schematic" which will be used in pre-labs through this course. Also, an introduction to technical report writing is provided.

➢ Equipment Required:

1. Personal computer with PSPICE 9.1 and Microsoft word.

➢ Introduction:

Through the course this lab students have to prepare for experiments by studying theoretical background related to experiment, and by doing pre-labs which includes simulation of circuits. Computer simulation programs, such as PSpice, simulates the behavior of electric circuits on a digital computer and tries to emulate both the signal generators and measurement equipment such as multimeters, oscilloscopes, and curve tracers. Though, by using PSPICE it is like you have a virtual circuit lab on your personal computer.

Also, by using PSPICE students can easily subject the circuit to various stimuli (such as input signals and power supply variations) and to see the results plotted out graphically using PSpice's post processor called Probe. Therefore, students can obtain results before they come to lab, and the laboratory experiments become reinforcement to the subject matter at hand.

We will be using PSPICE 9.1 student version for simulation, and you can use newer versions provided by OrCAD. The first part of this experiment is dedicated to introduce students to the different applications of PSPICE that are related to the lab.

The second part of this experiment is dedicated to introduce students to the guidelines of report writing. The Bachelor of Science degree in Electrical Engineering involves numerous courses that require written reports. These courses also include laboratories, which require reports too. The fact is, once you graduate, industry will require you to write well. In some cases, you will be involved in writing proposals or possibly final design reports. Certainly, you will always be required to write short reports and memos detailing your activities. The second part of this experiment is dedicated to introduce you to the basic guidelines of report writing.

➢ Installing PSPICE 9.1

- To download PSPICE 9.1 student version, visit the following link: http://www.electronics-lab.com/downloads/circutedesignsimulation/?page=5
  You will find a list of programs, scroll down until you see **PSPICE 9.1 Student Version**, then click download. The setup is straight forward, however, in case you needed help check the following video and follow steps. https://www.youtube.com/watch?v=tCFjjHY94Ro

➢ Building and simulating circuits

- To open PSPICE 9.1, search your computer for **"Schematic"** and open the program. This is shown in figure.1 for windows 10 users. **Note: zoom in to see figure details if not clear.**

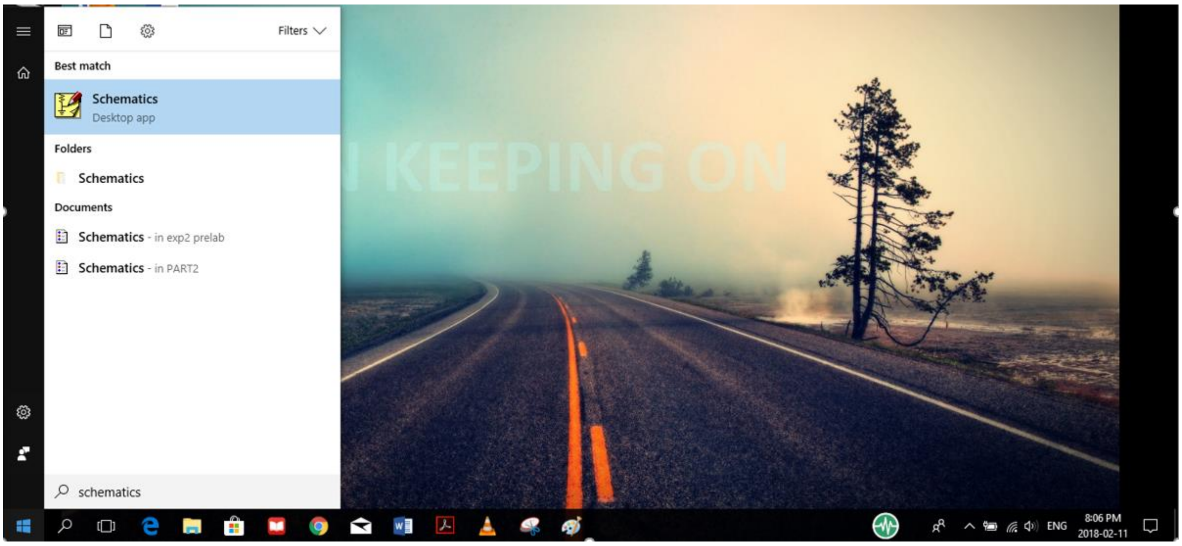


Figure.1

- Click on Schematics icon, then you will see the window shown on figure.2

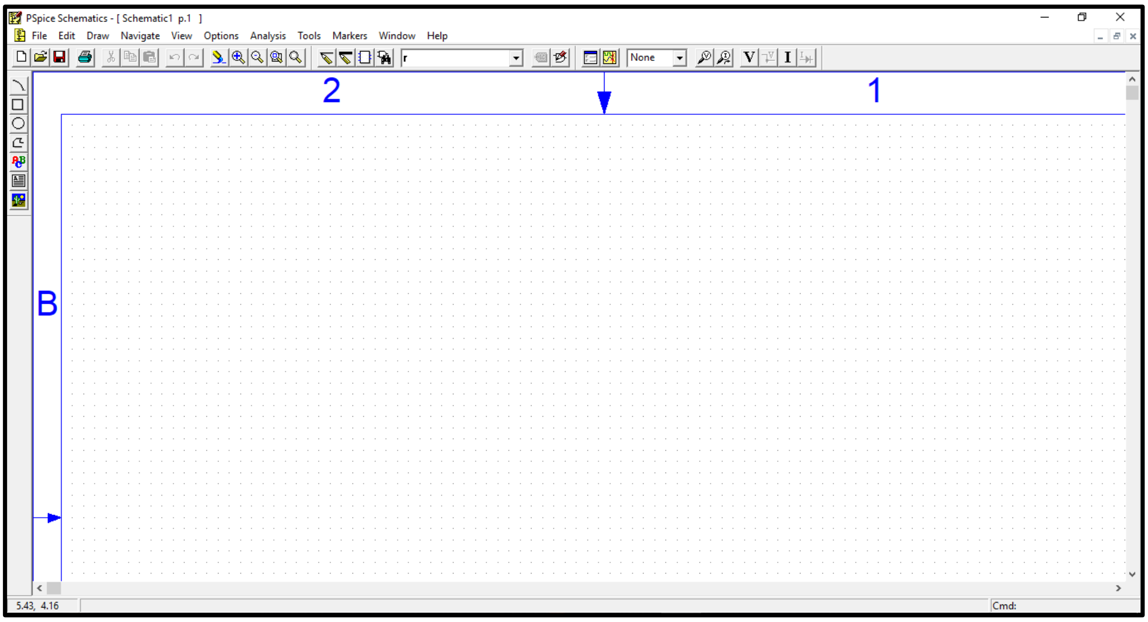


Figure.2

- To build the circuit of figure.3 on PSPICE, click on **“get new part”** icon shown in figure.4:

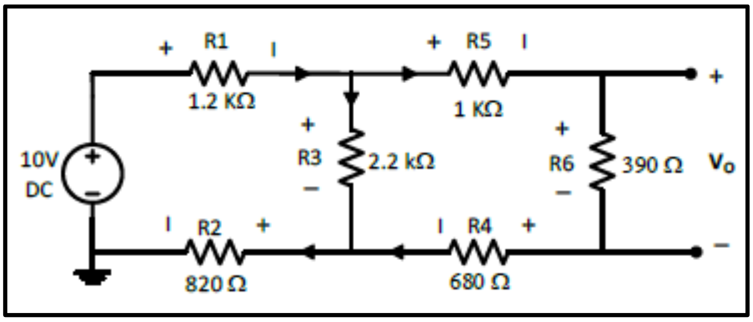


Figure.3

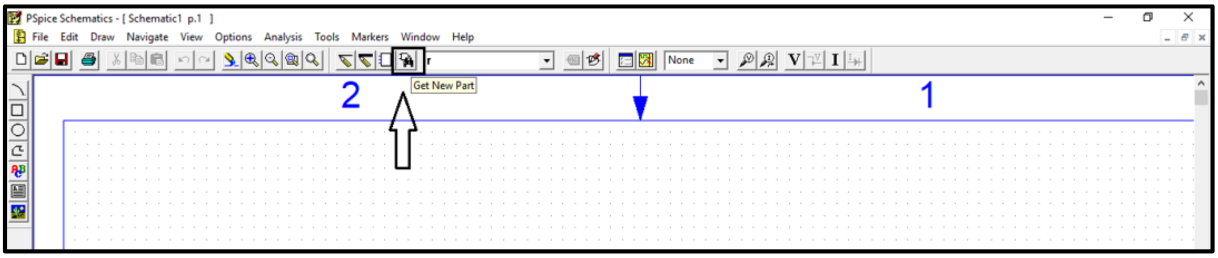


Figure.4

- When you click on get new part icon, you will see the window shown on the left side in figure.5. This is a list of all components available in the program. Click on “advanced” so you can see the picture of part you want to add as shown on the right side in figure.5.

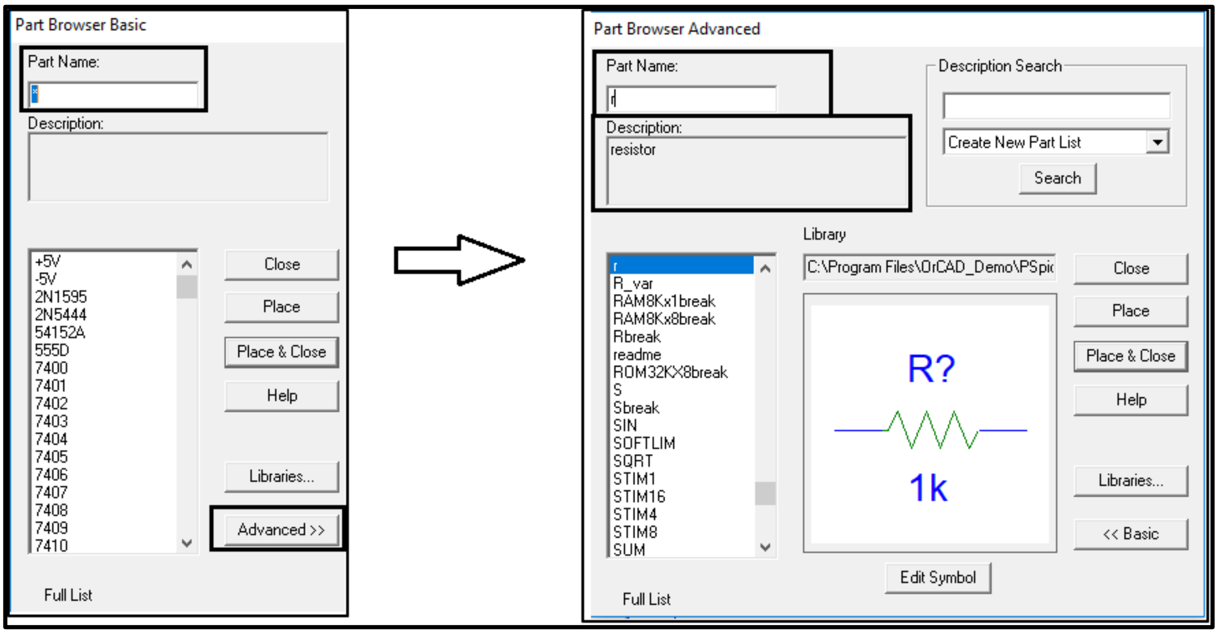


Figure.5

- To add a circuit part, you can search by typing part name in the specified field. After finding the part you want, click on “place and close”, then click wherever you want to add a part. After you finish adding components, click on “esc” on your keyboard to end the mode.

- Note that each circuit element in PSPICE have a specific name that might be different from its name in circuit analysis. Table.1 provides the names of mostly used circuit elements in the circuit lab.

Table.1

Circuit element	Part name in PSPICE
Resistor	R
Capacitor	C
Inductor	L
DC Voltage source	vdc
DC Current source	Idc
Periodic square voltage source	vpulse
AC Voltage source: two types	vsin (sinusoidal voltage source used in transient analysis)
	vac (variable frequency source used in ac sweep analysis)
ground	gnd_analog
Operational amplifier	ua741

- Note that when you place parts, each circuit part in PSPICE has a name and a value as shown in figure.6 Always make sure to place circuit parts in PSPICE in a similar way to the circuit given in manual. For example, check the way the parts are placed in figure.6.

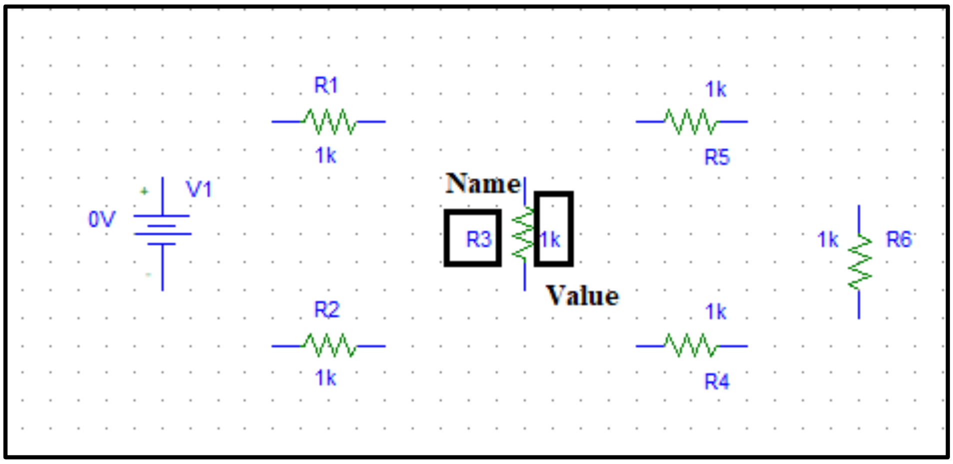


Figure.6

- To connect the components, click on "draw wire" icon shown in figure.7, **a common mistake** that students do is clicking on "draw bus" icon which is next to "draw wire". Avoid doing this mistake! also avoid drawing wire over components (shorting parts). Click where you want each vertex of the wire. Each click ends a wire segment and starts a new one

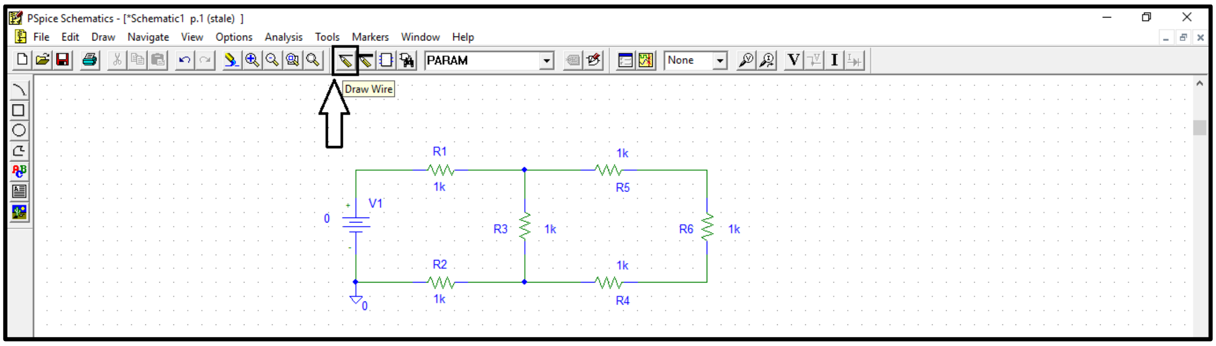


Figure.7

- To change a part value e.g. resistance, voltage source … etc. double click on its value and the window in figure.8 will show up, type the value you want inside the box then click ok.

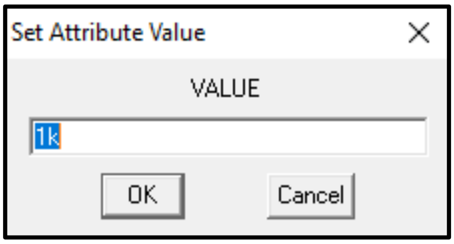


Figure.8

- When elements such as resistors and voltage sources are given values, it is convenient to use unit prefixes. PSPICE supports the prefixes listed in table.2. Note that the letter must immediately follow the value – no spaces. Also, PSPICE is case insensitive so, there is no difference between 1M and 1m in PSPICE.

Table.2

PSPICE Unit Prefixes		
K - kilo - 10^{3}	MEG - mega - 10^{6}	G - giga - 10^{9}
M - Millie - 10^{-3}	U - micro - 10^{-6}	N - Nano - 10^{-9}

- **PSPICE requires that all schematics have a ground**, the voltage there will be zero and all other node voltages are referenced to it. **If you do not place a ground, you will get an error and will not be able to simulate your circuit**. The part you need is either the analog ground (GND_analog) or the earth ground (GND_earth) which are equivalent, you can get them from “get new part option”. In this example, we used GND_analog.
- Your schematic is finished now and ready for saving, click on “save” then on “simulate” button as shown in figure.9.

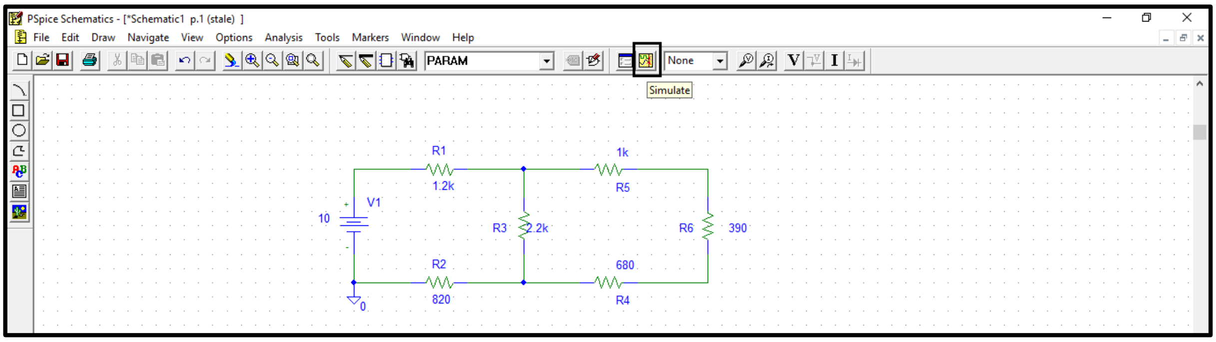


Figure.9

- When you click on "simulate" button the window shown in figure.10 will show up (simulation output window), if your circuit doesn't have any errors you will see the message in the box in figure.10.

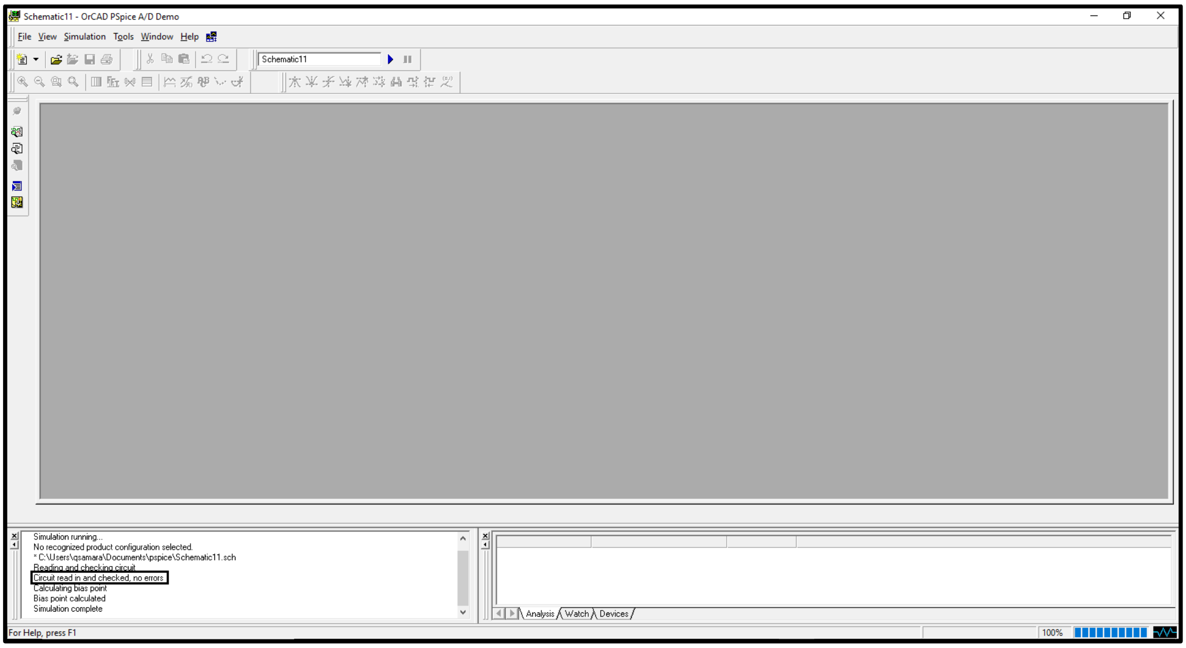


Figure.10

- Return to the schematics window and click on **"V"** and **"I"** buttons shown in figure.11. By clicking on these buttons, PSPICE displays the voltage on each node "**with respect to ground"** and the current on each branch.

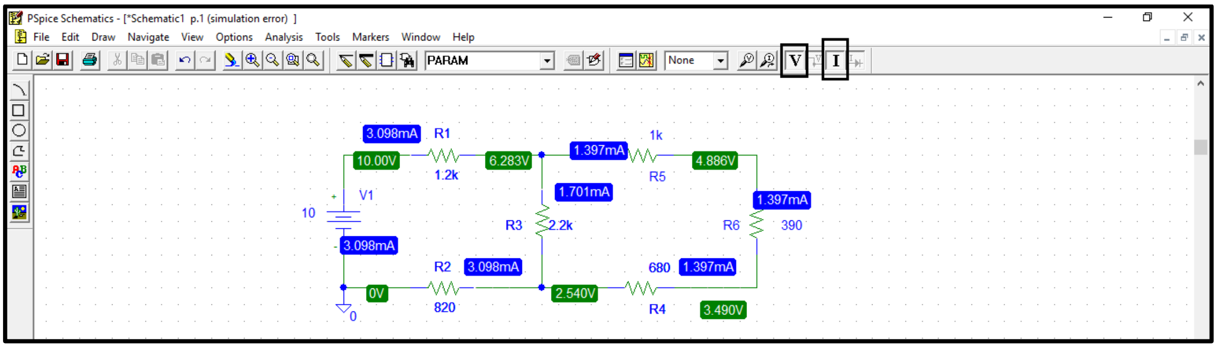


Figure.11

- **The Passive Sign Convention and PSPICE:**

  All currents and voltages in PSPICE and Schematics obey the passive sign convention shown in Figure.12. The voltage across the element is defined positive at node 1 with respect to node 2, and current is entering a device from its "1" end and leaving its "2" end.

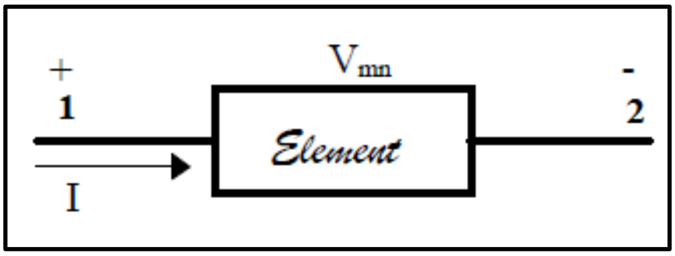


Figure.12

All two leaded passive components have an **implied** "1" end and a "2" end (not visible on schematic). Whenever you place a component, it takes a default position, for example, a resistor, capacitor, or inductor will take a default position with its "1" end to the left as shown in figure13-(a). A component may be rotated by activating it, then right-clicking and selecting Rotate, or by typing the letter "r" (see b). Each rotation moves the component counterclockwise by 90º. To get the "1" end facing up, you must rotate the component 3 times from its default position as indicated in (c).

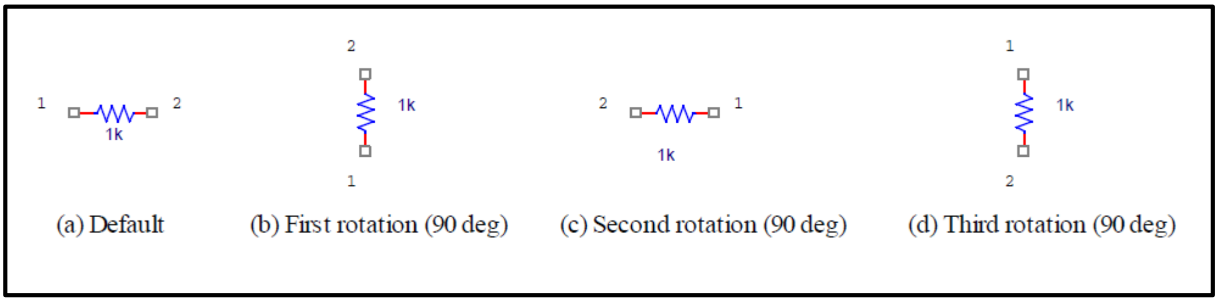


Figure.13

- Knowing about component layout is important when you are viewing your results in Probe to be discussed later.

➢ Types of Analysis Performed by PSPICE

PSPICE is capable of performing four main types of analysis: Bias Point, DC Sweep, AC Sweep/Noise, and Time Domain (transient).

- **Bias Point:**

  The Bias Point analysis is the starting point for all analysis. In this mode, the simulator calculates the DC operating point of the circuit. **This is the type which was explained in the previous section.**

- **DC Sweep**

  The DC Sweep analysis varies a circuit part e.g. voltage source, global parameter (resistor) over a specified range in an assigned number of increments in a linear or logarithmic fashion.

- **Time Domain (transient)**

  The Time Domain (transient) analysis is probably the most popular analysis. In this mode, you can plot the various outputs as a function of time.

- **AC Sweep/Noise**

  The AC Sweep/Noise analysis varies the operating frequency in a linear or logarithmic manner. It linearizes the circuit around the DC operating point and then calculates the network variables as functions of frequency.

➢ Example on DC Sweep

- To perform a DC sweep for the same circuit of the previous section, click on the "setup analysis" icon shown in figure 14.

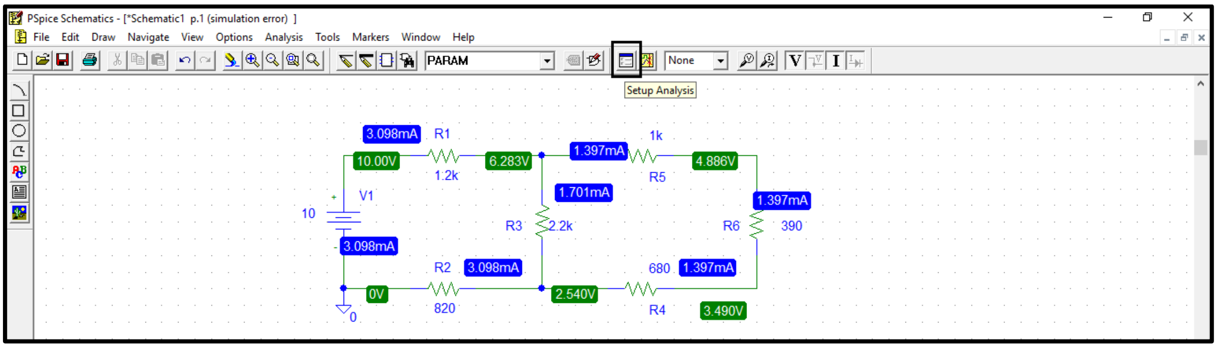


Figure.14

- The window shown in figure.15 will show up:

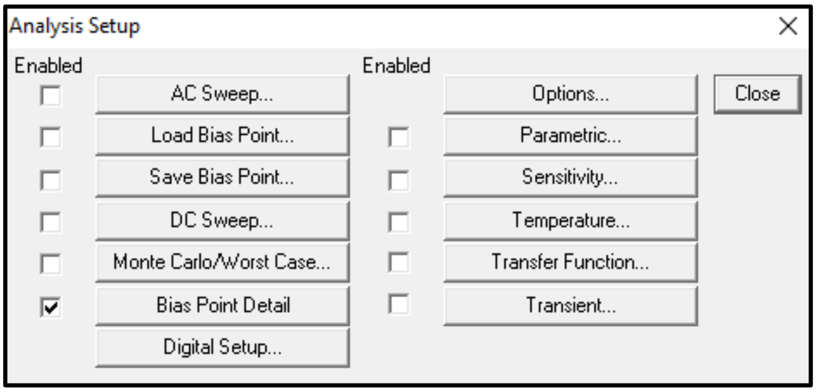


Figure.15

- Click on "DC sweep" and the window shown in figure 16 will show up, you have to fill the spaces indicated inside boxes in figure.16.

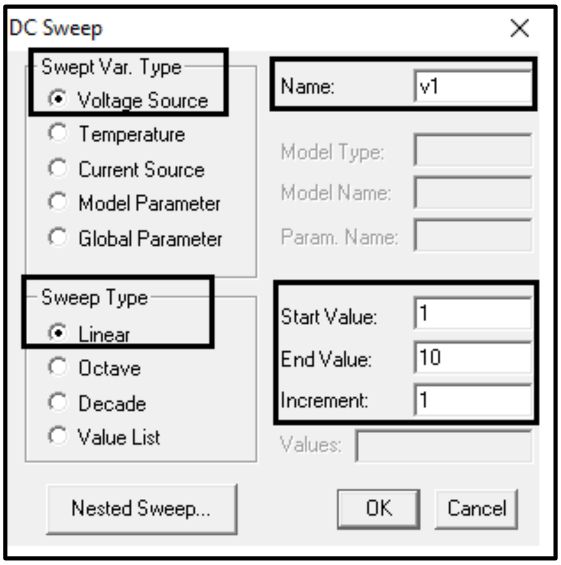


Figure.16

- Click on simulate, then the simulation output window will be as shown in figure.17:

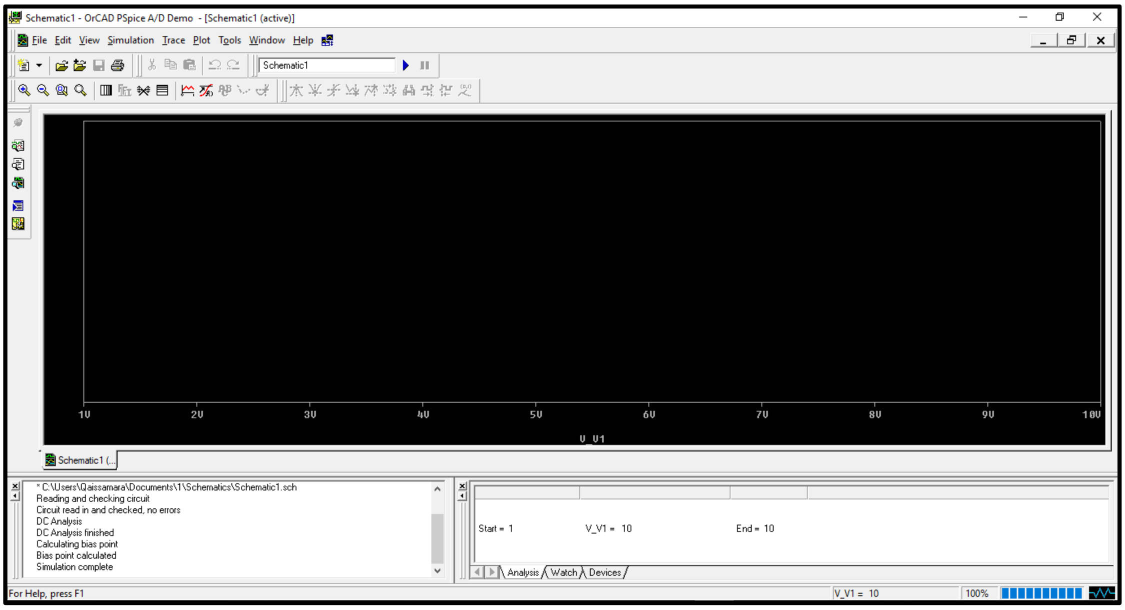


Figure.17

- The output now is displayed as curve rather than a point, to display the voltage across R6 for example click on "Voltage Marker" and place it where you want as shown in figure.18.

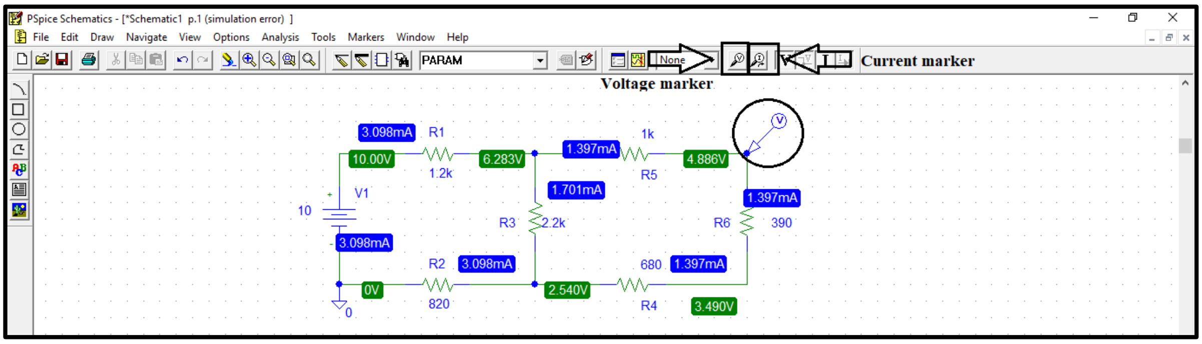


Figure.18

- When you place the marker, the output simulation window will be as shown in figure.19. this is a curve of the voltage at the node you placed the marker **with respect to ground.**

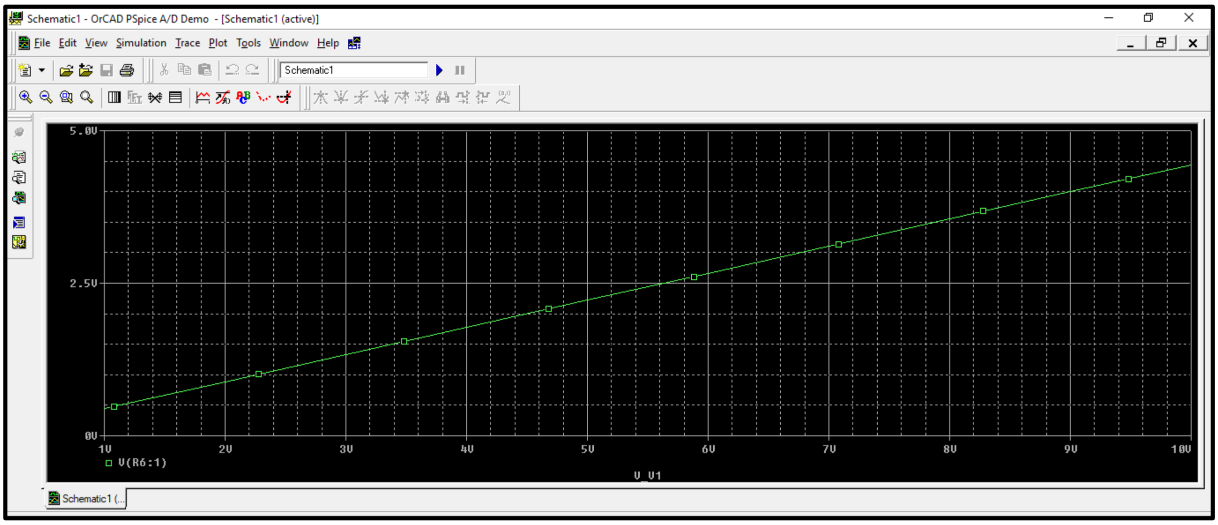


Figure.19

- If you want to display the voltage on R6 only, then go to schematic window, click on marker, and select "mark voltage differential" as shown in figure.20. (you will place two markers)

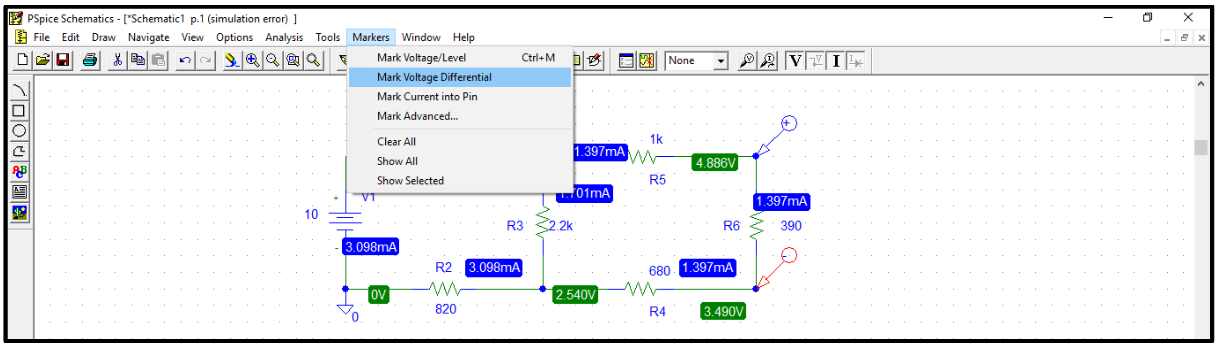


Figure.20

- The output now will be as shown in figure.21. **Note the difference on y-axis range!**

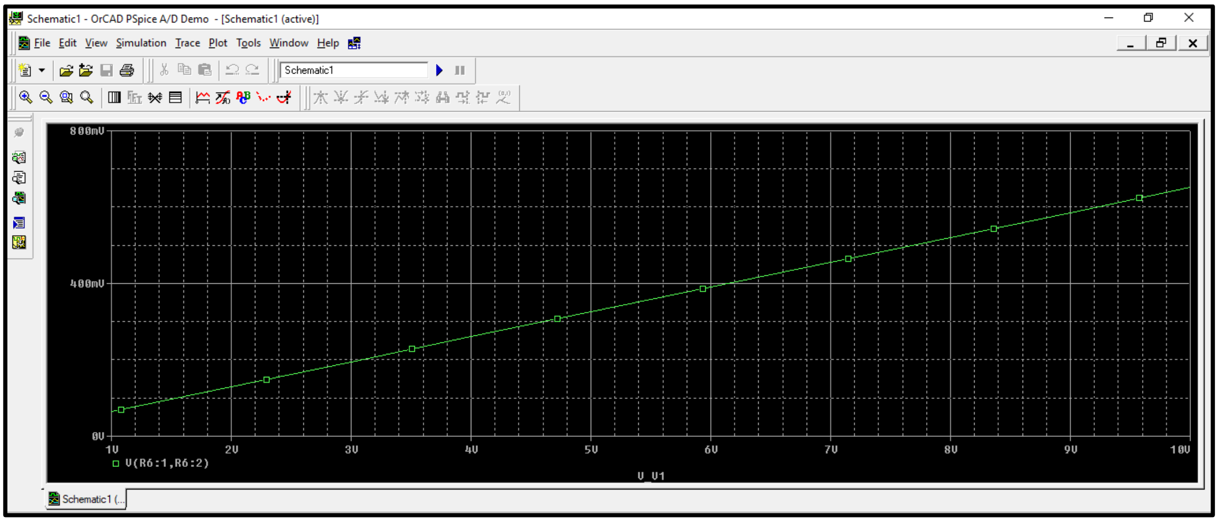


Figure.21

- You also can add curves, (traces) from simulation output window by clicking on traces and select add trace as shown in figure 22. You show current trace by clicking upon Current Maker (note that current marker must be placed on the terminals of a part to display current).

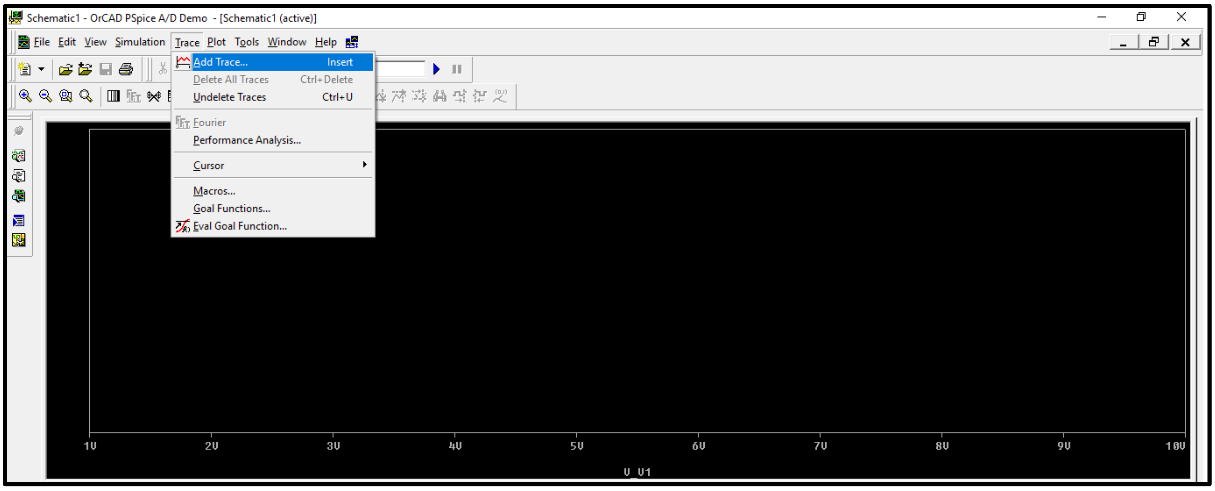


Figure.22

- Then the window shown in figure 23 will show up, in this window you will find a list of all variables in the circuit: (voltages and currents on all parts) to add a trace just click on it and it will show up in the field indicated in the box in figure.23.

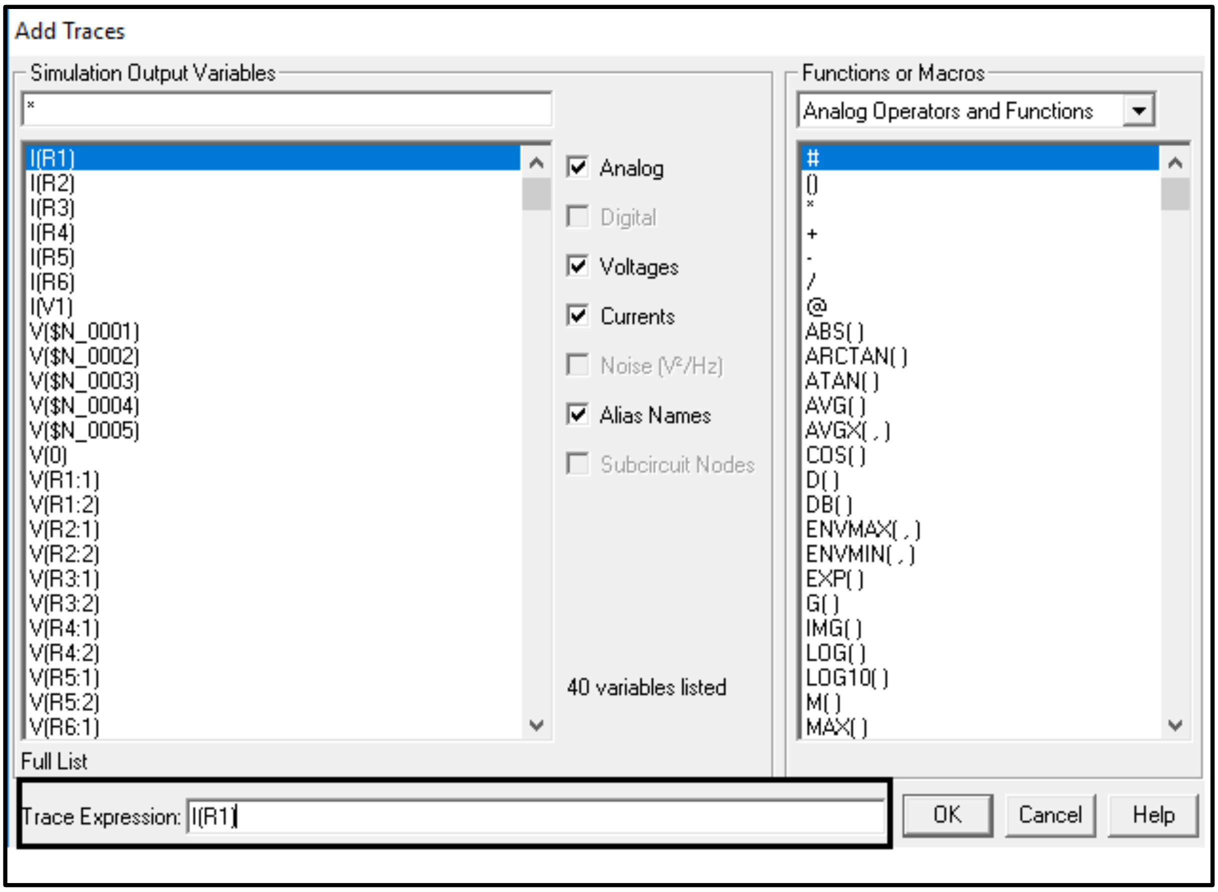


Figure.23

- When you finish selecting variables click on ok, and the trace you selected will be shown in simulation output window as shown in figure.24.

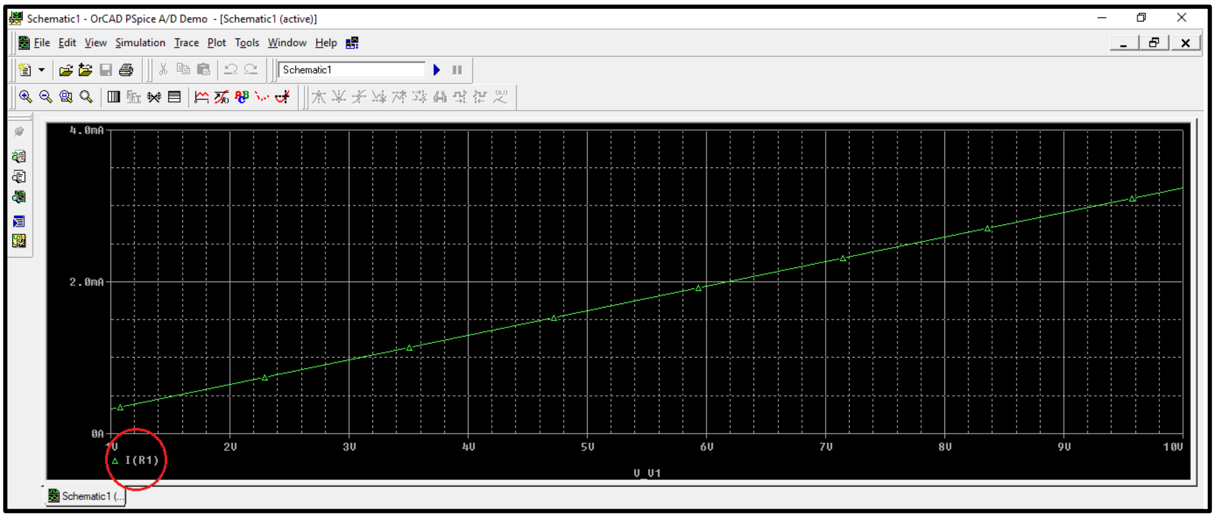


Figure.24

- In the "add trace window you can also add a trace that is a combination of more than one circuit variable e.g. to add a trace of the power dissipated in R6, note the equation inserted in figure 25.

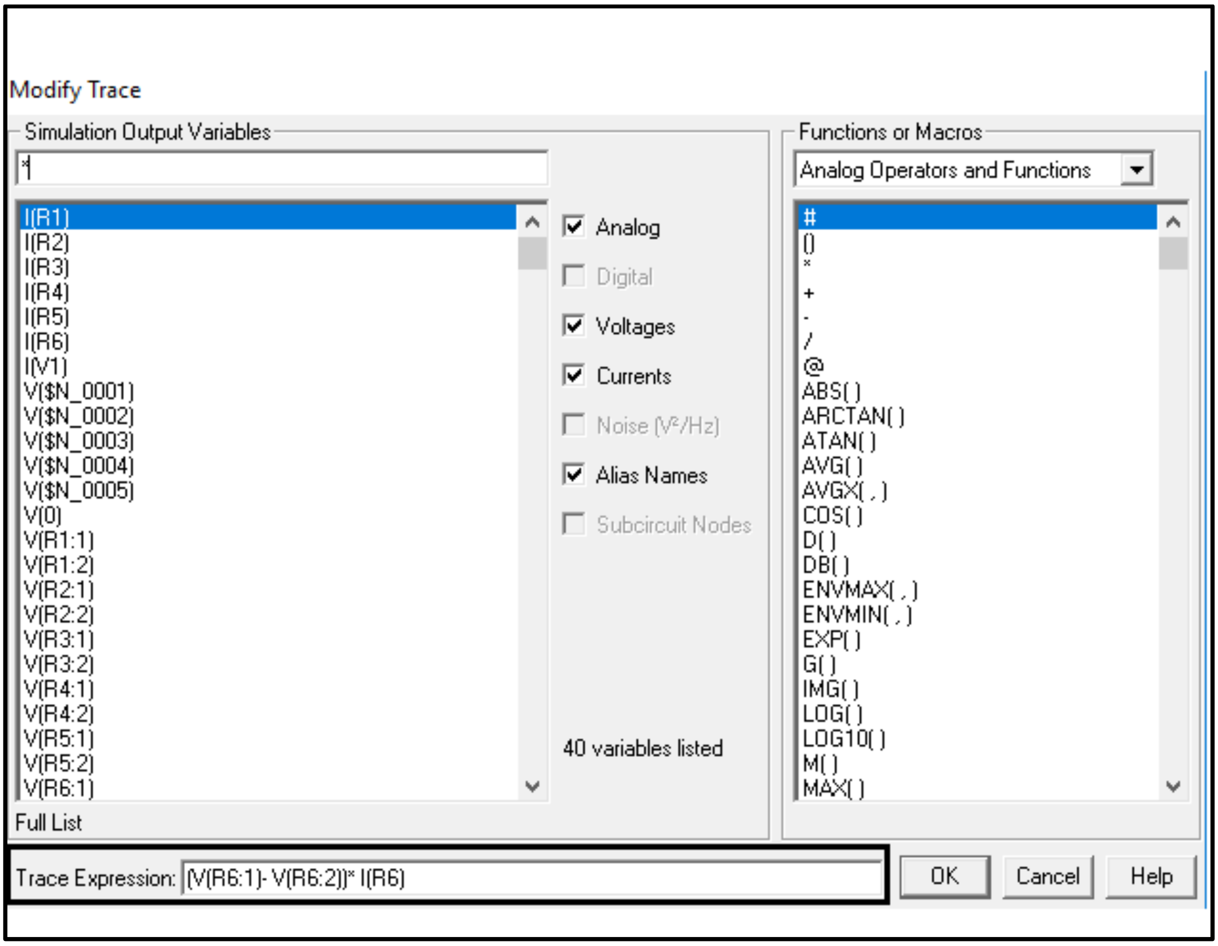


Figure.25

- The output is displayed in figure.26, note that the y-axis unit now is Watt.

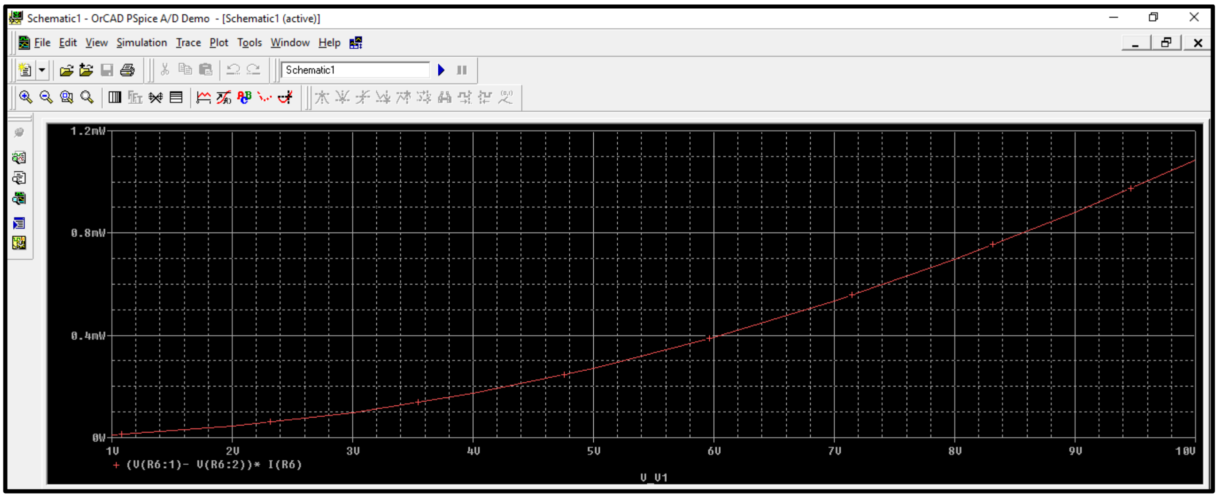


Figure.26

- **To perform a DC sweep for a resistor in the same circuit**, you need to add "param" part, go to "get new part" and search for "param", as shown in figure.27.

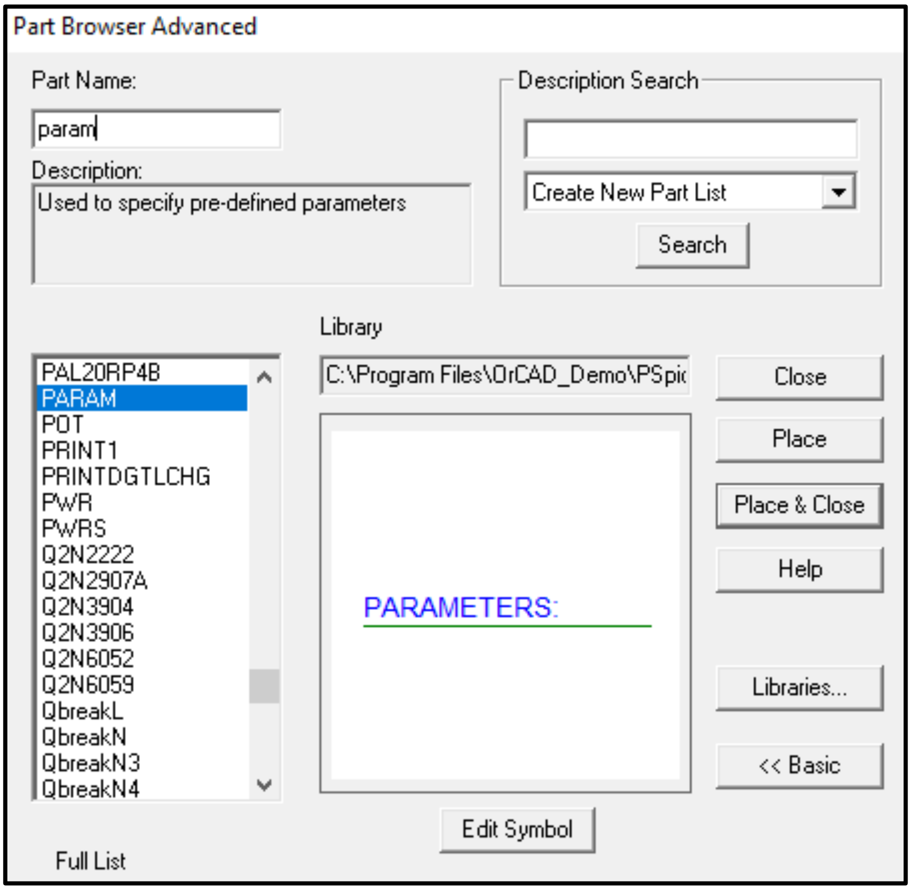


Figure.27

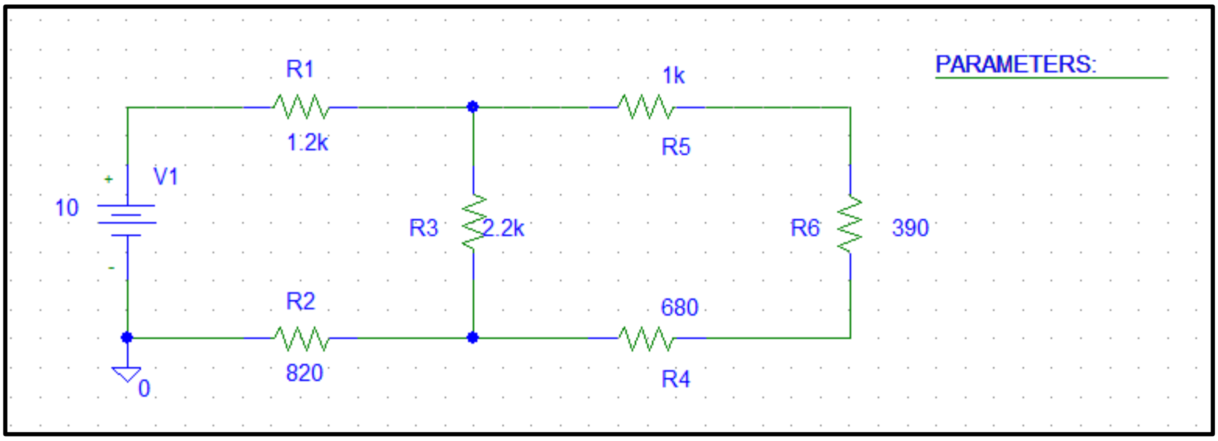


Figure.28

- Double click on "param", and the window shown in figure.29 will show up, insert parameters as shown in figure.

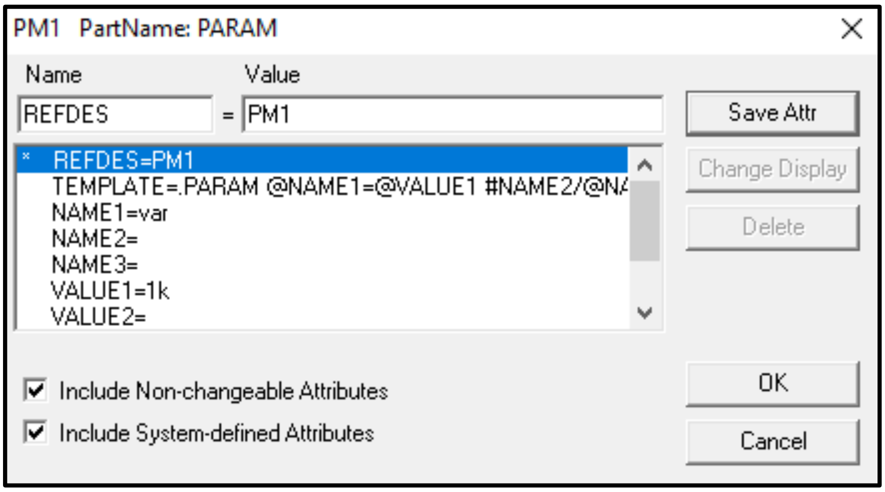


Figure.29

- Click on "setup analysis" and select DC sweep, fill in parameters as shown in figure 30.

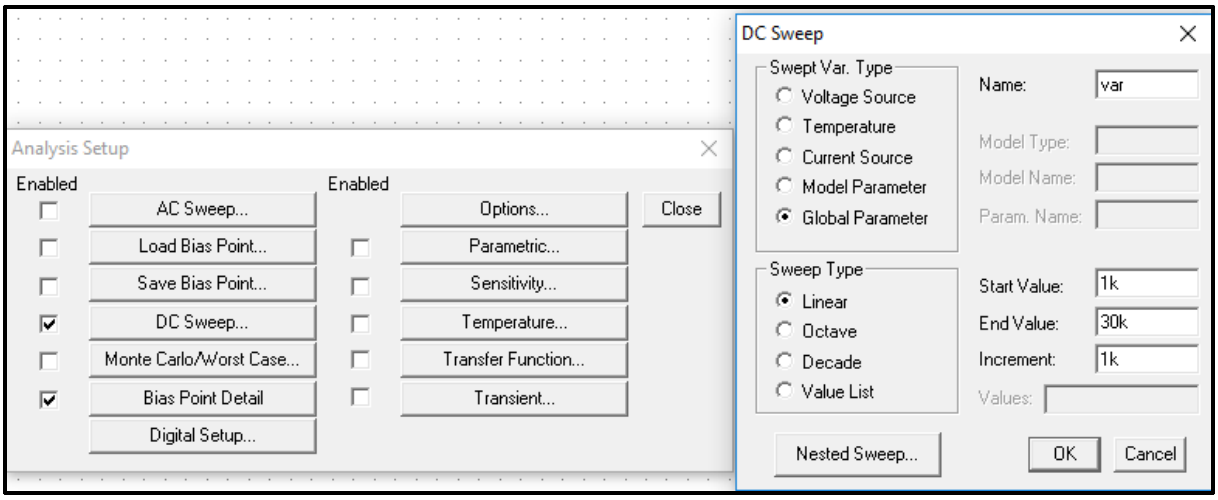


Figure.30

- Type the name of the global parameter inside curly brackets {}, as shown in figure 31.

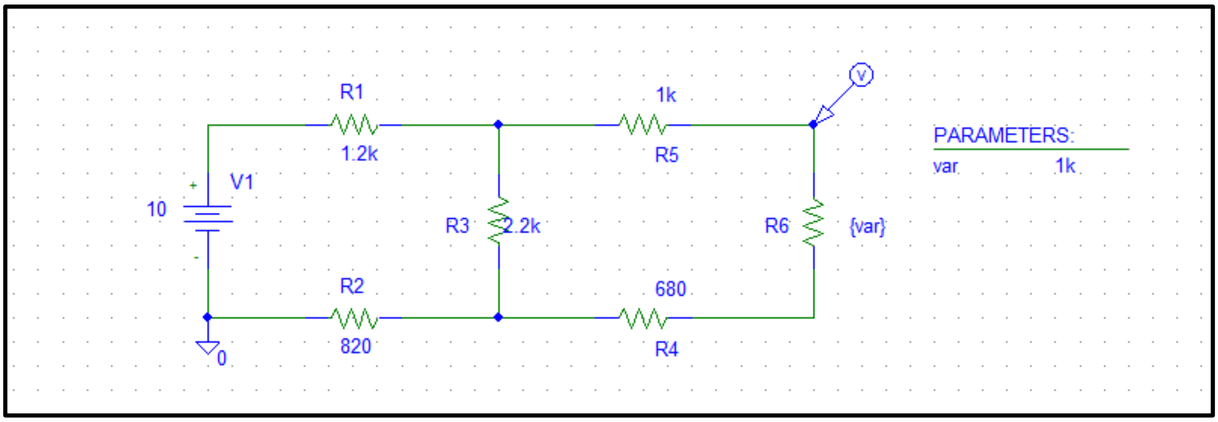


Figure.31

- Click simulate and check the output as shown in figure.32.

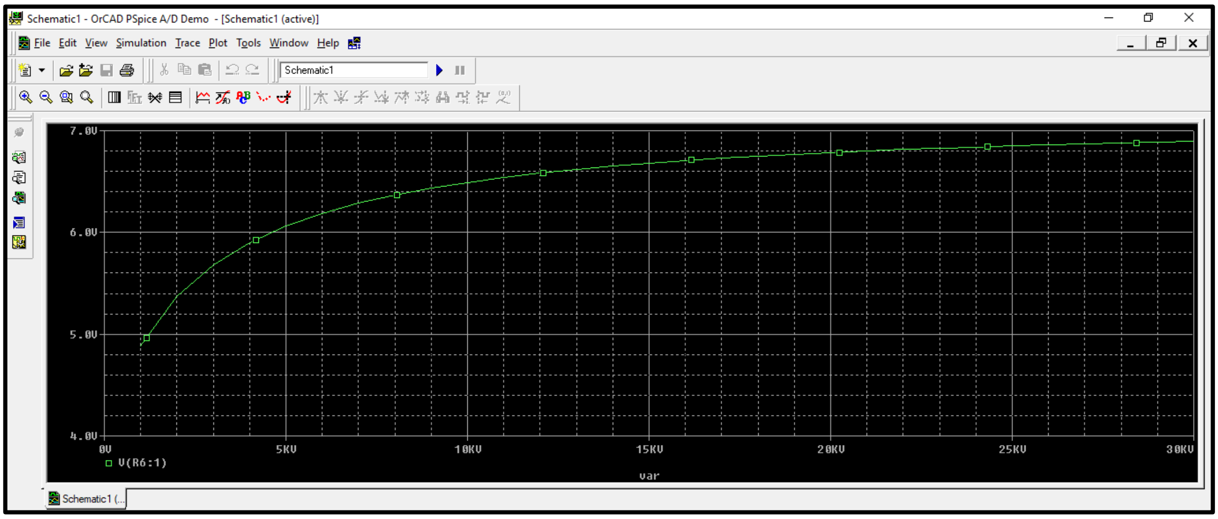


Figure.32

- **To change trace properties such as width and color, right click on the trace and select properties as shown in figure.33.**

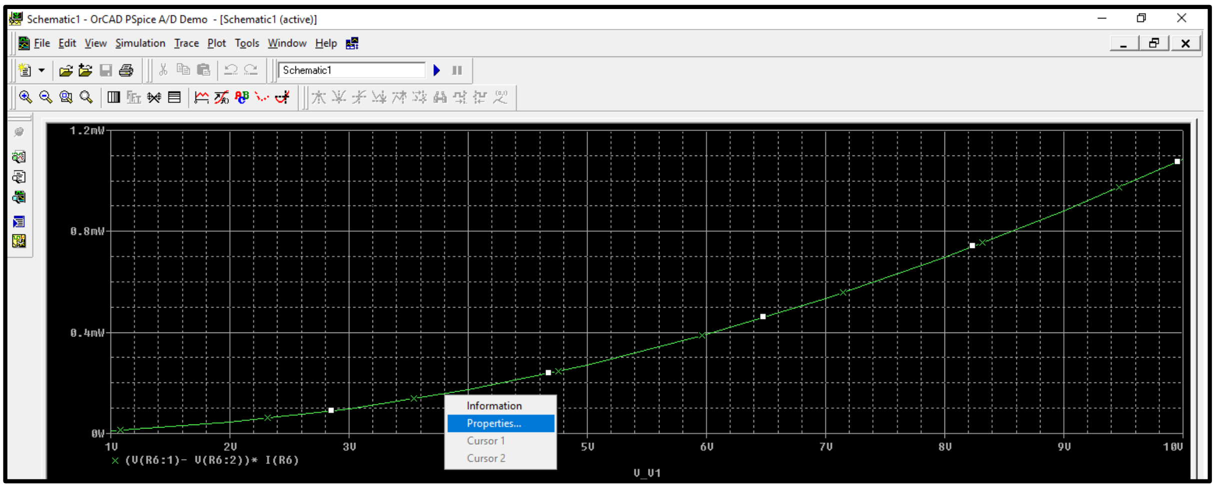


Figure.33

- The window shown in figure.34 will show up, you can change trace properties.

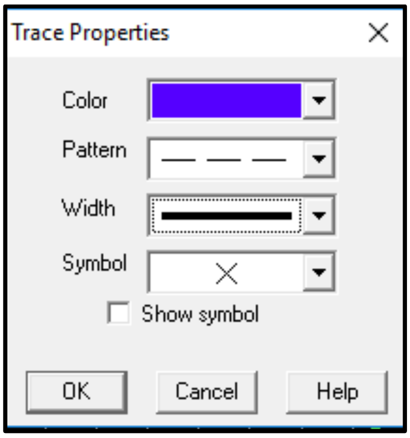


Figure.34

- After changing trace properties, the simulation output window will be shown in figure.35, **always make sure to increase the trace width in your prelab.**

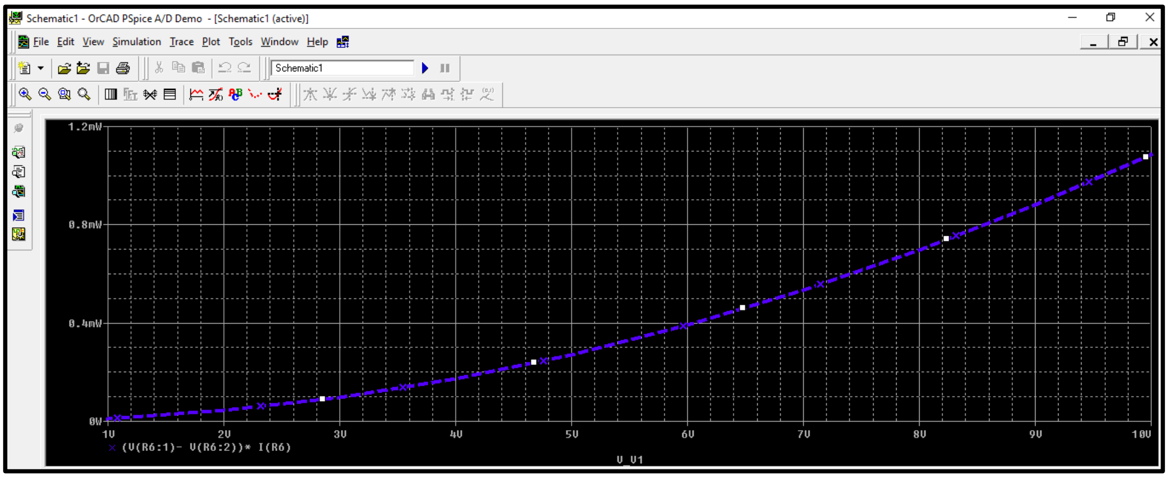


Figure.35

- **To copy the output curve to a word document**, go to the simulation output window and click on window then select "copy to clipboard", then select ok as shown in figure.36.

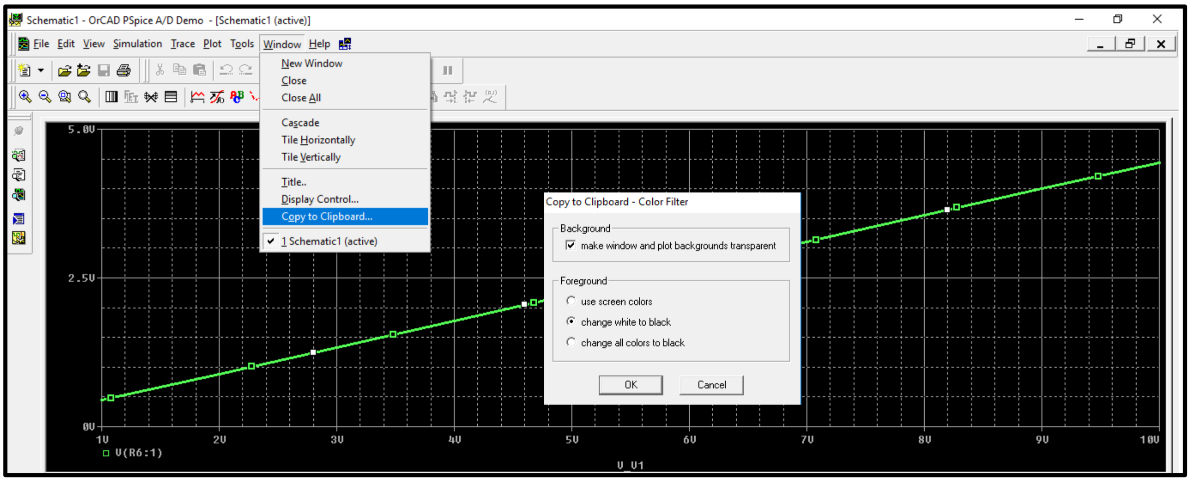
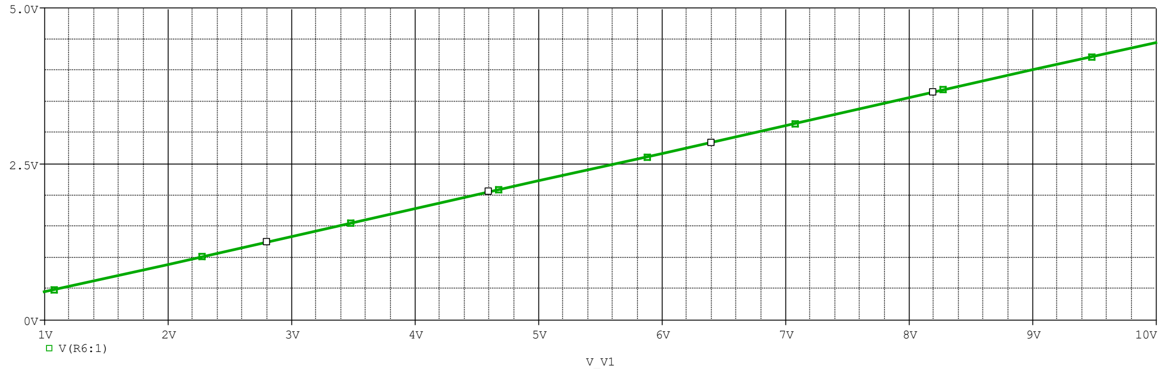


Figure.36

- Then click “paste” or “ctrl+v” inside word document and the output curve is copied a shown.



➢ Example on Transient analysis

- The circuit shown in figure.37 contains a square wave source which is a function of time, so we will use transient analysis.

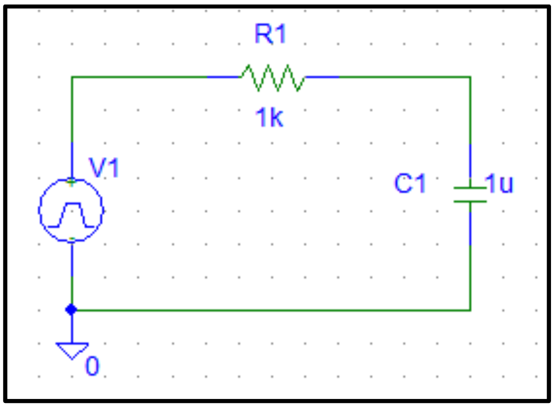


Figure.37

- The source name is “vpulse” and can be found on the components list as shown in figure.38.

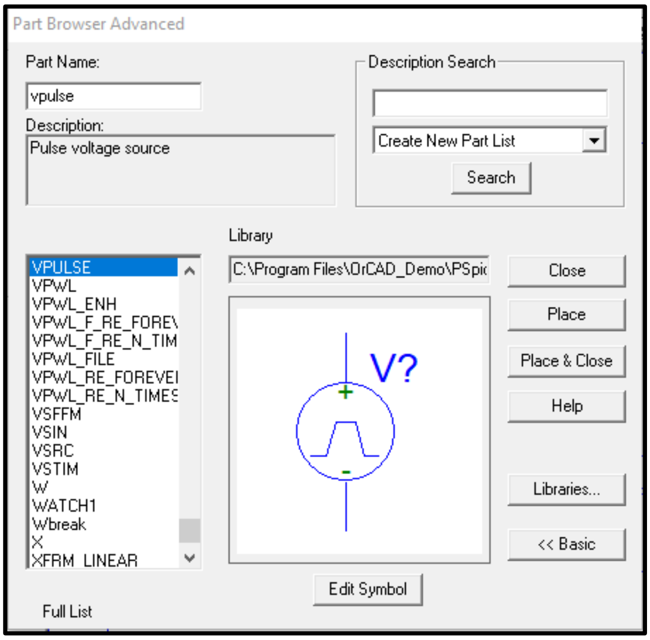


Figure.38

- Double click on the source to insert its parameters, the window in figure.39 will show up.

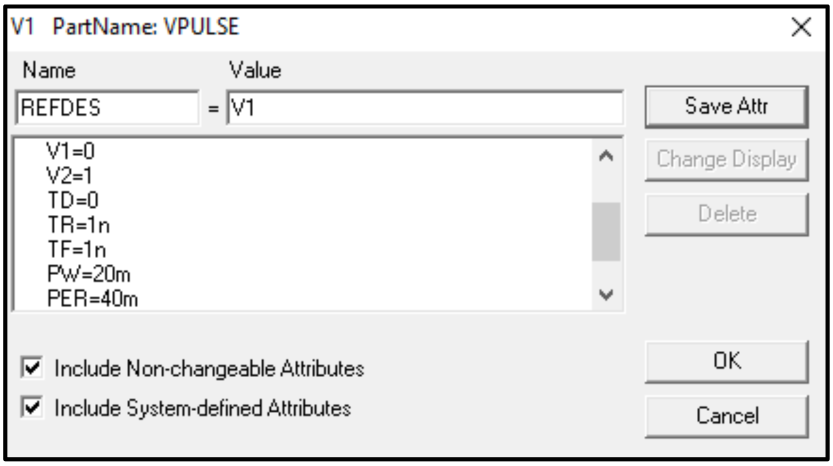


Figure.39

- Table.3 provides the details of this parameters.

Table.3

Parameter	Description	Value
V1	Lowest value	Usually 0
V2	Highest value	Depending on the question e.g. 1
TD	Delay Time	Usually 0
TR	Rise Time	1n
TF	Fall Time	1n
PW	Pulse Width	(1/ 2xFrequency) e.g. 20m
PER	Period	(1/ Frequency) e.g. 40m

- To perform a transient analysis for the circuit of figure.37, click on the "setup analysis" icon and select "Transient", the window in figure.40 will show up.
- There are **two important parameters** that must be determined carefully in transient analysis so you can picture the output properly, these are: **Final time, and step ceiling.**
- **Final time** is the time till which you want to see the output signal, it is usually selected three times the period of the source.
- **Step ceiling** controls the accuracy **(smoothness)** of the output plots, it is usually obtained by dividing the final time on 10^4.

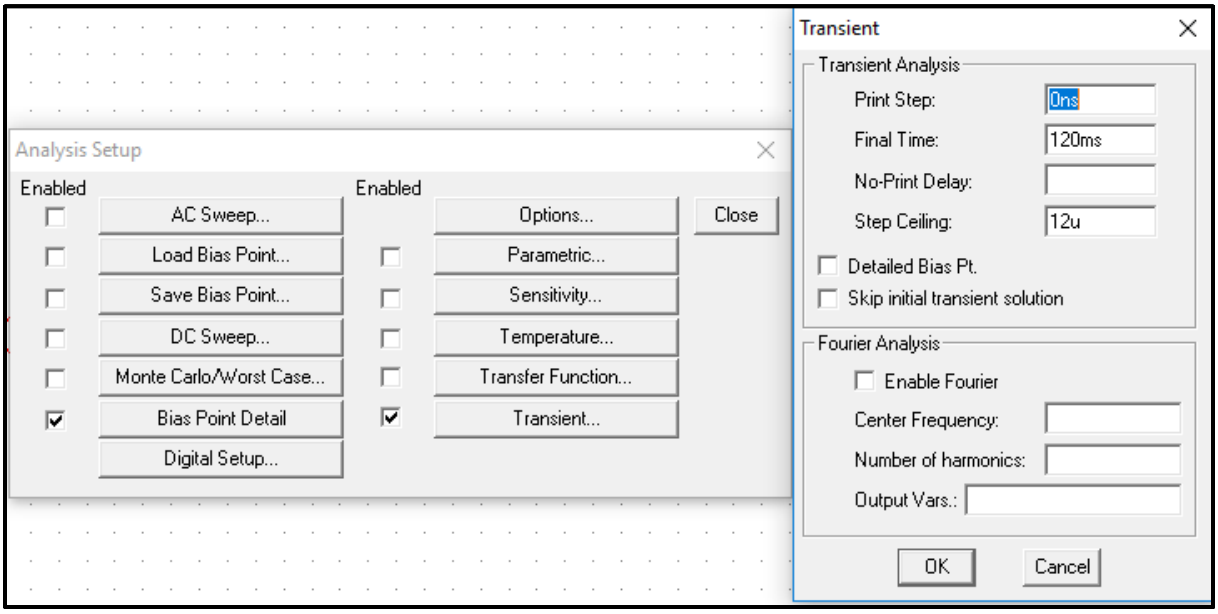


Figure.40

- The output is now displayed in figure.41.

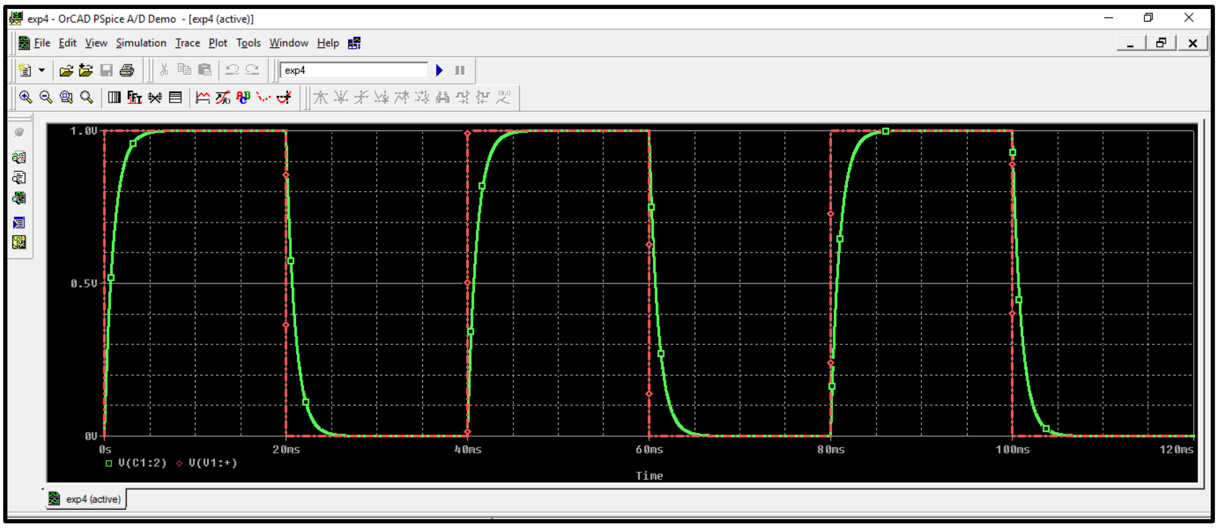


Figure.41

- Cursors are used often used to determine certain points on traces to use cursors click on “toggle cursors” icon shown in figure.42.
- There are two cursors in PSPICE (a1), and (a2), you can move between them by clicking the right and the left buttons of the mouse.
- Each cursor has a reading of x-axis and a reading of y-axis, look at figure.42.

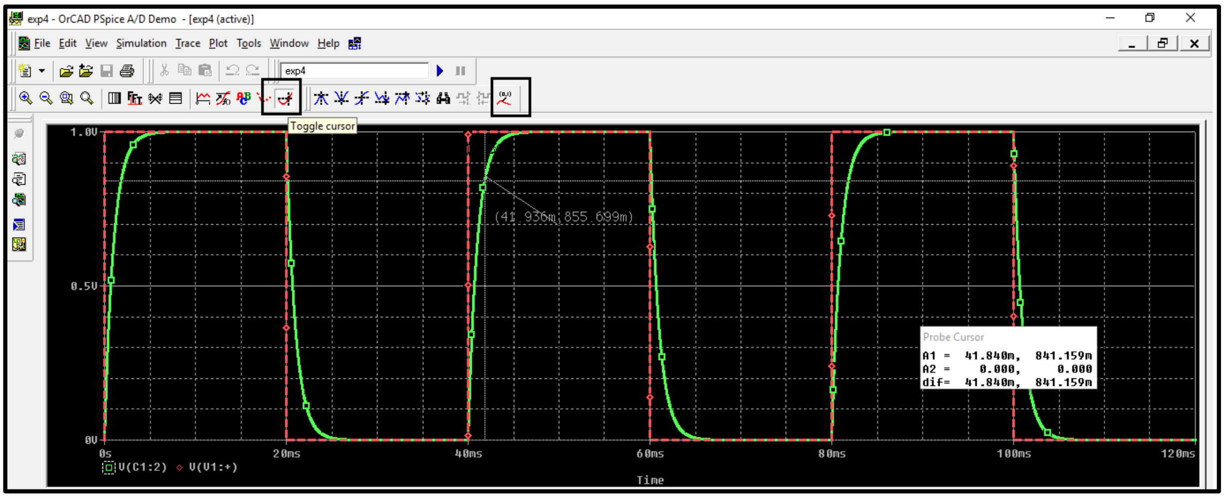


Figure.42

- Another example on transient analysis, sinusoidal voltage source (vsin).

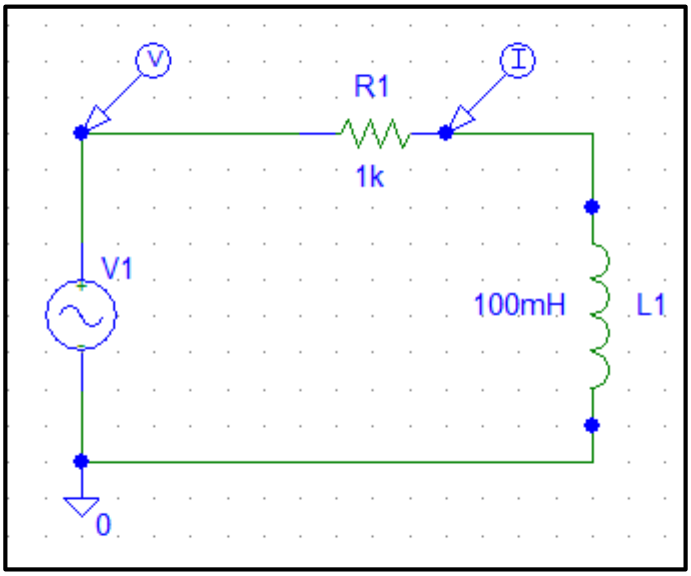


Figure.43

- Source settings.

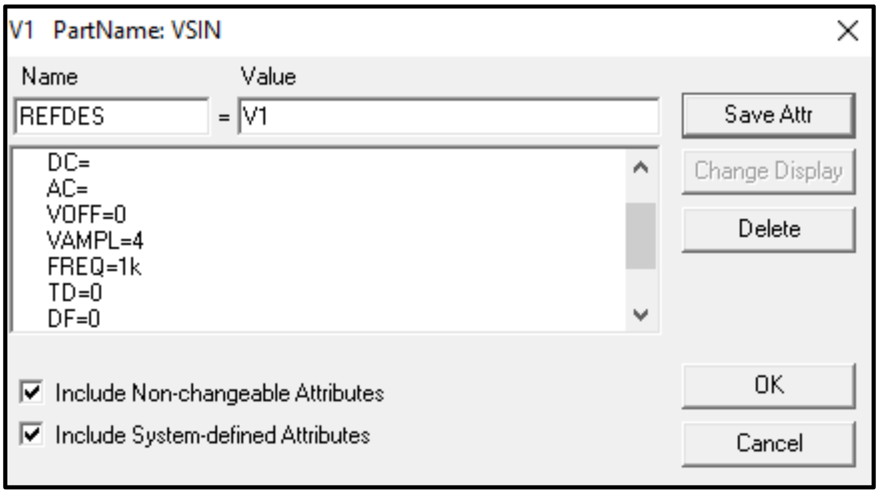


Figure.44

- In the simulation output window, we are viewing the current and voltage, but the voltage is in the range of (-4 to 4) and the current is in the range of (-4m to 4m) so we need **to add a separate y-axis to the output window.**
- Click on "plot", then select "add y-axis", note that you have different y-axes as shown in figure.45, you can move between axes by clicking on the axis you want to add a trace to.

Figure.45

- Voltage and current traces are shown in figure.46, each trace is on a different y-axis range.

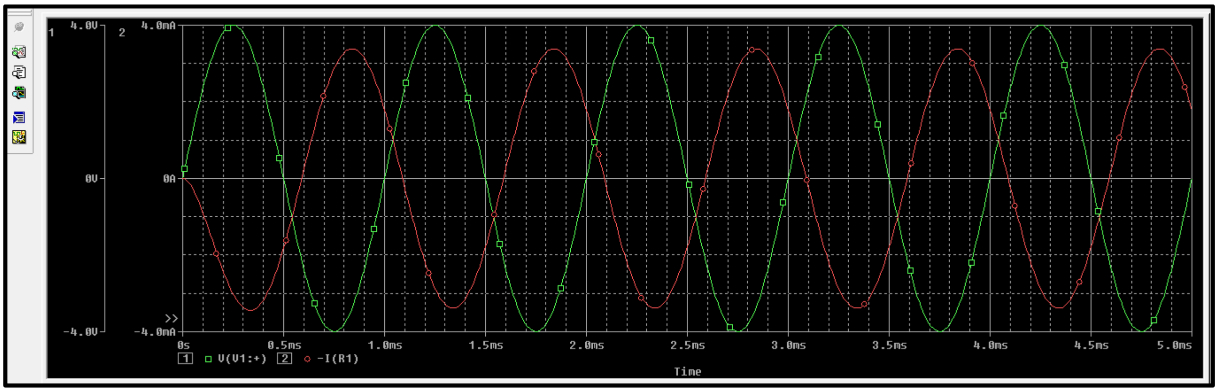


Figure.46

➢ Example on AC sweep analysis

- The circuit shown in figure.47 contains an (vac) source which is used for AC sweep analysis.
- Double click on the ac source to insert setting: only insert voltage magnitude to 1.

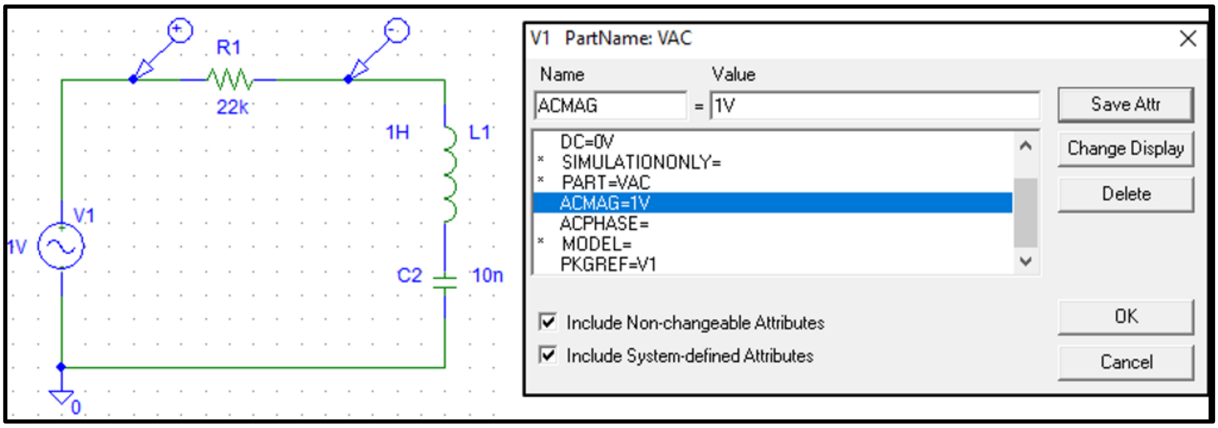


Figure.47

- Setup analysis setting: usually select "AC sweep type" to decade.

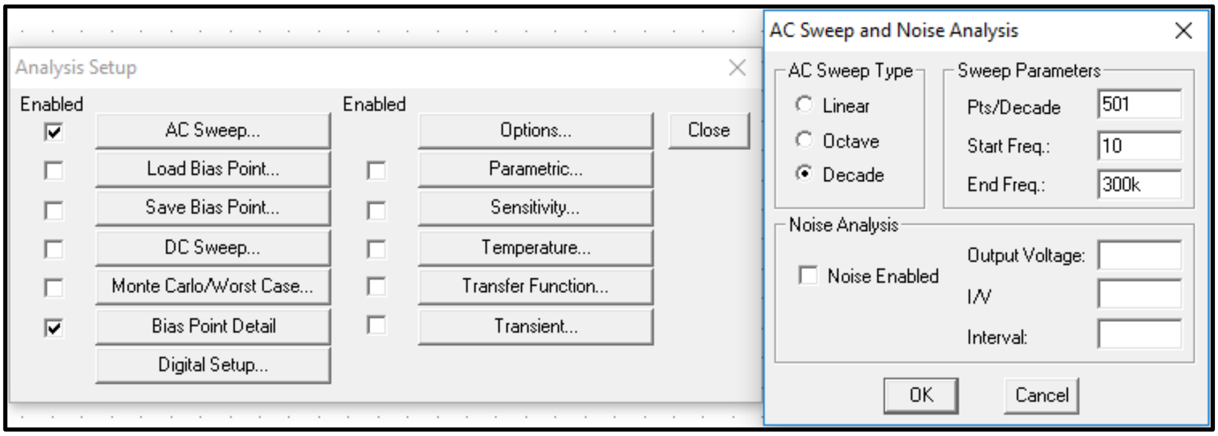


Figure.48

- Output window:

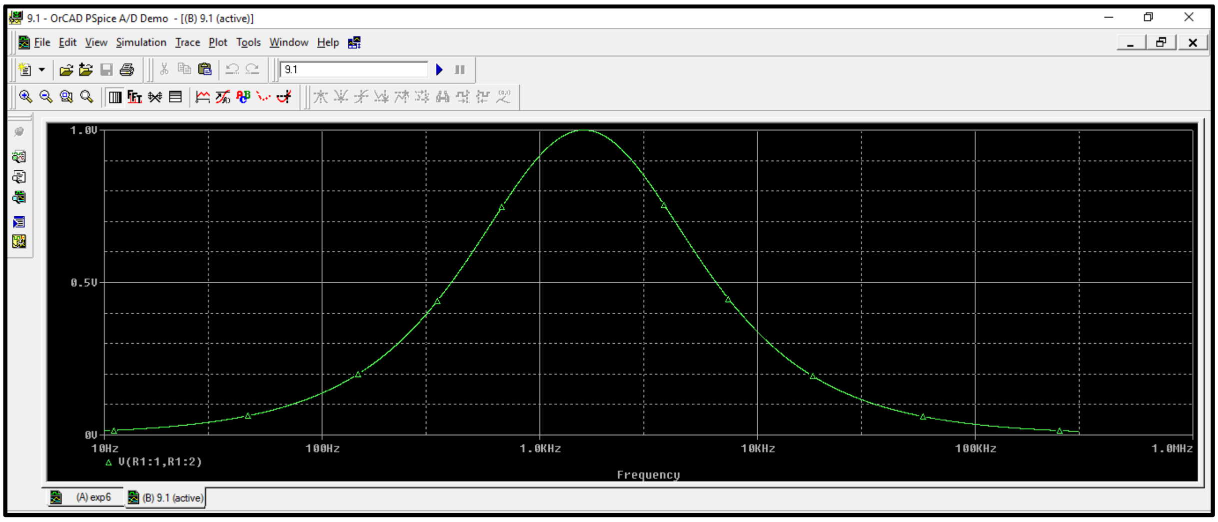


Figure.49

- To get the magnitude in dB:

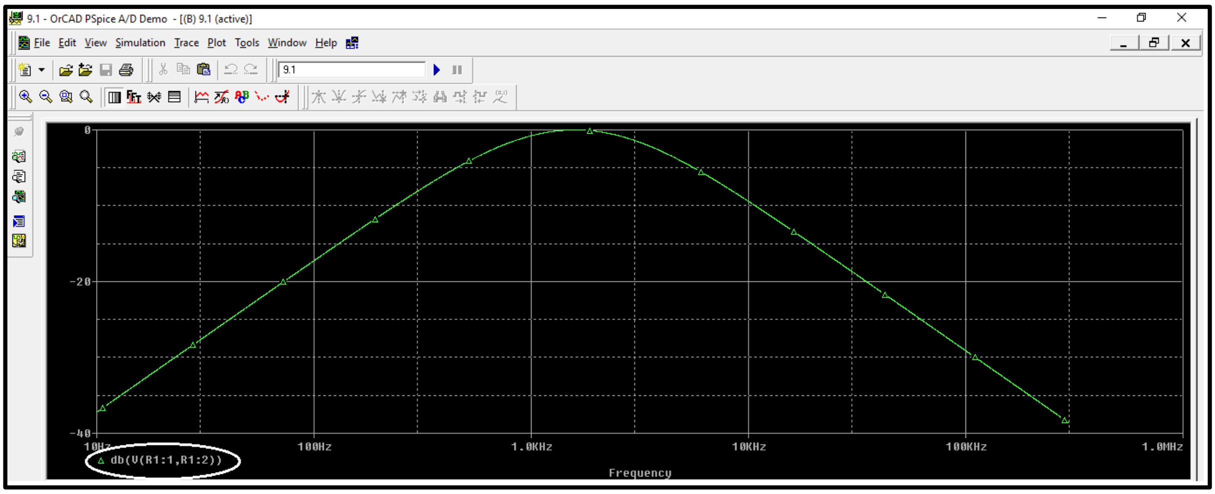


Figure.50

- To get the phase:

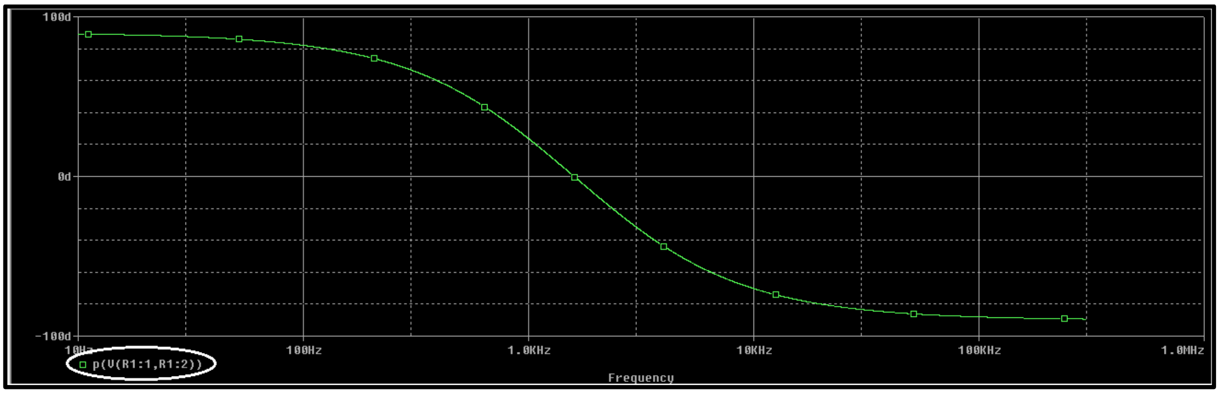


Figure.51

- Another example:

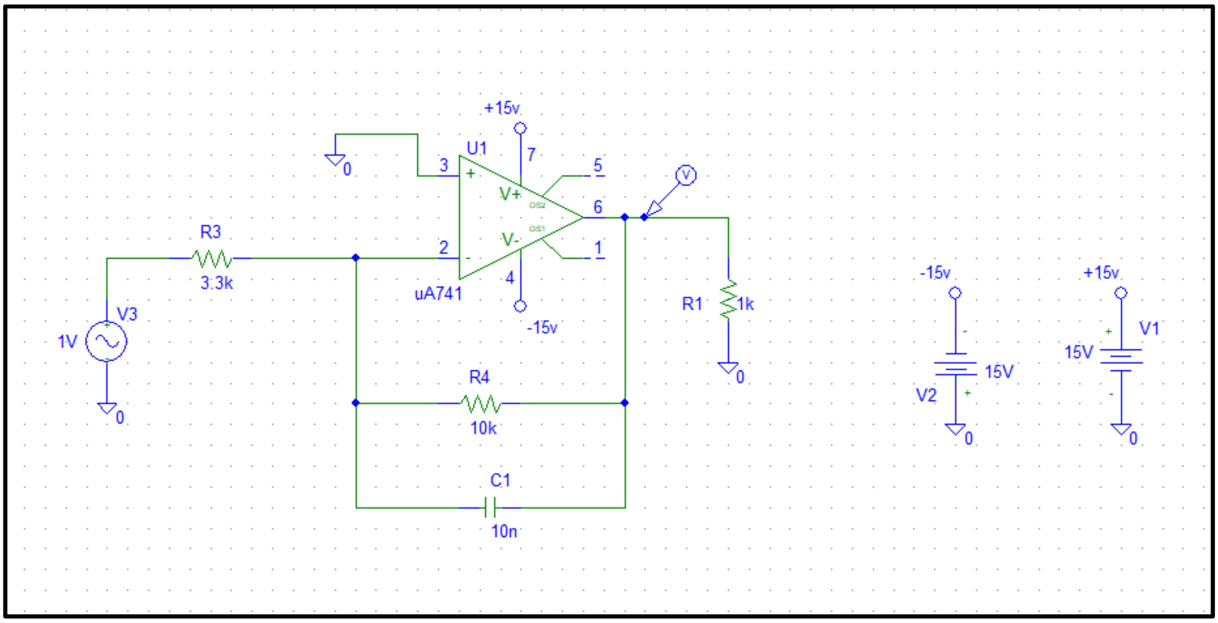


Figure.52

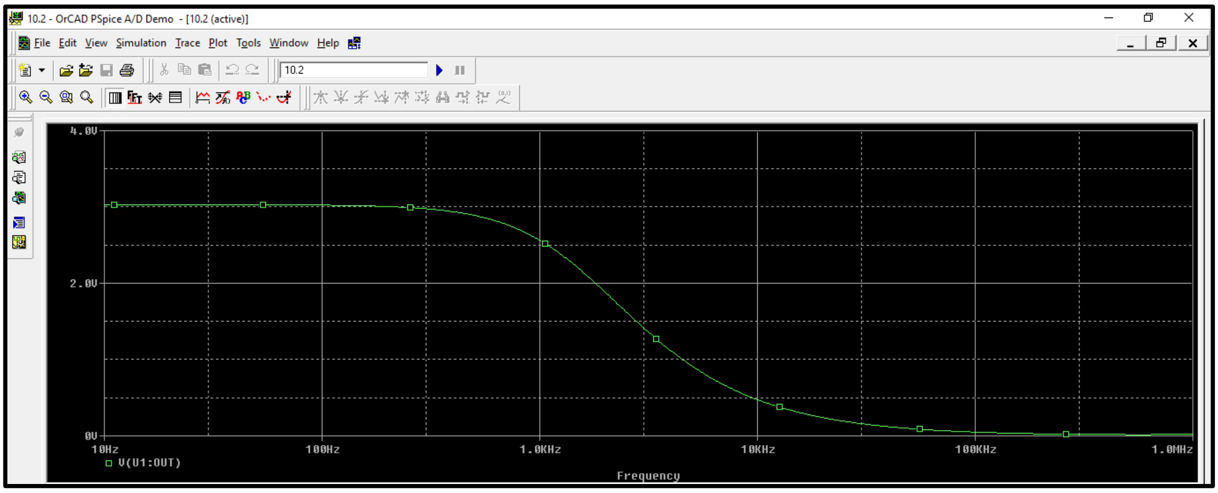


Figure.53

➢ Report writing guidelines

An experiment report is an important tool to communicate the experiment results and findings to others, it should be organized and written in clear a way. **Reports should be original and contain the basic required elements detailed below, copying from any source will result in a zero grade and proper academic punishment.**

The report must contain the following sections:

1. **Cover page**

   Cover page must contain: course number and name, section number, lab experiment name and number, author name, date performed, lab partner(s) name(s), and the instructor's name.

2. **Table of contents:**

   A list sections headings and page numbers.

3. **Abstract**

   This section provides a brief summary explaining the aim of the experiment, and the methods used. Use your own words, do not copy objectives from manual.

4. **Theory**

   This section should include any relevant theory along with mathematical formulas. The following should be considered when writing the theory section:

   - Avoid copying from lab manual, summarize the theory in your own words.
   - If you used information from any resources, state them in references.
   - Explain symbols in the mathematical formulas, and use graphical representation of formulas (curves) where applicable.

5. **Procedure**

   This section describes in detail the way the experiment was conducted. This is very important so that anyone who reads it should be able to re-produce the experiment and its results. In this section, what was measured and how it was measured should be provided.

6. **Data, Calculations, and Analysis of results:**

   - Your report must include a section for data aside from the sheet you used to record data in the lab, which must be attached at the end of your report. The data should be recorded in clear and readable fashion, be provided in tables where possible, and should have units.
   - Calculations should be performed to get the required quantities from measured ones, you must show **a detailed sample calculation showing any equations used**.

- You have to discuss all results comparing with what was theoretically expected from prelab exercise and explaining any differences. Discuss results **qualitatively** i.e. no need to state numeric results for each result as the experiments usually contains measurements of many quantities, it is enough to state the general behavior of results. The discussion should contain answers to the following questions:
  - Is the result acceptable?
  - What is the behavior of graphs/plots?
  - What are the possible sources of error?
- Any questions in the procedure section must be answered in this section when discussing results.

7. **Conclusions**

   Restate the main objectives and how or to what degree they were achieved. What principles were validated by the experiment? Were there any major experimental complications? How the result can be improved in the future if the experiment is repeated (optional)?

8. **References:**

   List any references you used in writing your report, examples on IEEE references formatting are given below:

   example of textbook:

   [1] J.W. Nilsson and S.A. Riedel, Electric Circuits. Reading, MA: Addison-Wesley, 5th ed., 1996, pp. 111-113.

   example of Internet web page:

   [2] Approximate material properties in isotropic materials. Milpitas, CA: Specialty Engineering Associates, Inc. web site: www.ultrasonic.com, downloaded Aug. 20, 2001.

General Format Guidelines

1. Use bold font with size (14 point) for titles (Times Roman or Arial), and 12 points elsewhere. Also, use 1.5 line spacing, and justify all paragraphs.
2. Place page numbers on all pages, bottom (though title page is page 1, don't display the number 1 on the title page).
3. Equations are centered and the equation numbers are right justified. The equation number is placed in ( ). See example.

$$F(s) = \int_{-\infty}^{\infty} e^{-st} f(t)dt \qquad (1)$$

4. Center all tables and include **a heading and caption with the appropriate table number above each table**. For example, "Table 1: Voltage measurements for Part 3"

5. Figures must be centered, and the **figure number and caption are centered beneath** the figure. For example, “Figure 1: Circuit schematic of Butterworth filter”.
6. All graphs must be done with a computer (e.g. Microsoft Excel or even Matlab).
7. All graphs require labels and units on the axes, and require a legend for more than one set of y-axis data.
8. Include a leading zero when a number’s magnitude is less than 1 (use 0.83 instead of writing .83). Include a space between any number and an associated unit (i.e., 3.4 mA, not 3.4mA).
9. Use your word processor for Greek symbols for common engineering quantities as β, π, φ, ω, and Ω. Also use Microsoft word to make any necessary superscripts and subscripts. (Use $V = 10R^2$ instead of V = 10R^2).
10. Verb Tense:
    - Use past tense when describing a procedure that was implemented in order to produce your results. For example, “After constructing the circuit of Fig. 1, power was applied.”
    - Use present tense when analyzing the results and making conclusions. For example, “The data shows that the efficiency of the process is 92%.” Also, when making reference to a figure or data within the report, use present tense. For example, “The test setup is shown in Fig. 1.”

Report Grading Guidelines

The laboratory report grade will depend on the following:

- Is the report written well and in good English?
- Does the theory contain the necessary illustrative figures? Are these figures meaningful and clear?
- Do data and calculated quantities have correct units?
- Are calculations made correctly?
- Does the report contain all information to reproduce the experiment?
- Is the result correct and consistent with what is expected?
- Are the graphs complete, correct and properly labeled (title, axis labels)?
- Are all elements of the report included?
- Is the report submitted on time?

Experiment 2

Introduction to measurements and lab devices

- **Objectives:**

1. In this experiment, we shall study how to use the digital multi-meter, power supply, function wave generator, and the oscilloscope.
2. To make some measurements in the lab.
3. Measuring Phase between two signals.

- **Equipment Required:**

1. Board, resistors, capacitors and wires.
2. Digital Multi-meter GDM-8135.
3. Oscilloscope TDS 2002B.
4. Power Supply Unit TK286 [0-20V] Variable DC.
5. Function Wave Generator GFG-8215a.

- **Introduction:**

- Connection Board:

It is a board where connections are done. On the same board, there are the sources that may be used in the experiments such as the DC source, AC source, Function Generator and the 3 Phase Generator. Each big hole is surrounded by four other small holes form one node (short circuit). For example, to connect two big adjacent holes together as one node, a jumper must be placed between them in the two small adjacent holes, the board is shown in Figure 2.1.

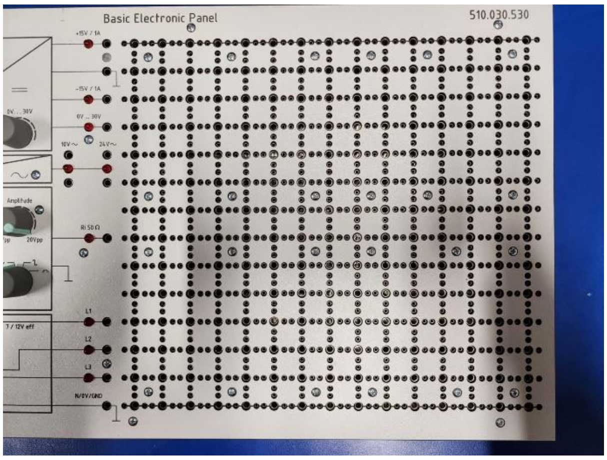


Figure 2.1: board.

- **Power Supplies:**

  Figure 2.2 shows the four types of sources that will be used. The first block is the DC Power Supply. It generates constant +15V or -15V, or can be used as variable DC source from 0-30V. The function generator generates the four types of signals (sinusoidal, triangle, square and pulse signals). The AC – Source 3p is the 3 phase generator that generates 3 phase voltages with $V_p = 7V_{rms}$ and $V_{LL} = 12V$.

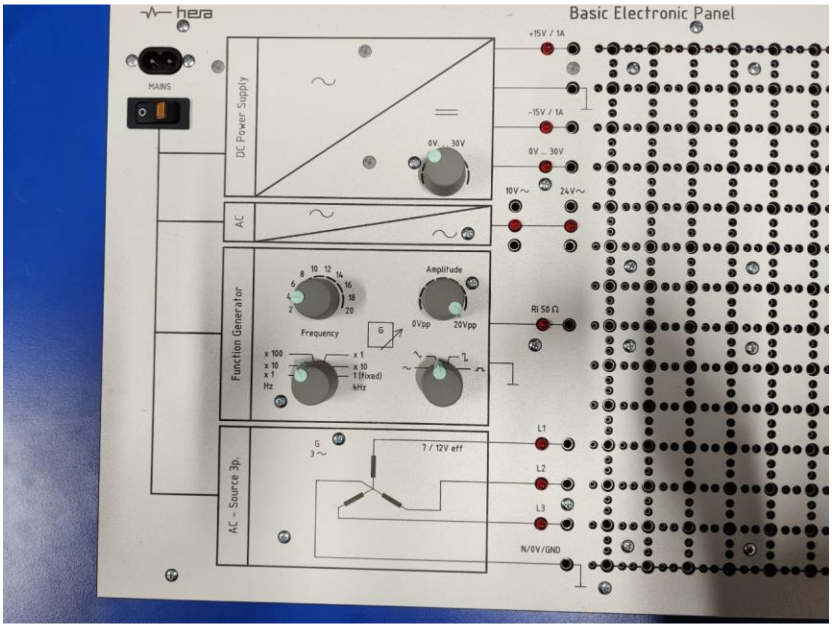


Figure 2.2: Power supplies.

- **Digital Multimeter:**

  Digital Multimeters are used to measure resistance, voltage and current. This makes the multimeter one of the most important instruments. We use the GDM-8135 in the lab as shown in Figure 2.3. This is typical of such instruments. The power switch is located on the lower right of the front panel, with green color.

Figure 2.3: Digital Multimeter GDM-8135, connected as ammeter.

- The multimeter has three buttons indicated above that determine which type of measurement is being made. For example, if you want to measure resistance, you have to select the resistance mode button. Likewise, current and voltage modes can easily be set.
- Next, you need to attach cables to the multimeter ports. You may use cable with a connector as shown in Figure 2.4.

Figure 2.4: Cable with a connector.

- A black cable inserted in the black port called 'common' or COM. Another red cable is inserted in one of the three red ports depending on the quantity to be measured.
- **To measure voltage or resistance**, the red cable is inserted in the upper red port which is marked (V-Ω).

Upper red Port: Voltage / resistance
Black Port: common
Lower red Ports: Current

- **To measure current**, the red cable is inserted in one of the two lower red ports which are marked 2A max, 20A max. The First port is fused and can measure currents up to 20 A. The second port is used to measure small currents of 2A or less and is fused.
- In circuit lab you will have two multimeters, one is configured to measure currents, and the other is configured to measure voltages and resistances. So, cables are connected to the appropriate ports, and you do not need to reconnect them each time you want to measure a different quantity.

- Measuring resistance:

- Use the multimeter configured to measure voltage and resistance (the one with the red cable attached to the (V-Ω) port). To measure resistance, you need to set the multimeter to the resistance mode by pushing the **'KΩ'** button.

- You must connect the Ohmmeter cables **in parallel** with the element in the circuit for which you want to measure the resistance, as shown in figure 1.5.

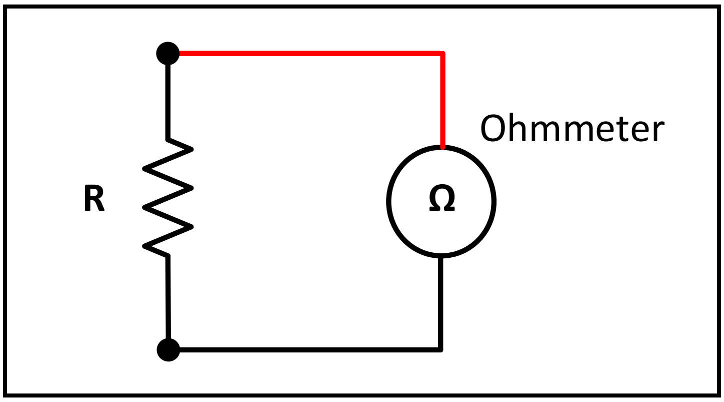


Figure 2.5: Multimeter connection to measure resistance.

- An Ohmmeter usually employs an internal battery across the resistance you are trying to measure. The battery drives a current into the resistor, which is measured by a current sensor. The value of the resistance is the calculated by dividing the battery voltage by the current. **Hence, be careful not to connect any external power supply to the resistor you are trying to measure because an extra current can damage the ohmmeter.** Also isolate the resistor you want to measure from the circuit, since the multimeter calculates the overall Thevenin resistance connected to its terminals**.**
- **Selecting suitable range:** You will see numbers appear on the multimeter display.
  - If these numbers are all zero, then the range setting on the multimeter is too high and you should lower it. The range of measurement is selected by the buttons below the multimeter display.
  - If only the number 1 appears in the left most position, then the range setting is too low and the resistance value is 'out of range'. In this case, increase the range setting.
  - If the display shows actual values, then this is the value of the resistor. Try out different range settings to see which one gives you the most precision.
  - Remember to check your range setting before reading the display e.g. if the range setting is 20 KΩ the display unit is kΩ, if it is 200 Ω the display unit is Ω.

- Measuring voltage:

- Use the multimeter configured to measure voltage and resistance (the one with the red cable attached to the (V-Ω) port). To measure voltage, set the multimeter to the voltage mode by pushing the **'V'** button.
- A voltmeter is connected **in parallel** with the circuit element where the voltage measurement is desired as shown in figure 2.6. Since the voltage across parallel elements

is the same, the voltage measured by the meter will be the same as the voltage across the element to which the meter is connected.

- The internal impedance of the voltmeter is very large, to avoid drawing extra current, thus disturbing the voltage it is trying to measure.

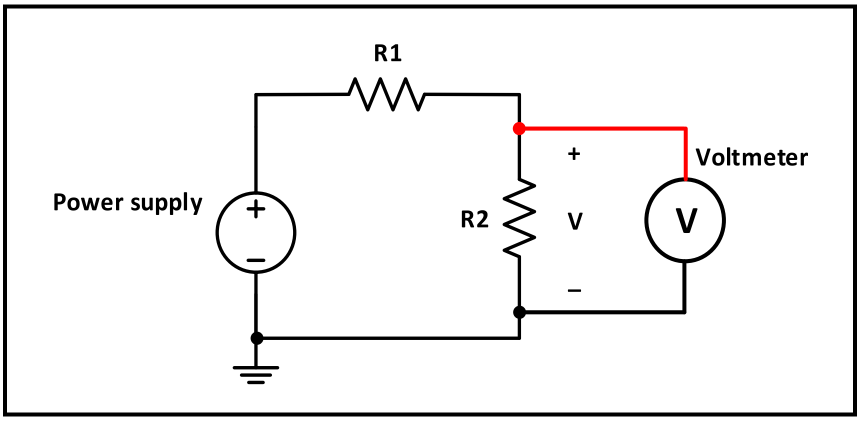


Figure 2.6: Multimeter connection to measure voltage.

- There are two different types of voltage measurements: AC and DC. The button to the left of the voltage mode button toggles the multimeter from AC to DC mode.
- When measuring voltage in DC mode, the multimeter measures the time average of the signal. In AC mode, the multimeter measures the R.M.S (root mean square) of the signal.
- **A common mistake** is to measure AC voltage in DC mode, the reading will be zero since the average value of AC voltage is zero.
- The red cable is connected to the higher voltage, while the black cable is connected to the lower voltage to produce positive measurement in DC mode.

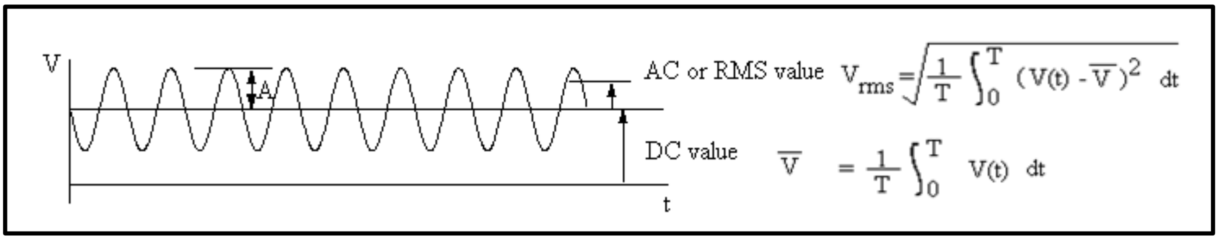


Figure 2.7: Sinusoidal signal

- **Selecting suitable range** for voltage measurement is done in the same way for resistance measurement. Note that the unit of measurement is (mV) or (V) depending on the selected range.
- **To protect the instrument**, you should always select the highest range first when making measurements of unknown voltages, then lower it till you find the suitable range.

- **Measuring current:**

- Use the multimeter configured to measure current (the one with the red cable attached to the (2A) port). To measure current, set the multimeter to the current mode by pushing the **'mA'** button.
- An ammeter is connected **in series** with the element in the circuit through which you want to measure the current. This means, for example, if you want to measure the current through a cable between two devices, you will have to disconnect that cable, and reconnect it through the meter, as shown in figure 2.8.
- Since the current in each element of a series circuit is the same, the current flow through the meter will be the same as the current flow to the element of interest.
- The ammeter has a very small internal resistance so it does not disturb the current it is trying to measure. Due to this fact, however**, if it is connected by accident in parallel with the circuit element, a large amount of current will flow through it, thus damaging it. That is why ammeters are usually protected by a current limiting fuse.**

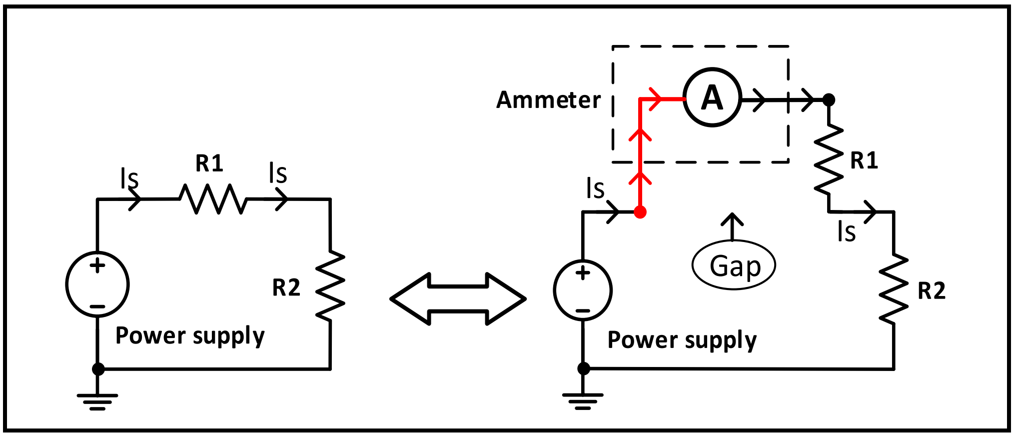


Figure 2.8: Multimeter connection to measure current.

- Current measurement has AC and DC modes similar to voltage measurement.
- The red cable is connected such that the current flows into it and leaves out the black cable to produce positive measurement in DC mode.
- **Selecting suitable range** for current measurement is done in the same way for resistance measurement. Note that the unit of measurement is (mA) or (µA) depending on the selected range.
- **To protect the instrument**, you should always select the highest range first when making measurements of unknown currents, then lower it till you find the suitable range.

- **Function Wave Generator:**

The function generator can be used to produce three different kinds of signals as a function of time, square wave, triangular wave and sine wave.

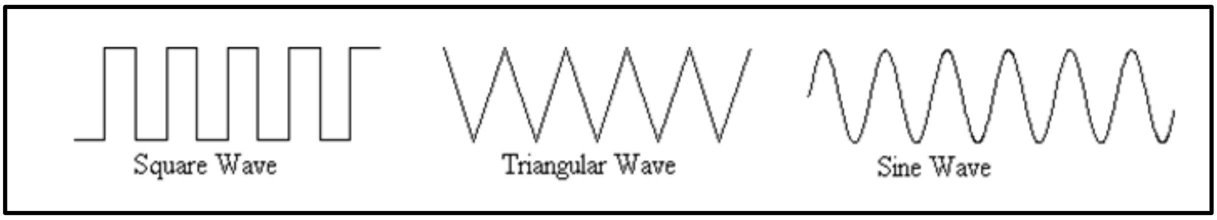


Figure 2.9: Waveforms

The function buttons determine the type of signal the instrument produces. With the sine-wave button pushed in, a sine wave will be produced, for example. The function generator also has range settings to determine the frequency range of the signal. The amplitude of the output signal is controlled using “AMPL” knob. The Function Wave Generator used in lab is the GFG-8215a as shown in Figure 1.10

Figure 2.10: Function Wave Generator GFG-8215a

- **Oscilloscope:**

The oscilloscope is a device designed to display a voltage signal in visual form. It can used for some quantitative measurements, such as voltage amplitude and frequency, it also can be used to compare two separate signals and estimate their relative characteristics. The scope used in lab is the Tektronix TDS 2002B as shown in Figure 2.11.

- **Basic Set Up:**

The oscilloscope is turned on using the power button indicated in figure 2.11. After a few seconds a horizontal line should appear on the screen. The line, called a 'trace' is a plot of voltage (on the vertical scale) against time (on the horizontal scale).

- **Note:** Autoset button used to get the signal direct without any setting on oscilloscope.

Figure 2.11: Oscilloscope Tektronix TDS 2002B

The TDS 2002B is a digital oscilloscope that functions very much like a computer. Most of the settings can be changed through various menus accessible from the top control panel:

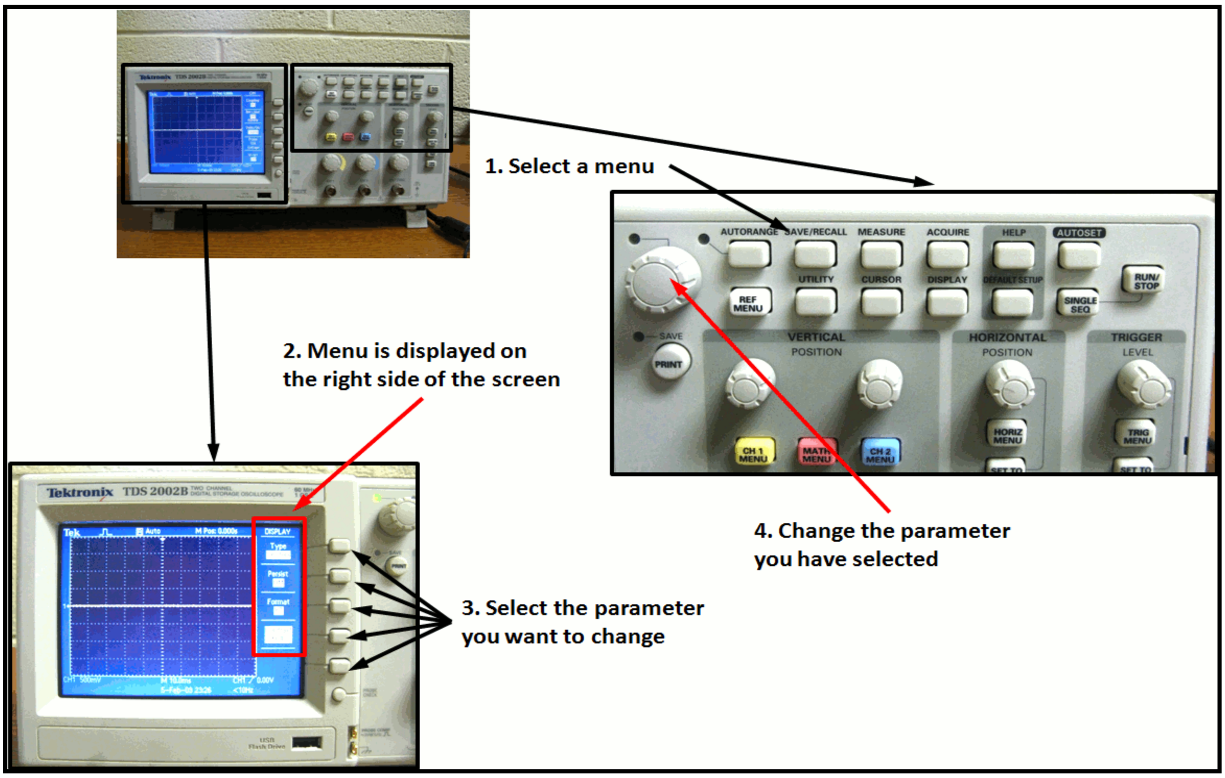


Figure 2.12: Oscilloscope menu

Note that the oscilloscope screen has graduations (or 'divisions'). This is to enable you to make quantitative measurements of signals displayed. You can also try the ***MEASURE*** and ***CURSOR*** menus. The ***MEASURE*** menu allows you to measure various quantities (frequency, mean, peak-to-peak, etc...) for one or two channels. The ***CURSOR*** menu uses two lines to measure any feature on the vertical or horizontal axes. Try these out for yourself.

In the section of the front panel labeled "VERTICAL" set the **CH 1 MENU** and **CH 2 MENU**

In the HORIZONTAL section set the SEC/DIV knob to 5ms/div (as you turn the knob, you should see the value at the bottom of the screen (just above the date) change). You should see a trace on the screen. Adjust the intensity and focus if needed. Center the line using the position knobs in the HORIZONTAL and VERTICAL sections.

- **Connecting Signals:**

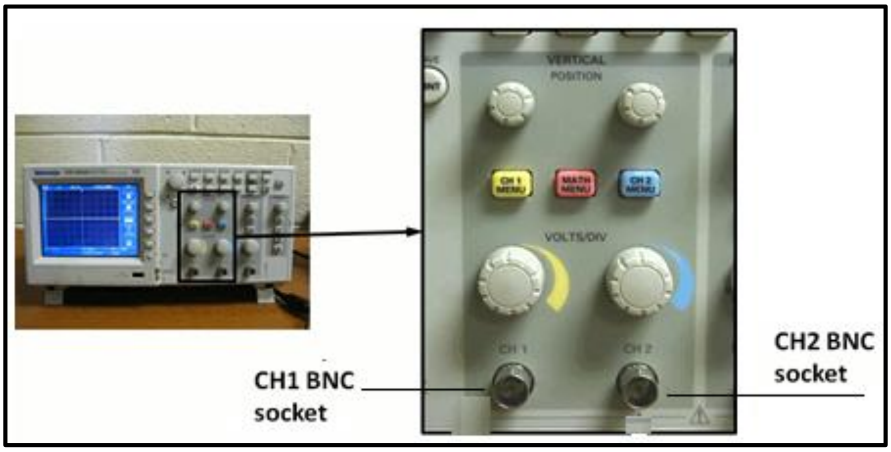


Figure 2.13: connecting signals to oscilloscope

Signals are connected to the scope through one of two BNC sockets, located as shown above. These accept cables with connectors of the type shown below.

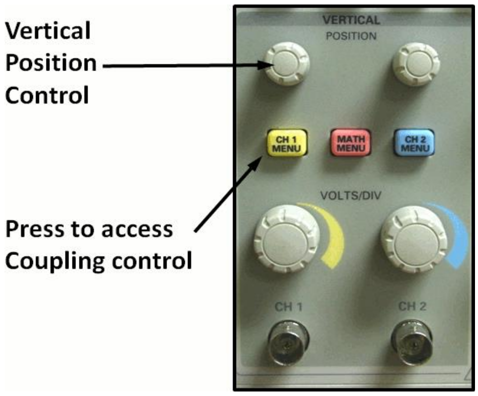
BNC connector

There are two sockets, because the scope can simultaneously display up to two signals. The signals and connectors are identified as CH 1 and CH 2, for channels 1 and 2. Note that the signal is actually provided to the scope through the central pin of the BNC sockets. The outer shield is a ground connection (not just a common) **so you must be a little careful what you connect to it. (E.g. if you set the power supply to provide 5V relative to ground and then connect the 5V to the BNC shield the resulting mismatch might cause problems the scope.)** When you have made your connections, you need to tell the oscilloscope what to display using the **CH 1 MENU** and **CH 2 MENU** buttons. Pressing these buttons will toggle the display of their respective channel. Channel 1 signal appears as a yellow trace, while Channel 2 signal is blue. It is therefore possible to display the signals from both channels at the same time.

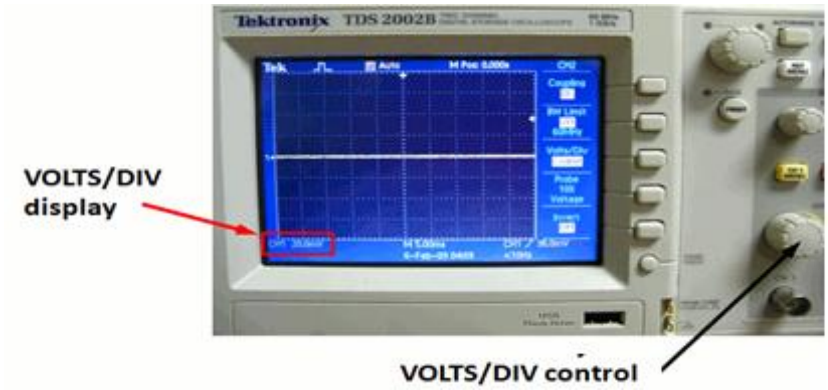
The **MATH MENU** button can be used to display an arithmetic combination of two signals. You can select from addition, subtraction, multiplication, addition or Fourier Transform (FFT). The resulting signal is plotted in red.

- **Setting the Vertical Scale:**

Scopes give you the same control over displaying a signal as you would have over plotting it on a piece of graph paper. You can independently control the origin and the scale of the voltage displayed on the vertical axis. There are duplicate controls for the two channels.



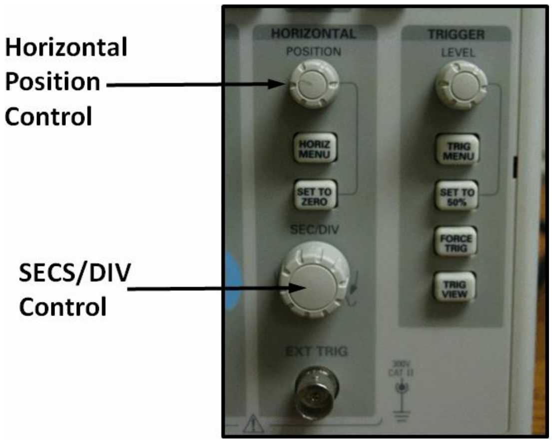
The vertical 'POSITION' controls are used to set the origin. Turning the POSITION control for CH1 for examples simply moves the signal up and down on the screen (i.e. the origin is moving up and down). If your objective is to make an absolute voltage measurement (rather than just look at the form of the signal) you will want to know quantitatively where the origin you've set is. You can do this by selecting GND coupling to plot zero volts on the screen.



For controlling the scale of the voltage axis there is a VOLTS/DIV control. This sets the voltage represented by each of the large vertical divisions on the screen. The actual value set is indicated by the number in their lower left corner of the screen and may be varied from 5 mV per division to 5 V per division.

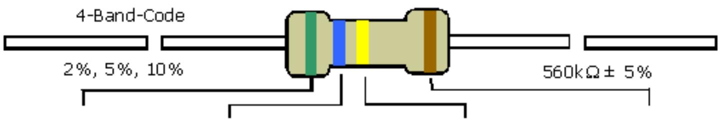
If you need finer resolution in the increments of VOLTS/DIV you can press the **CH 1 MENU** and change *Volts/Div* from Coarse to Fine. Note that it is perfectly possible to set the POSITION and/or VOLTS/DIV control so that the signal you want to look at, or even the voltage origin, is off the screen so that you see no trace. If you think this has happened the best way to get your signal back is to increase the VOLTS/DIV setting, and then to rotate POSITION knob until the trace appears.

- Setting the Horizontal Scale:



The controls for the horizontal (time) axis are located as shown in the picture. The same horizontal scale is used for both channels, so there is only one set of controls. For controlling the time origin (i.e. for moving the signal horizontally across the screen) there is a POSITION control. For controlling the scale of the time axis there is a knob labeled 'SEC/DIV'. This sets the time represented by each of the large horizontal divisions on the screen. The actual value set is indicated at the bottom of the screen (just above the date) and may be varied from 5 nanoseconds per division to 50 seconds per division.

- Resistor Color Code:



COLOR	1st BAND	2nd BAND	3rd BAND	MULTIPLIER	TOLERANCE
Black	0	0	0	1Ω	
Brown	1	1	1	10Ω	± 1% (F)
Red	2	2	2	100Ω	± 2% (G)
Orange	3	3	3	1KΩ	
Yellow	4	4	4	10KΩ	
Green	5	5	5	100KΩ	±0.5% (D)
Blue	6	6	6	1MΩ	±0.25% (C)
Violet	7	7	7	10MΩ	±0.10% (B)
Grey	8	8	8		±0.05%
White	9	9	9		
Gold				0.1	± 5% (J)
Silver				0.01	± 10% (K)

➢ **Procedure**

Part A: Measuring resistance

1. Use digital multi meter (DMM) to measure resistances you have been given, record results.

Part B: Measuring voltage

1. Connect the circuit shown in figure 2.14, set the input power supply to 10V, measure the voltage across each resistor, record and discuss results.

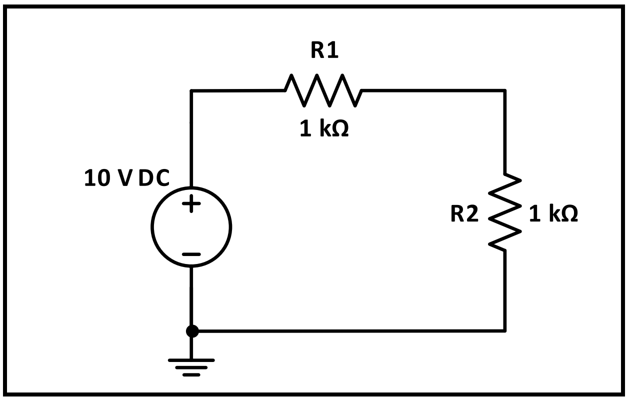


Figure 2.14

Part C: Measuring current

1. For the same circuit of figure.2.14 measure the current through each resistor, record and discuss results.

Part D: Using oscilloscope and function wave generator

1. Connect the circuit shown in Figure 2.15, set the input voltage to sinusoidal voltage at 8 V_{PP} and 1 kHz (you will need to connect channel 1 of the oscilloscope to the input voltage).

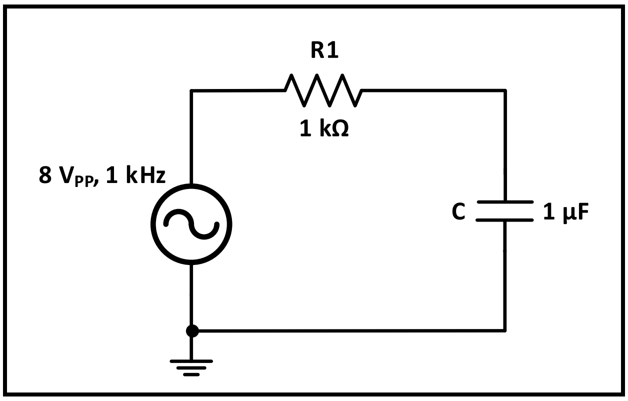


Figure 2.15

2. Connect channel 2 of the oscilloscope to the capacitor, use measure menu to view characteristics of the voltage across capacitor.
3. Use DMM to measure the current in the circuit, and the voltage across the resistor and the capacitor.

Experiment 3

Simple Resistive Circuits

➢ Objectives:

1. To verify the KVL and KCL.
2. To verify the voltage divider rule.
3. To verify the current divider rule.
4. To examine the effects of short and open circuited resistor in electrical circuits.

➢ Equipment Required:

1. Digital Multimeter.
2. Power Supply.
3. Board, wires, and resistances.

➢ Introduction:

Kirchhoff's Voltage Law: states that the algebraic sum of all the voltages around any closed path (loop or mesh) is zero.

Applying Kirchhoff's voltage law to the first and the second loops in the circuit shown in Figure 3.1 yields:

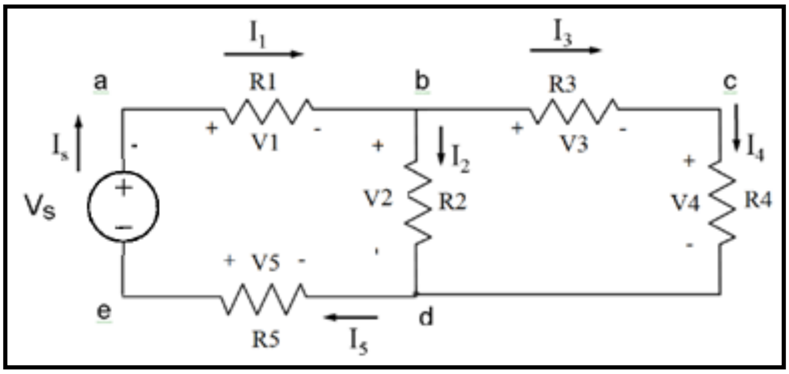


Figure 3.1

Loop 1: -Vs +V1 +V2 - V5 = 0 ……………………………………………………………1- a

Loop 2: -V2 +V3 +V4 = 0 …………………………………………………………………1- b

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

Applying Kirchhoff's current law to the first four nodes in the circuit shown in Figure 2.1 yields the following equations;

Node a: -Is + I1 = 0 …………………………………………………………………..(2-a)

Node b: -I1 + I2 + I3 = 0 ……………………………………………………………(2-b)

Node c: -I3 + I4 = 0 …………………………………………………………………(2-c)

Node d: -I2 - I4 + I5 = 0 ……………………………………………………………(2-d)

The Voltage-Divider and Current-Divider Circuits

At times—especially in electronic circuits—developing more than one voltage level from a single voltage supply is necessary. One way of doing this is by using a **voltage-divider circuit,** such as the one in Figure 3.2.

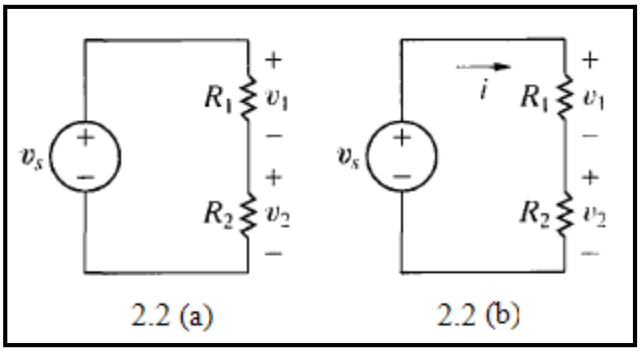


Figure 3.2 (a) A voltage-divider circuit and (b) the voltage-divider circuit with current i indicated.

We analyze this circuit by directly applying Ohm's law and Kirchhoffs laws. To aid the analysis, we introduce the current i as shown in Figure 2.2 (b). From Kirchhoffs current law, R1 and R2 carry the same current. Applying Kirchhoffs voltage law around the closed loop yields

vs = iR1 + iR2……………………………………………………………………...(3)

$$i = \frac{v_s}{R_1 + R_2}$$ ……………………………………………………….……...(4)

Now we can use Ohm's law to calculate v1 and v2:

$$v_1 = iR_1 = v_s\frac{R_1}{R_1 + R_2}$$ …………………………………….………………...(5)

$$v_2 = iR_2 = v_s\frac{R_2}{R_1 + R_2}.$$ …………………………………………………....(6)

Equations (5) and (6) show that v1 and v2 are fractions of vs. Each fraction is the ratio of the resistance across which the divided voltage is defined to the sum of the two resistances.

Because this ratio is always less than 1.0, the divided voltages v1 and v2 are always less than the source voltage vs.

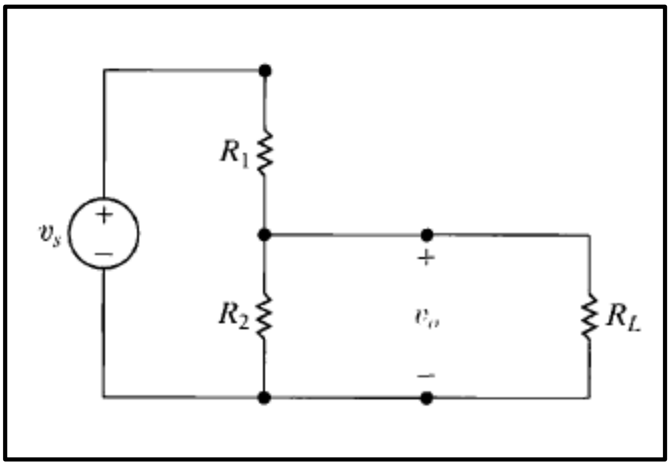


Figure 3.3: A voltage divider connected to a load R_L

If you desire a particular value of v2, and vs is specified, an infinite number of combinations of R1 and R2 yield the proper ratio. For example, suppose that vs equals 15 V and v2 is to be 5 V. Then v2/vs =1|3 and, from Eq. 2, we find that this ratio is satisfied whenever R2 = 0.5 R1 Other factors that may enter into the selection of R1 and hence R2, include the power losses that occur in dividing the source voltage and the effects of connecting the voltage-divider circuit to other circuit components. Consider connecting a resistor RL in parallel with R2, as shown in Figure 2.3. The resistor RL acts as a load on the voltage-divider circuit. A **load** on any circuit consists of one or more circuit elements that draw power from the circuit. With the load RL connected, the expression for the output voltage becomes

$$v_o = \frac{R_{eq}}{R_1 + R_{eq}} v_s, \quad \text{……………………………………………………………...….(7)}$$

$$R_{eq} = \frac{R_2 R_L}{R_2 + R_L} \quad \text{………………………………………………………………...….(8)}$$

$$v_o = \frac{R_2}{R_1[1 + (R_2/R_L)] + R_2} v_s. \quad \text{…………………………………………….....(9)}$$

Note that Eq.(9) reduces to Eq.(6) as RL—» ∞ , the voltage ratio vo/vs is essentially undisturbed by the addition of the load on the divider. Another characteristic of the voltage-divider circuit of interest is the sensitivity of the divider to the tolerances of the resistors. By tolerance we mean a range of possible values. The resistances of commercially available resistors always vary within some percentage of their stated value.

➢ Prelab:

****(Refer to procedure section for circuits figures)**

When asked to find voltages and currents using PSPICE, you have to attach a clear picture of your PSPICE circuit in your pre-lab document with voltages and current shown on it, **in all experiments of this manual.**

- ### Part A: Kirchhoff's Laws

1. Simulate the circuit of Figure 3.4 using PSPICE to find the voltage across and the current through each resistor.

- ### Part B: Voltage Divider

1. Simulate the circuit of Figure 3.5 using PSPICE to find the voltages V_{1-2}, V_{2-3}, and V_{3-0}.

2. For the circuit of Figure 3.6: R_p is a 10 kΩ potentiometer (three-terminal resistor), Calculate R_{AB} and R_{BC} so that $V_o = 3$ V.

3. For the circuit shown in Figure 3.7, use PSPICE to find the value of V_O for $R_L = 1$ kΩ, 10 kΩ, 100 kΩ, 500 kΩ.

- ### Part C: Current Divider

1. Simulate the circuit of Figure 3.8 using PSPICE to find the value of current in each resistor in the circuit.

- ### Part D: Short-and-Open Circuited Resistor in Series-Parallel Circuits

1. Simulate the circuit of Figure 3.9 using PSPICE to find the voltage across and the current through each resistor.

➢ **Procedure:**

Part A: Kirchhoff's Laws

1. Connect the circuit of Figure 3.4, set V_{in} to 10 V.

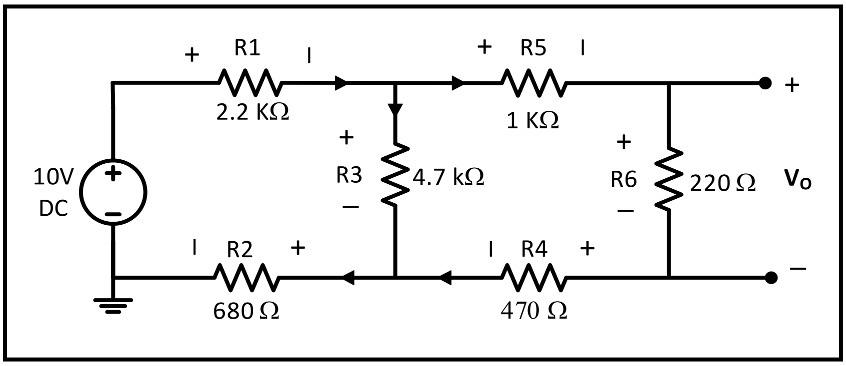


Figure 3.4

2. Measure the current though and the **voltage drop across** each resistor using Digital Multimeter (DMM), record results in Table 3.1.

Question 1: Using measured values of I and V, verify KCL on the upper middle node, verify KVL on the left loop.

Part B: Voltage Divider

1. **For the circuit of Figure 3.5:**

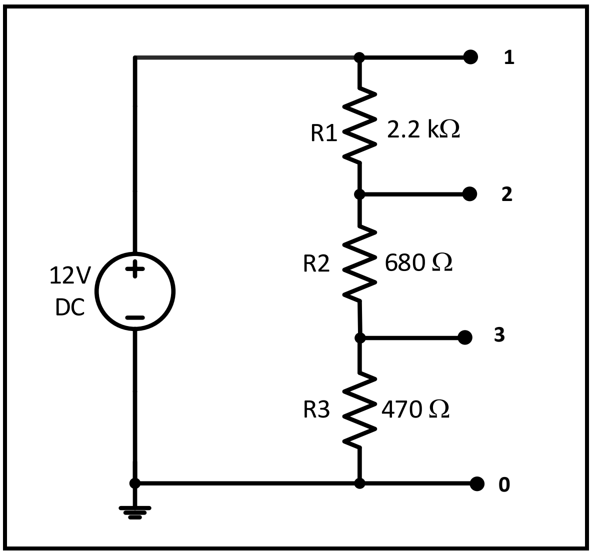


Figure 3.5

a. Connect the circuit, set the power supply output to 12 V.

b. Measure voltages $\mathbf{V_{1\text{-}2}}$, $\mathbf{V_{2\text{-}3}}$, **and** $\mathbf{V_{3\text{-}0}}$, record results in Table 3.2.

Question 2: From measured values, around which resistor did the largest and smallest voltage drops occurred? Explain why?

Question 3: Using measured values of V, verify that ($V_{1-2} + V_{2-3} + V_{3-0} = 12V$).

2. **For the circuit of Figure 3.6:**

   a. Connect the circuit with the power supply set to 12 V.

   b. Change the pot. till $V_o = 3V$, measure R_{AB}, R_{BC} and record results in Table 3.3.

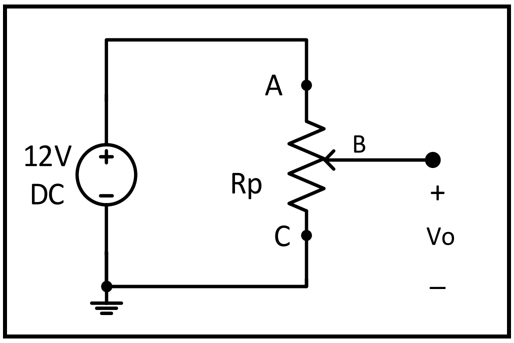


Figure 3.6

3. **Connect the circuit of Figure 3.7.**

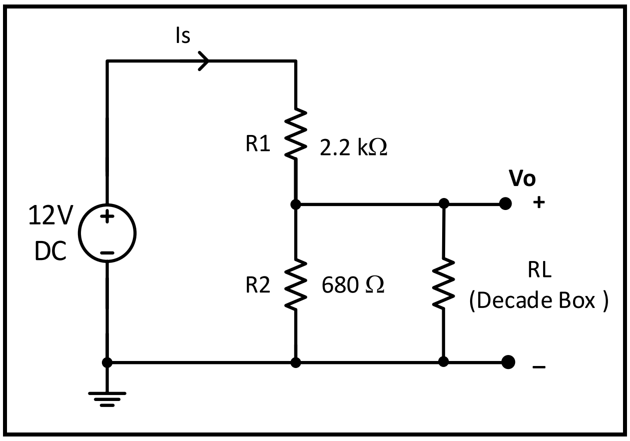


Figure 3.7

   a. Measure V_o for R_L = 1 kΩ, 10 kΩ, 100 kΩ, 500 kΩ and ∞, record results in Table 3.4.

Question 4: Using measured values, plot V_O vs. R_L. Explain the change in V_O as R_L is changed.

Part C: Current Divider

1. Connect the circuit shown in Figure 3.8, set Vin to 15 V.

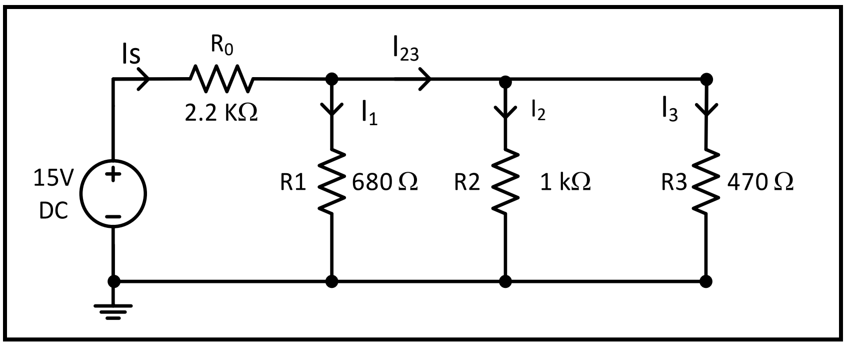


Figure 3.8

2. Measure the values of I_s, I_1, I_{23}, I_2, and I_3, record results in Table 3.5.

Question 5: From measured values of I, through which resistor did the largest and smallest currents passed? Explain why?

Question 6: Using measured values of I, verify that ($I_s = I_1 + I_2 + I_3$).

Part D: Short-and-Open Circuited Resistor in Series-Parallel Circuits

1. Connect the circuit of Figure 3.9, set Vin to 12 V.

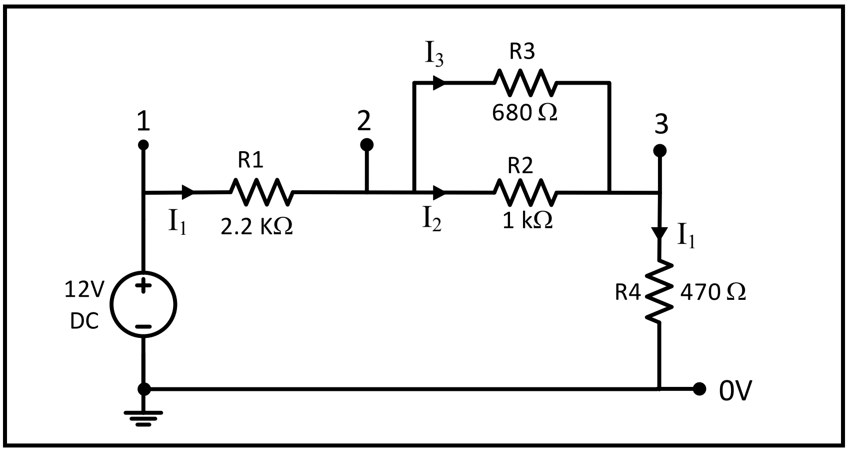


Figure 3.9

2. Measure voltages $V_{1\text{-}2}$, $V_{2\text{-}3}$, and $V_{3\text{-}0}$, and currents I_1, I_2, and I_3, record results in Table 3.6.
3. **Replace R_1 by a short circuit.** Measure the resulting voltages $V_{1\text{-}2}$, $V_{2\text{-}3}$, and $V_{3\text{-}0}$, and currents I_1, I_2, and I_3, record results Table 3.6.
4. Reinsert R_1 back in the circuit, replace R_2 by short circuit, repeat same measurements in step (3), and record results in Table 3.6.
5. Return the circuit to the original state of figure 3.9, consider now **removing R_1**. Measure the resulting voltages $V_{1\text{-}2}$, $V_{2\text{-}3}$, and $V_{3\text{-}0}$, and current I_1 if this were to occur. Record results in Table 3.7.
6. Reinsert R_1 back in the circuit, remove R_2, repeat same measurements in step (5), and record results in Table 3.7.

Question 7: What is the value of the short circuit voltage? Is there any current in the circuit when a resistor is open circuit?

Before Leaving the Laboratory

Be sure the following is completed before you leave the laboratory.

(a) Check to be sure that you have all the required measured values.

(b) Have the laboratory instructor check your laboratory readings.

(c) Restore your laboratory station (equipment and chairs) to the condition they were in when you arrived.

Thank you for your cooperation.

Experiment 4
Network Theorems

➢ Objectives:

1. To verify the proportionality and superposition theorems.
2. To verify Thevenin's theorem.
3. To verify that a network can be replaced by an equivalentΔ-Y network.
4. To verify the reciprocity theorem of single-source DC network.

➢ Equipment Required:

1. Digital Multimeter.
2. Power Supply.
3. Board, wires, and resistances.

➢ Introduction:

Solving for currents and voltages in multi-loop electric circuits can be quite complicated, particularly for AC circuits. The Kirchhoff's voltage law and Kirchhoff's current law always apply but using them may lead to long systems of equations. Certain theorems help with network analysis:

Superposition Theorem:

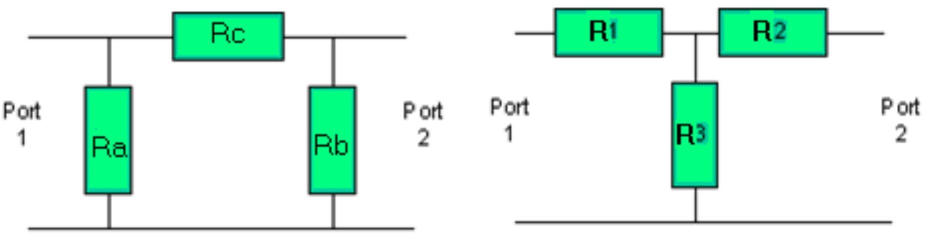
For multiple-source circuits, the circuit respond from each source can be calculated individually, and the results added (superimposed).

Thevenin's Theorem:

Combinations of voltage sources and resistors with two terminals can be replaced by a single ideal voltage source and a series resistor.

Delta-to-Wye (Pi-to-Tee) Equivalent Circuits:

Any pi network can be transformed to an equivalent T network. This is also known as the Wye-Delta transformation, which is the terminology used in power distribution and electrical engineering. The pi is equivalent to the Delta and the T is equivalent to the Wye (or Star) form



Pi Network T Network

The impedances of the pi network (Ra, Rb, and Rc) can be found from the impedances of the T network with the following equations:

Ra = ((R1*R2)+(R1*R3)+(R2*R3)) / R2

Rb = ((R1*R2)+(R1*R3)+(R2*R3)) / R1

Rc = ((R1*R2)+(R1*R3)+(R2*R3)) / R3

Note the common numerator in all these expressions which can prove useful in reducing the amount of computation necessary.

The impedances of the T network (R1, R2, R3) can be found from the impedances of the equivalent pi network with the following equations:

R1 = (Ra * Rc) / (Ra + Rb + Rc)

R2 = (Rb * Rc) / (Ra + Rb + Rc)

R3 = (Ra * Rb) / (Ra + Rb + Rc)

Note the common denominator in these expressions.

➢ **Prelab**:

****(Refer to procedure section for circuits figures)**

Part A: Proportionality

1. For the circuit of Figure 4.1, use PSPICE to generate a plot of (V_O) (use differential voltage marker), for a V_{in} sweep from 0 to 15 V in a 1.5V step, use cursors to mark data point at V_{in} = 5 and 10 V.

Part B: Superposition

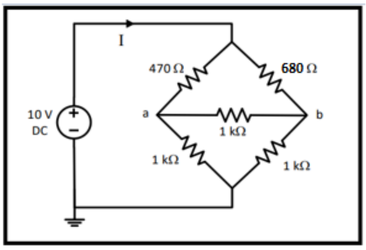
1. Use PSPICE to determine the voltages at all nodes and the current in all the branches for the circuits in **Figures 4.2 to 4.4**.

Part C: Thevenin's Theorem

1. Find and draw the Thevenin equivalent with respect to the terminals X, Y for the circuit in Figure 4.5 **(Show calculation of $V_{Thevenin}$ and $R_{Thevenin}$).**
2. Simulate the circuit of Figure 4.5 using PSPICE to determine the value of voltage around and current through the 330 Ω resistor.
3. Simulate Thevenin equivalent circuit that you found in step 1 shown in Figure 4.7 using PSPICE to determine the value of voltage around and current through the 330 Ω resistor.

Part D: Δ-Y Transformation

1. For the circuit shown below, calculate the equivalent Y for the Δ formed by the three 1kΩ resistors, draw the resulting circuit.
2. Simulate the circuit using PSPICE, find the value of the current I, and measure the voltage V_{ab}.
3. Simulate the circuit resulting from replacing the Δ formed by 1kΩ resistors with the equivalent Y found in step 1.



Part E: Reciprocity Theorem

1. Simulate the circuits of Figure 4.9 and Figure 4.10 using PSPICE to find the value of the current (I).

➢ **Procedure**:

Part A: The Proportionality Theorem

1. Connect the circuit of Figure 4.1.

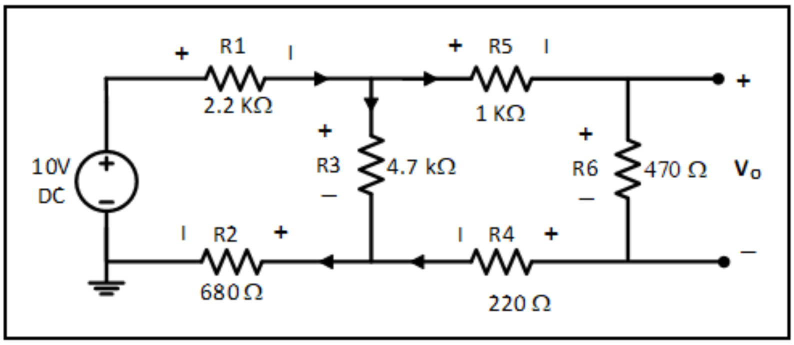


Figure 4.1

2. Set Vin to 5 then 10 V respectively. In each case measure V_O and record it in Table 4.1.

Question 1: Using measured values of V_O calculate K for both cases in the previous step, record results in Table 4.1. Compare results and discuss.

Part B: The Superposition Theorem

1. Connect the circuit of Figure 4.2 and set the power supply voltages as indicated.

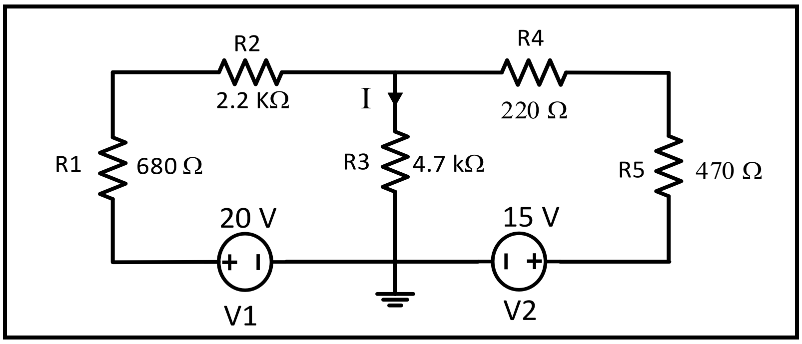


Figure 4.2

2. Measure the voltage across and the current through the 2.2 kΩ resistor, record results in Table 4.2.
3. Connect the circuit of Figure 4.3.

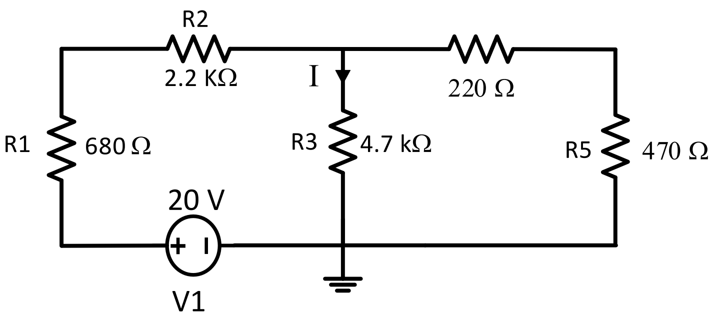


Figure 4.3

4. Measure the voltage across and the current through the 4.7 kΩ resistor, record results in Table 4.2.
5. Connect the circuit of Figure 4.4

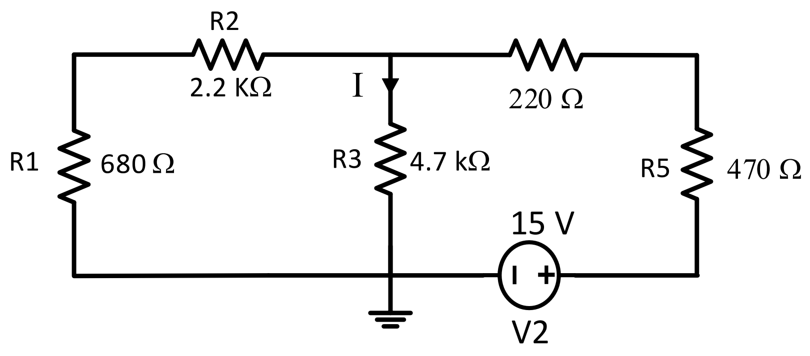


Figure 4.4

6. Measure the voltage across and the current through the 4.7 kΩ resistor, record results in Table 4.2.

Question 2: Add the voltage and current measurements made in steps 2 and 4 record results in Table 4.2. Does the sum equal the voltage and current measurement in step 6?

Part C: Thevenin Theorem

1. Connect the circuit of Figure 4.5 and set the power supply voltage to 10V.

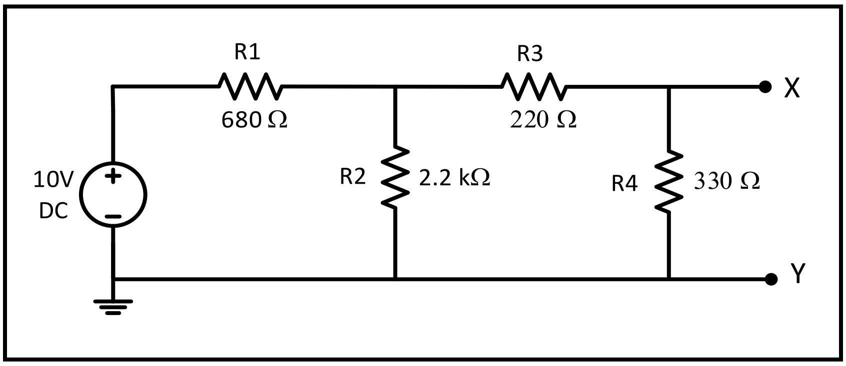


Figure 4.5

2. Measure the voltage across and the current through the 680 Ω resistor, record results in the Table 4.3.
3. Remove the 330 Ω resistor from the circuit and measure the open circuit voltage between terminals X and Y, record results in the Table 4.4.
4. Short circuit the terminals X and Y and measure the short circuit current I_{XY}, record results in the Table 4.4.

5. The first method to calculate R_{Th} is by: $R_{Th} = \frac{V_{OC}}{I_{SC}}$, calculate R_{Th} and record result in Table 4.4.
6. The second method to calculate R_{Th} is by using test voltage source. First kill all independent sources in the circuit (set the voltage source to zero as shown in figure 4.6)
7. Connect a 4 V test voltage source to points X and Y, measure the resulting current, and finally calculate $R_{Th} = \frac{V_T}{I_T}$. Record results in Table 4.5

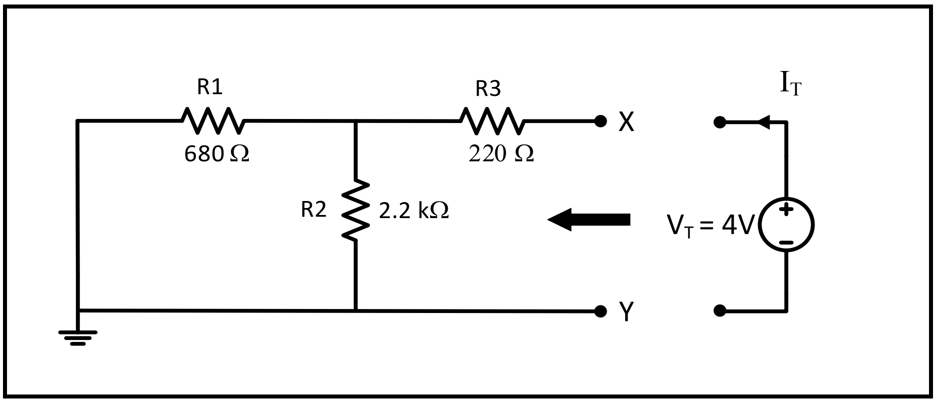


Figure 4.6

8. The third method to calculate R_{Th} is by connecting an **ohm meter** between points X and Y in place of the test source in figure 4.6, **measure R_{Th},** record result in Table 4.6.
9. Based on your calculation of $V_{Thevenin}$ and $R_{Thevenin}$ in your pre lab, connect Thevenin's equivalent circuit for the circuit of Figure 4.5 as shown in figure 4.7 (use decade box for $R_{Thevenin}$), measure the voltage across and the current through the 330 Ω resistor, record results in Table 4.7.

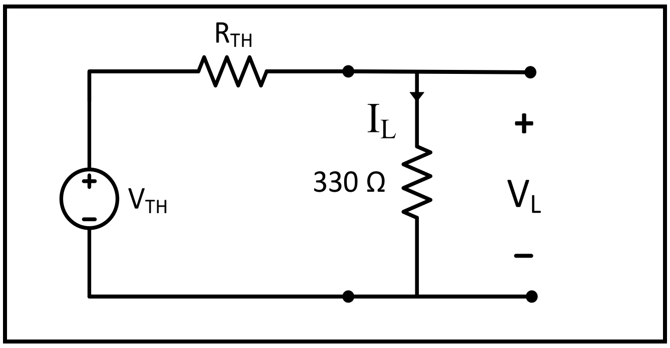


Figure 4.7

Question 3: Compare the measured values of load voltage and current in Thevenin's circuit and in the original circuit in figure 4.5. Compare the value of R_{Th} in the three methods of measurement.

Part D: The Reciprocity Theorem

1. Connect the circuit of Figure 4.8, measure and record the current I in Table 4.8.

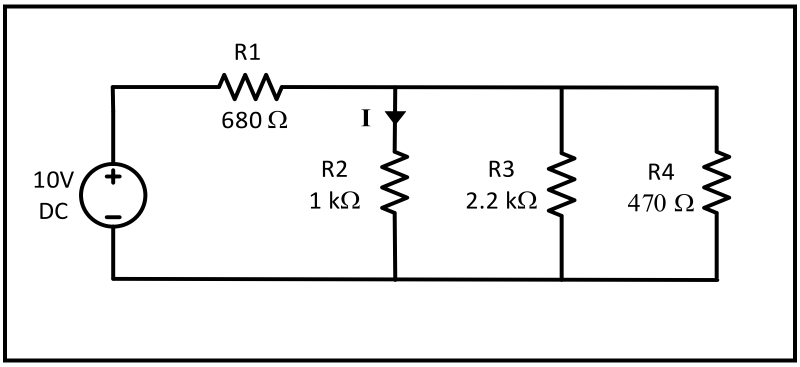


Figure 4.8

2. Connect the circuit of Figure 4.9, measure and record the current I in Table 4.8.

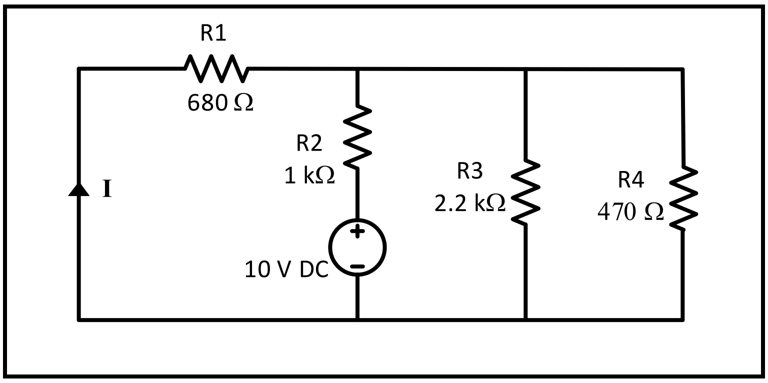


Figure 4.9

Question 5: Compare the currents measurements in steps 1 and 2.

Before Leaving the Laboratory

Be sure the following is completed before you leave the laboratory.

- Check to be sure that you have all the required measured values.
- Have the laboratory instructor check your laboratory readings.
- Restore your laboratory station (equipment and chairs) to the condition they were in when you arrived.

Thank you for your cooperation.

Experiment 5
Capacitors – Series and Parallel

➢ Objectives:

1. To explore the idea of the capacitance of a component.
2. To investigate what happens when capacitors are connected in series and in parallel

➢ Equipment Required:

1. Power supply.
2. Capacitors and resistors.
3. Oscilloscope
4. Digital Multimeters
5. Stop Watch

➢ Introduction:

- **Capacitance:**

Capacitance (symbol C) is a measure of a capacitor's ability to store charge. A large capacitance means that more charge can be stored. Capacitance is measured in farads, symbol F.

- Charge and Energy Stored:

The amount of charge (symbol Q) stored by a capacitor is given by:

Charge, $Q = C \times V$ where:

Q = charge in coulombs (C)
C = capacitance in farads (F)
V = voltage in volts (V)

When they store charge, capacitors are also storing energy:

Energy, $\mathbf{E = \frac{1}{2}QV = \frac{1}{2}CV^2}$, where E = energy in joules (J).

Note that capacitors return their stored energy to the circuit. They do not 'use up' electrical energy by converting it to heat as a resistor does. The energy stored by a capacitor is much smaller than the energy stored by a battery so they cannot be used as a practical source of energy for most purposes.

- **Capacitors in Series and Parallel:**

Combined capacitance (C) of
capacitors connected in **series: 1/C = 1/C1 +1/C2 + 1/C3 + ...**

Combined capacitance (C) of
capacitors connected in **parallel:** **C = C1 + C2 + C3 + ...**

Two or more capacitors are rarely deliberately connected in series in real circuits, but it can be useful to connect capacitors in parallel to obtain a very large capacitance, for example to smooth a power supply.

- **Charging a capacitor through a resistor:**

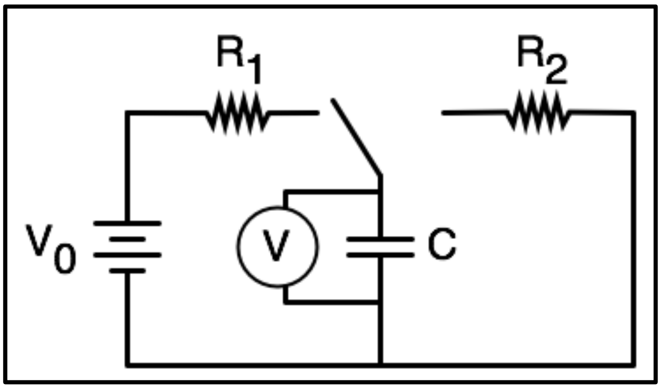


Figure 5.1

When the switch (the left-tilted line in the top center of the figure) is moved to the left, current flows from the battery with voltage V0, through the resistance R1 and charges the capacitor C. As charge builds up on the capacitor, its voltage increases according to the following curve and equation:

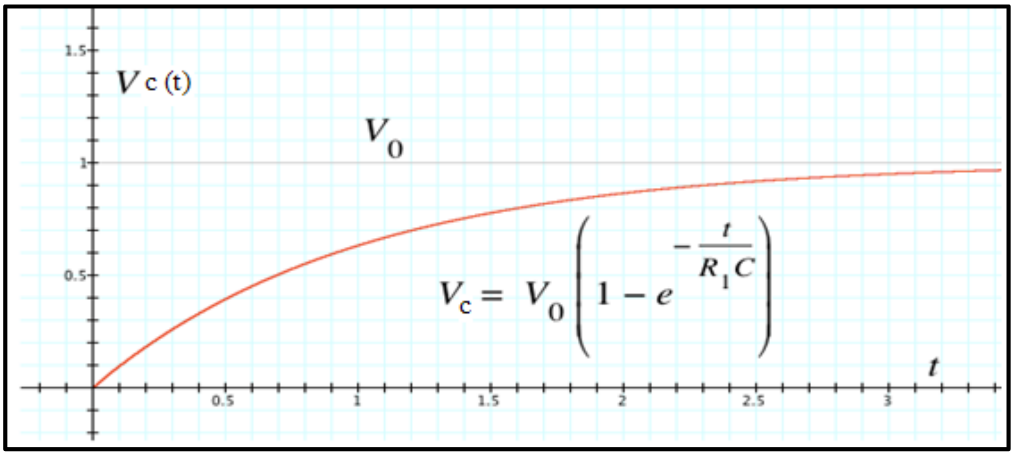


Figure 5.2

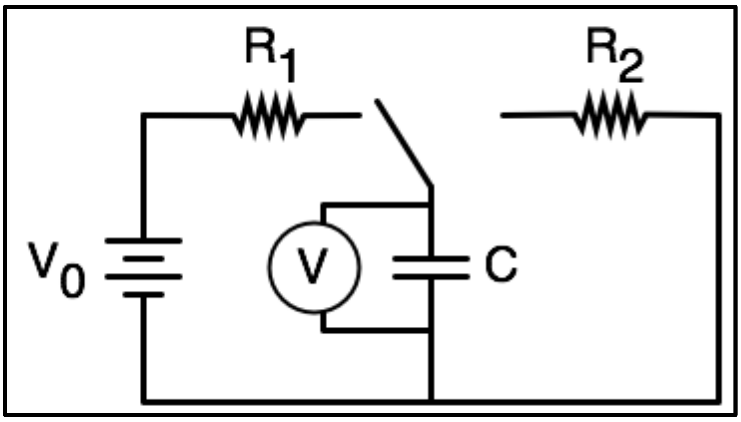


Figure 5.3

When the switch is moved to the right, current flows from the capacitor C with voltage V_0, through the resistance R_2 and discharges the capacitor C. As charge runs off the capacitor, its voltage decreases according to the following curve and equation:

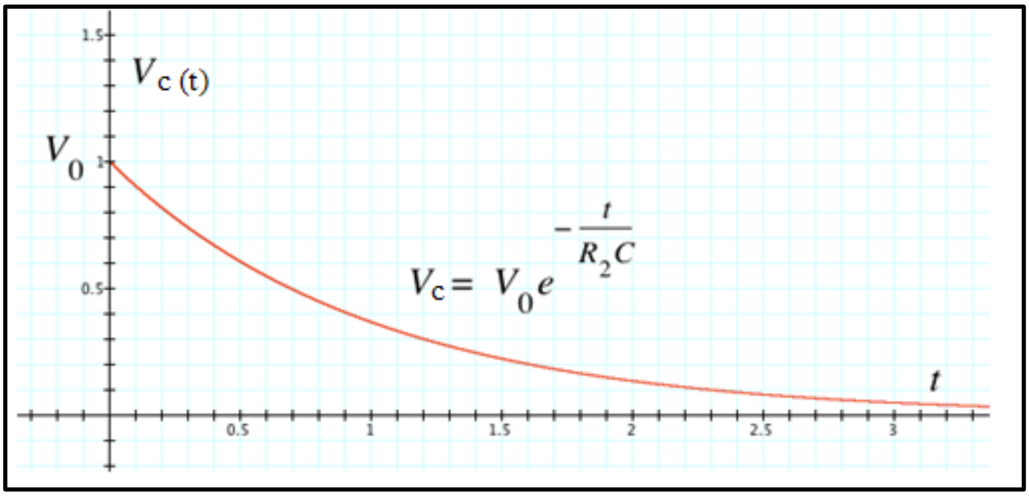


Figure 5.4

➢ Procedure:

In this experiment, you are going to explore the idea of capacitance charging and discharging. When you connect a DC power supply to a capacitor you will find that the current is maximum at the beginning (where the capacitor behaves as a short circuit if it is not initially charged) then current decreases with time until it becomes minimum (where the capacitor behaves as an open circuit). The voltage behaves as the opposite of the current.

Part A: Capacitance

1. Connect the circuit of Figure 4.5 (note capacitor polarity), set voltmeter to 20 V range, set ammeter to 200 μA range, and leave the power supply turned off (switch at 0 position).

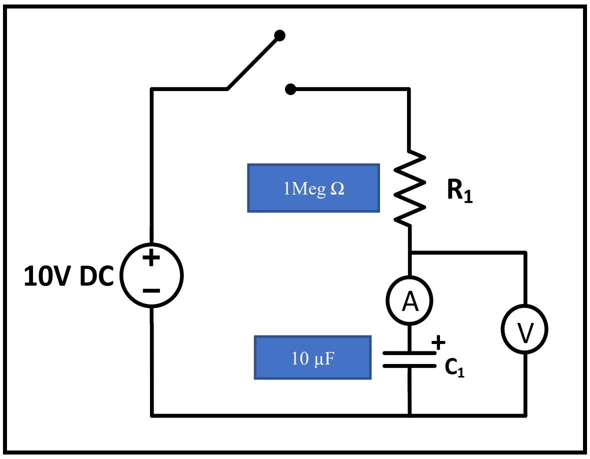


Figure 5.5

2. Insure that C_1 is fully discharged by shorting the terminals of the capacitor using a wire.
3. When the power supply is turned on (switch set to position 1), the current will initially jump to 10 μA, since the meter will not respond instantly this first reading is assumed and entered in Table 5.1.
4. Now prepare to take some rapid timed readings of the ammeter **(use your watch to detect the time),** turn on the power supply (set switch to position 1) and start recording current measurements 2 seconds after closing the switch according to Table 5.1.
5. Turn off the power supply (set switch to 0 position), and temporarily short circuit the capacitor C_1 by a wire to restore the original starting conditions. Then repeat step 4, but taking readings of the voltage across capacitor, record results in Table 5.1.

Question 1: Use Excel to plot graphs of voltage and current against time. Plot them on the same graph with different y-axes, discuss the behavior of these plots.

The plotted curves are for current and voltage, but charge on capacitor causes these effects. Charge present at any time is found from current which is a measure of rate of flow of charge ($I = dQ/dt \rightarrow Q = \int I \ dt$) thus, Q is given by integration or by area under current-time curve.

Question 2: Use excel to estimate the charge in the capacitor using data in Table 5.1 (I and t), record results in the same table (check the links at the end of the experiment to know how). It should be found that charge on C_1 is directly proportional to the voltage across it. ($Q \propto V \rightarrow Q = K \times V$) Where K is a constant of proportionality, it is termed the capacitance and given the letter C. $Q = C \times V$

Question 3: Use excel to plot a graph of Q against V and measure the slope to find the constant of proportionality (the capacitance). Compare it with the actual value used in lab.

Part B: Capacitors in series

1. Connect the circuit of Figure 5.6 (note capacitor polarity), set voltmeter to 20 V range, set ammeter to 200 µA, and leave the power supply turned off (switch at 0 position).
2. Insure that C_1 and C_2 are fully discharged by shorting their terminals using a wire for few seconds.

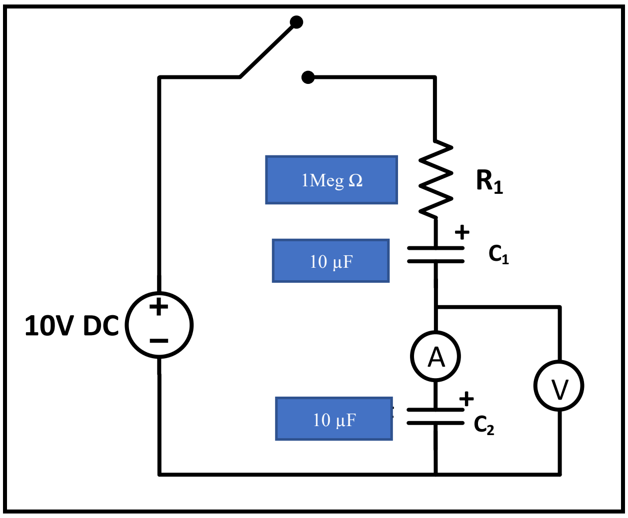


Figure 5.6

3. When the power supply is turned on (switch set to position 1), the current will initially jump to 10 µA, since the meter will not respond instantly this first reading may be assumed and entered in Table 5.2.
4. Now prepare to take some rapid timed readings of the ammeter **(use your watch to detect the time),** turn on the power supply (set switch to position 1) and start recording current measurements 2 seconds after closing the switch according to Table 5.2.
5. **When t = 40 s, disconnect the resistor and quickly and** record V_{C2} (voltage around C_2), then move the upper terminal of voltmeter to the point between C_1 and R_1 to read (V_{C1+C2}) the voltage around $C_1 + C_2$, record results in Table 5.3.

Question 4: Calculate V_{C1} from $V_{C1+C2} = V_{C1} + V_{C2}$, record results in Table 5.3

Question 5: Use Excel to estimate the value of charge in the capacitor using data in Table 5.2 (I and t), record results in the same Table.

Question 6: From Q = C*V and using data in Table 5.3, find the values of C_1, C_2, and C_{total} then find a relationship between C_{total} C_1, C_2 for the series connection (note that total charge found in question 5 is equal in both capacitors since they are connected in series).

Part C: Capacitors in parallel

1. Connect the circuit of Figure 5.7 (note capacitor polarity), set voltmeter to 20 V range, set ammeter to 200 µA, and leave the power supply turned off (switch at 0 position).
2. Repeat steps (2-4) in previous part for the circuit of Figure 5.7 using the timing found in Table 5.4, record results in the same Table.
3. **Immediately** after taking the current reading at 120 seconds, **disconnect the resistor and quickly** and read the voltmeter V_{CP} record result in Table 5.5.

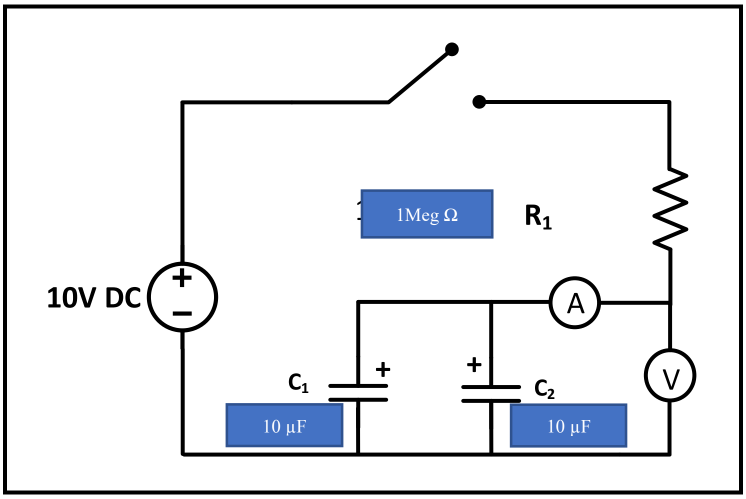


Figure 5.7

Question 6: Use Excel to estimate the value of charge in the capacitor using data in Table 5.4 (I and t), record results in the same Table.

Question 7: From Q = C*V and using data in Table 5.5, calculate the total parallel capacitance (PC) and compare it with C_1, C_2 you have calculated before. Write down a formula for PC.

Useful Links for calculations:

How to plot 2 y-axes: https://www.youtube.com/watch?v=8vzhSjNGcSo

How to find area under curve (charge): https://www.youtube.com/watch?v=U6EWnEsdR5A

Estimate area under curve (charge): https://www.youtube.com/watch?v=BlggkVgssNc

How To Find Slope Using Excel: https://www.youtube.com/watch?v=ZXAFNeEN-Ik

Experiment 6

First Order Circuits

➢ Objectives:

1. To examine the behavior of first-order circuits in response to a step input.

➢ Equipment Required:

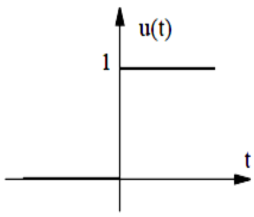
1. Function generator
2. Digital Multimeter.
3. Oscilloscope
4. Board, wires, capacitors, inductor box, and resistances.

➢ Introduction:

- **First-Order Circuits with DC sources (Step Response):**

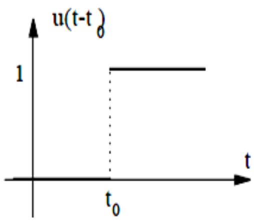
Unit Step function is defined as:

$$\begin{cases} u(t) = 0 & \text{for } t \leq 0^- \\ u(t) = 1 & \text{for } t \leq 0^+ \end{cases}$$



Defining $t' = t - t_0$, one can see that $u(t') = u(t - t_0)$ is:

$$\begin{cases} u(t - t_0) = 0 & \text{for } t \leq t_0^- \\ u(t - t_0) = 1 & \text{for } t \leq t_0^+ \end{cases}$$



Step functions are one way to illustrate switched circuit as is shown in the following.

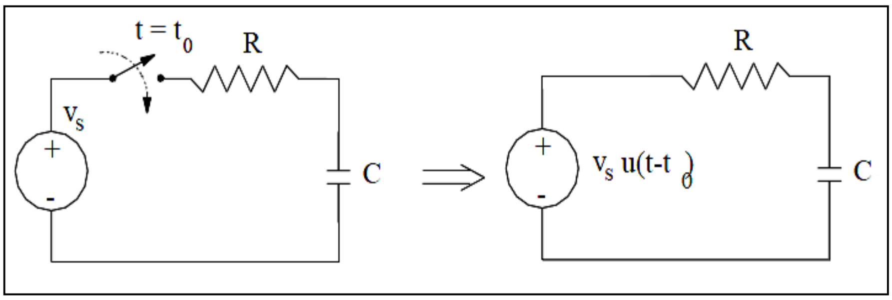


Figure 6.1

- **Step response of an RC circuit:**

Consider the RC circuit above. The switch closes at time t = 0 and the capacitor has an initial voltage of V_0. For t > 0, KVL results in Ric + V_C = Vs, or:

RC dvc/dt +vc = Vs

Vc (0^+) = Vc (0^-) = V_0

We have found the natural solution to RC circuit to be:

$$v_{C,n} = Ke^{-\frac{t}{\tau}} \quad \text{and} \quad \tau = RC$$

To find the forced response, V_C,f = A. Substituting for V_C,f in the differential equation, we get:

$$RC\frac{dA}{dt} + A = \mathbf{V_s} \quad \rightarrow \quad v_{C,f} = A = \mathbf{V_s}$$

$$v_C = v_{C,n} + v_{C,f} = Ke^{-\frac{t}{\tau}} + \mathbf{V_s}$$

Constant K is found from the initial condition: Vc (0^+) = Vc (0^-) = V_0:

Thus, the capacitor voltage waveform is:

$\mathcal{V}c(t) = K\, e^{-t/T}$ +Vs for t >= 0

$\mathcal{V}c(0+)$ = K + Vs

So: K= $\mathcal{V}c(0^+)$ –Vs

$\mathcal{V}c$ (t) = Vs + (Vc (0+) –Vs) $e^{-t/T}$

$\mathcal{V}c$ (∞) =Vs

$\mathcal{V}c$ (t) = $\mathcal{V}c$ (∞) + [$\mathcal{V}c(0^+)$ - $\mathcal{V}c$ (∞)] $e^{-t/T}$

$\mathcal{V}c$ (0^+) = initial voltage

$\mathcal{V}c$ (∞) = Final voltage

$\mathcal{V}c$ (t) = $\mathcal{V}c$ (∞) + [$\mathcal{V}c(0^+)$ - $\mathcal{V}c$ (∞)] $e^{-t/T}$

If we wait long enough, (mathematically: t→∞, practically: 5τ) the circuit will reach DC steady condition again, current in the capacitor becomes zero and its voltage reaches Vs as can be see either from the circuit or from the expression for V_C(t).

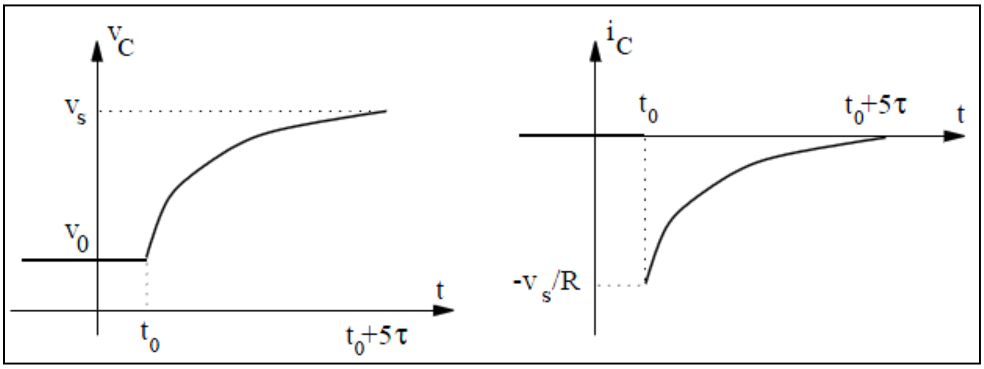


Figure 6.2

If the switch was closed at time t = t0 instead of time zero, the capacitor voltage waveform would be:

$\mathcal{V}c(t) = \mathcal{V}c(\infty) + [\mathcal{V}c(t0^+) - \mathcal{V}c(\infty)]\, e^{-(t-to)/T}$

Since V0 is the initial value of V_C and Vs is its final value, the above equation can be re-written as:

$$\begin{matrix} v_C \\ \text{at Time } t \end{matrix} = \left[\begin{matrix} \text{Initial Value of} \\ v_C \end{matrix} - \begin{matrix} \text{Final Value of} \\ v_C \end{matrix} \right] \times e^{-\frac{t-t_0}{\tau}} + \begin{matrix} \text{Final Value of} \\ v_C \end{matrix}$$

In fact, all voltages and currents in the circuit (also called "Response") will have the same waveform:

$$\begin{matrix} \text{Response} \\ \text{at Time } t \end{matrix} = \left[\begin{matrix} \text{Initial Value of} \\ \text{Response} \end{matrix} - \begin{matrix} \text{Final Value of} \\ \text{Response} \end{matrix} \right] \times e^{-\frac{t-t_0}{\tau}} + \begin{matrix} \text{Final Value of} \\ \text{Response} \end{matrix}$$

- **Step response of an RL circuit:**

Consider the RL circuit shown in Figure 6.3

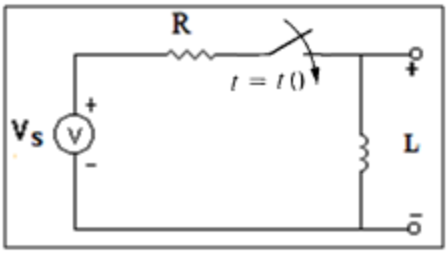


Figure 6.3

The switch closes at time t = t0 and the inductor has an initial current of $I_L(to^-)$. We can find the inductor current waveform following the procedure similar to one used for step response of RC circuits. Alternatively, we can use the "Response" formula identified above. Here the response of interest is iL. The time constant of the circuit is τ = L/R. The final value of the Response is iL(t → ∞) when the switch is closed and circuit has reached a DC steady state condition. Replacing the inductor with a short circuit, we find iL(t→∞) = i_s= Vs /R. Substituting in the response formula above, we get

$iL(t) = iL(\infty) + [\, iL(to^+) - iL(\infty)\,]\, e^{(-t-to)/T} \qquad t \geq to$

➢ Prelab:

****(Refer to procedure section for circuits figures)**

When simulating the circuits in all following parts, use a (V_{Pulse}) source from PSPICE library as the input voltage source, and connect a 50 Ω resistance in series with source (output resistance of the source), an example of the settings of the source are as shown below.

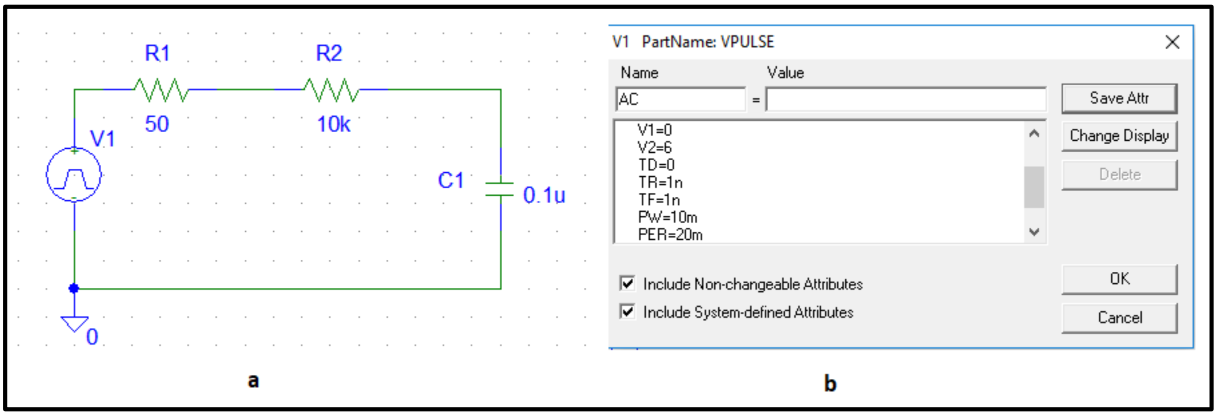


Figure 6.4: Simulation example and details.

Part A: Step response of First-order RC circuit

For the circuit of Figure 6.5:

1. Calculate V_C (t) using the general solution formula, show calculation of time constant (τ).
2. Use PSPICE to do transient analysis of the circuit. **Show V_C(t) and use cursors to measure time constant (τ).**
3. For the same circuit show V_R(t) using a **differential voltage marker,** and use **cursors to measure time constant (τ).**

Part B: Step response of First-order RL circuit

For the circuit of Figure 6.7:

1. Calculate V_L (t) using the general solution formula, show calculation of time constant (τ).
2. Use PSPICE to do transient analysis of the circuit. **Show V_L(t) and use cursors to measure time constant (τ).**
3. For the same circuit show V_R(t) using a **differential voltage marker,** and use **cursors to**

➢ Procedure

Part A: Step response of First-order RC circuit

1. Connect the circuit shown in Figure 6.5, with the function generator set to produce a square wave of: $V_{in} = 6\ V_{PP}$, f =50 Hz, and DC offset 3 V.

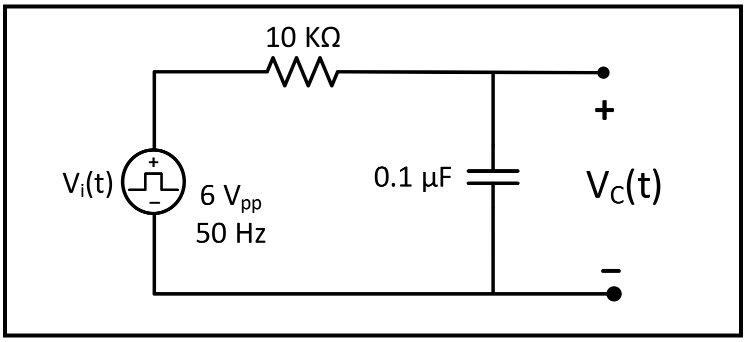


Figure 6.5

2. Measure the actual value of the 10 kΩ resistor, record result in Table 6.1.
3. Set the oscilloscope to display the waveform $V_C(t)$ produced by this circuit.
4. Estimate the value of τ_c with the help of cursors on the screen of the scope, **take a picture of the waveform (with the value of τ_c shown) to be included in lab report.**

Question 1: Use the computed value of τ_C in step 4, and the **actual value** of your resistor plus the **output resistance** of the function generator to compute the value of the capacitance C (τ_c = RC). The result should equal 0.1μF ± 20%.

5. **Repeat steps 1-4 for the circuit of Figure 6.6,** but this time showing $V_R(t)$ instead of Vc(t), and estimating the value of τ_R.

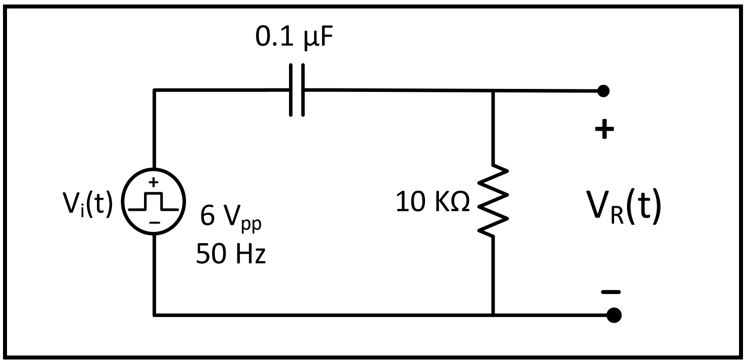


Figure 6.6

Question 2: In your report discuss the behavior of the graphs of $V_C(t)$ and $V_R(t)$, also compare the experimental values of time constant with the values calculated in prelab.

Part B: Step response of first-order RL circuit

1. Connect the circuit shown in Figure 6.7, with the function generator set to produce a square wave of: $V_{in} = 6\ V_{PP}$, f =50 Hz, and DC offset 3 V.
2. Measure the DC resistance of the inductor, and the actual value of the 1 kΩ resistor using DMM, record result in Table 6.2.

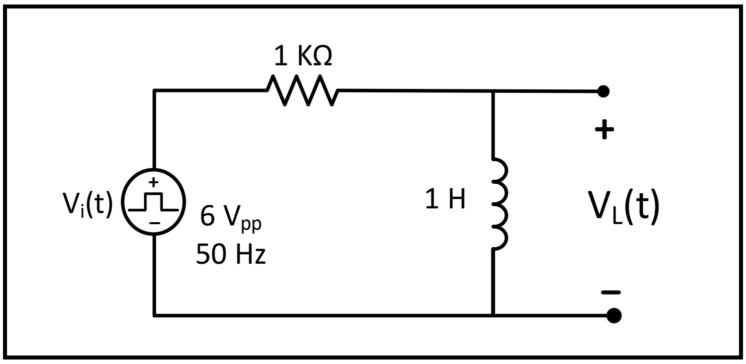


Figure 6.7

3. Set the oscilloscope to display the waveform $V_L(t)$ produced by this circuit.
4. Estimate the value of τ_L with the help of cursors on the screen of the scope, **take a picture of the waveform (with the value of τ_L shown) to be included in lab report.**

Question 3: Use the computed value of τ_L in step 4, and the **actual value** of your resistor plus the output resistance of the function generator plus the resistance of the inductor to compute the value of the Inductance L ($\tau_L = L/R$). The result should equal 1H ± 20%.

5. **Repeat steps 1-4 for the circuit of Figure 6.8,** but this time showing $V_R(t)$ instead of $V_L(t)$, and estimating the value of τ_R.

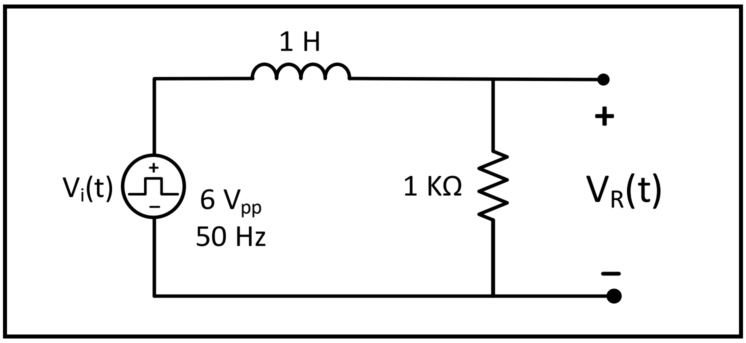


Figure 6.8

Question 4: In your report discuss the behavior of the graphs of $V_L(t)$ and $V_R(t)$, also compare the experimental values of time constant with the values calculated in prelab.

Experiment 7

Second Order Circuits

➢ **Objectives:**

1. To examine the behavior of second-order circuits in response to a step input.

➢ **Equipment Required**:

1. Function generator.
2. Digital Multimeter.
3. Oscilloscope
4. Board, wires, capacitors, inductor box, and resistances.

➢ **Introduction:**

The Natural and Step Response of a Series *RLC* Circuit

The procedures for finding the natural or step responses of a series *RLC* circuit are the same as those used to find the natural or step responses of a parallel *RLC* circuit, because both circuits are described by differential equations that have the same form. We begin by summing the voltages around the closed path in the circuit shown in Figure 7.1 Thus

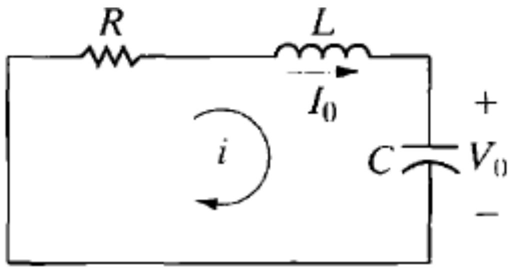


Figure 7.1 A circuit used to illustrate the natural response of a series *RLC* circuit.

$$Ri + L\frac{di}{dt} + \frac{1}{C}\int_0^t i\,d\tau + V_0 = 0. \quad \text{.........(7.1)}$$

We now differentiate Eq 6.1 once with respect to *t* to get

$$R\frac{di}{dt} + L\frac{d^2i}{dt^2} + \frac{i}{C} = 0. \quad \text{.........(7.2)}$$

which we can rearrange as

$$\frac{d^2i}{dt^2} + \frac{R}{L}\frac{di}{dt} + \frac{i}{LC} = 0. \quad \text{.........(7.3)}$$

From Eq. 7.3, the characteristic equation for the series *RLC* circuit is

$$s^2 + \frac{R}{L}s + \frac{1}{LC} = 0. \quad \text{……………………………………………………..……..(7.4)}$$

The roots of the characteristic equation are

$$s_{1,2} = -\frac{R}{2L} \pm \sqrt{\left(\frac{R}{2L}\right)^2 - \frac{1}{LC}}, \quad \text{…………………………………………………..…..(7.5)}$$

Or

$$s_{1,2} = -\alpha \pm \sqrt{\alpha^2 - \omega_0^2}. \quad \text{……………………………………………………..…….(7.6)}$$

The neper frequency *(a)* for the series *RLC* circuit is

$$\alpha = \frac{R}{2L} \text{ rad/s,} \quad \text{…………………………………..……………………….…………(7.7)}$$

and the expression for the resonant radian frequency is

$$\omega_0 = \frac{1}{\sqrt{LC}} \text{ rad/s.} \quad \text{…………………………..…………………………….…………..(7.8)}$$

Note that the equation for neper frequency of the series *RLC* circuit differs from that of the parallel *RLC* circuit, but the equations for resonant and damped radian frequencies are the same.

The current response will be overdamped, underdamped, or critically damped according to whether $wo^2 < \alpha^2$, $wo^2 > \alpha^2$, or $wo^2 = \alpha^2$, respectively.

Thus the three possible solutions for the current are as follows:

$$i(t) = A_1e^{s_1t} + A_2e^{s_2t} \text{ (overdamped),}$$
$$i(t) = B_1e^{-\alpha t}\cos\omega_d t + B_2e^{-\alpha t}\sin\omega_d t \text{ (underdamped),}$$
$$i(t) = D_1te^{-\alpha t} + D_2e^{-\alpha t} \text{ (critically damped).}$$

……………………...…..(7.9)

When you have obtained the natural current response, you can find the natural voltage response across any circuit element. For convenience, we assume that zero energy is stored in the circuit at the instant the switch is closed.

Applying Kirchhoff s voltage law to the circuit shown in Figure 7.2 gives:

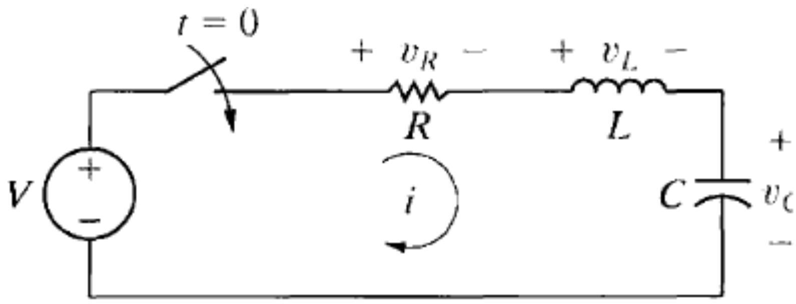


Figure 7.2: A circuit used to illustrate the step response of a series *RLC* circuit

$$V = Ri + L\frac{di}{dt} + v_C$$ ……………………………………………………….(7.10)

The current *(i)* is related to the capacitor voltage (%•) by the expression

$$i = C\frac{dv_C}{dt}.$$ ………………………………………………….………..…...…(7.11)

from which

$$\frac{di}{dt} = C\frac{d^2v_C}{dt^2}$$ ……………………………………………………………………...(7.12)

Substitute Eqs. 6.11 and 6.12 into Eq. 6.10 and write the resulting expression as

$$\frac{d^2v_C}{dt^2} + \frac{R}{L}\frac{dv_C}{dt} + \frac{v_C}{LC} = \frac{V}{LC}$$ …………………………………………………(7.13)

The three possible solutions for *vc*are as follows:

$$v_C = V_f + A_1'e^{s_1t} + A_2'e^{s_2t} \text{ (overdamped)},$$

$$v_C = V_f + B_1'e^{-\alpha t}\cos\omega_d t + B_2'e^{-\alpha t}\sin\omega_d t \text{ (underdamped)},$$

$$v_C = V_f + D_1'te^{-\alpha t} + D_2'e^{-\alpha t} \text{ (critically damped)},$$

where *Vf* is the final value of *vc.* Hence, from the circuit shown in Figure 7.2, the final value of *vc* is the dc source voltage *V*

Note: Study the Natural and Step Response of a parallel *RLC* Circuit

➢ <u>Prelab</u>:

**(Refer to procedure section for circuits figures)

Part A: Step response of second-order Series RLC circuit

For the circuit of Figure 7.5:

1. For R = 10 kΩ, calculate the roots of the characteristic equation and write an expression for $V_C(t)$. Use PSPICE to do transient analysis of the circuit, and show $V_C(t)$.
2. Calculate the critical resistance R_C that will result in equal roots ($S_1 = S_2 = -\alpha$) and write an expression for $V_C(t)$. Use PSPICE to do transient analysis of the circuit and show $V_C(t)$.
3. For **R = 500 Ω,** calculate the roots of the characteristic equation, **showing the value of α and ω_d** and write an expression for $V_C(t)$. Use PSPICE to do transient analysis of the circuit, show $V_C(t)$, and measure **α and ω_d using cursors as shown in figure 7.4**.

Part B: Step response of second-order parallel RLC circuit

For the circuit of Figure 7.6:

1. For R = 4 kΩ, calculate the roots of the characteristic equation **showing the value of α and ω_d**. Write an expression of $V_C(t)$. Use PSPICE to do transient analysis of the circuit, show $V_C(t)$, and measure **α and ω_d using cursors as shown in figure 7.4**.
2. Calculate the critical resistance R_C that will result in equal roots ($S_1 = S_2 = -\alpha$) and write an expression for $V_C(t)$. Use PSPICE to do transient analysis of the circuit and show $V_C(t)$.
3. For R = 150 Ω, calculate the roots of the characteristic equation and write an expression for $V_C(t)$. Use PSPICE to do transient analysis of the circuit and show $V_C(t)$.

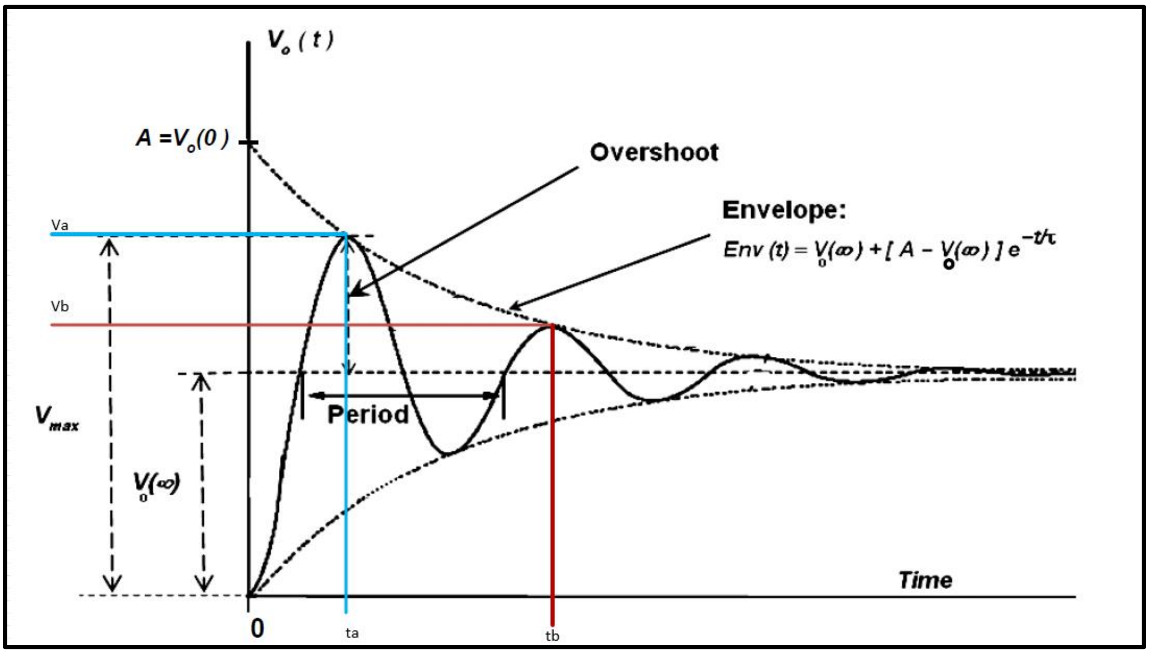


Figure 7.4: Example of calculating α and ω_d of underdamped response

Decay time constant $$\tau = \frac{t_b - t_a}{ln\left(\frac{V_a - V_{o(\infty)}}{V_b - V_{o(\infty)}}\right)}$$ …………………………..........(7.14)

Damping Coefficient $$\alpha = \frac{1}{\tau}$$ ……………………….........................(7.15)

Damped radian frequency $$\omega_d = \frac{2\pi}{t_b - t_a}$$ ……………………..........................(7.16)

➢ Procedure

Part A: Step response of second-order Series RLC circuit

1. Connect the circuit shown in Figure 7.5, with the function generator set to produce a square wave of: 3 V_{PP}, f = 100 Hz, and DC offset 1.5 V **(Use decade box for R)**

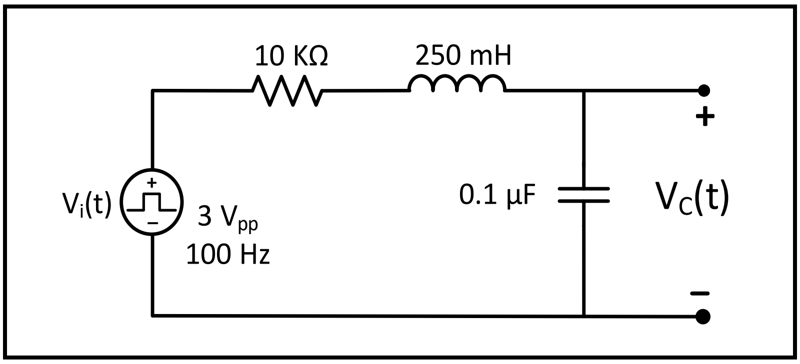


Figure 7.5

2. Measure the DC resistance of the inductor using DMM.
3. Connect CH2 of the oscilloscope across the capacitor.
4. Adjust the variable resistor over its entire range (10 kΩ to 1 kΩ) and note how the output waveform $V_C(t)$ varies.
5. Consider the following cases:

Case A: Over damped response

1. Set the variable resistor to 10 kΩ.
2. Take a picture of the resulting waveform, **to be included in lab report.**

Case B: Critically damped response

1. Set the variable resistor to the critical resistance ($R_{critical}$) found in the prelab.
2. Take a picture of the resulting waveform $V_C(t)$, **to be included in lab report.**

Question 1: Discuss the difference between the over and critical damped response as seen in the graphs.

Case C: Under damped response

1. Set the variable resistor so that the total circuit resistance is equal to 500 Ω.

Notice that the total resistance of the circuit is the sum of: the value of the variable resistor, the output resistance of the function generator, and the parasitic resistance of the inductor.

2. Use cursors to find the values of **V_a, t_a, V_b, t_b, and $V(\infty)$,** as shown in figure 7.4, record results in Table 7.1.
3. Take a picture of $V_c(t)$ (with the value of cursors shown), **to be included in lab report.**

Question 2: Using measured values in step 2 of case (c), calculate decay-envelope time constant (τ), the damping coefficient (α), and damped frequency (ω_d), compare these values with theoretical values calculated in prelab.

Part B: Step response of second-order parallel RLC circuit

1. Connect the circuit in Figure 7.6, keeping the previous setting of the function generator (Square wave output, 3 V_{PP}, DC offset 1.5 V, f = 100 Hz), **(Use decade box for R).**

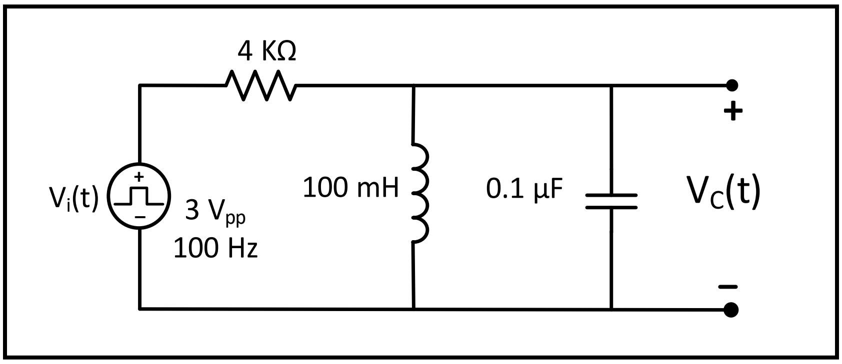


Figure 7.6

2. Measure the DC resistance of the inductor using DMM.
3. Connect CH2 of the oscilloscope across the capacitor.
4. Adjust the variable resistor (4 kΩ) over its entire range and note how the output waveform $V_c(t)$ varies.
5. Consider the following cases:

Case A: Under damped response

1. Set the variable resistor so that the total circuit resistance is equal to 4 kΩ.
   Notice that the total resistance of the circuit is the sum of the value of the variable resistor, the output resistance of the function generator and the parasitic resistance of the inductor.
2. Use cursors to find the values of **V_a, t_a, V_b, t_b, and $V(\infty)$,** as shown in figure 7.4, record results in Table 7.2.
3. Take a picture of $V_C(t)$ (with the value of cursors shown), **to be included in lab report.**

Question 3: Using measured values in step 2 of case (a), calculate decay-envelope time constant (τ), the damping coefficient (α), and damped frequency (ω_d), compare these values with theoretical values calculated in prelab.

Case B: Critically damped response

1. Set the variable resistor so that the total circuit resistance is equal to the critical resistance $R_{critical}$ found in the prelab.
2. Take a picture of the resulting waveform $V_C(t)$, **to be included in lab report.**

Case C: Over damped response

1. Set the variable resistor so that the total circuit resistance is equal to 150 Ω.
2. Take a picture of $V_C(t)$ as displayed by the oscilloscope, **to be included in lab report.**

Question 4: Discuss the difference between the over and critical damped response as seen in the graphs.

Before Leaving the Laboratory

Be sure the following is completed before you leave the laboratory.

- Check to be sure that you have all the required measured values.
- Have the laboratory instructor check your laboratory readings.
- Restore your laboratory station (equipment and chairs) to the condition they were in when you arrived.

Thank you for your cooperation.

Experiment 8
Impedance and Sinusoidal Steady State

➢ Objectives:

1. Examine the frequency dependent behavior of impedance.
2. Examine the sinusoidal steady-state response of RL, and RC circuits.
3. Measuring of self-inductance and capacitance

➢ Equipment Required:

1. Function generator
2. Digital Multimeter.
3. Oscilloscope
4. Board, wires, capacitors, inductor box, and resistances.
5. DC Power Supply.

➢ Introduction:

The general equation for a sinusoidal source is

$v = V_m \cos(\omega t + \varphi)$ (voltage source)

$i = I_m \cos(\omega t + \varphi)$ (current source)

Where V_m (or I_m) is the maximum amplitude, w is the frequency, and φ is the phase angle.

The frequency, w, of a sinusoidal response is the same as the frequency of the sinusoidal source driving the circuit. The amplitude and phase angle of the response are usually different from those of the source.

The best way to find the steady-state voltages and currents in a circuit driven by sinusoidal sources is to perform the analysis in the frequency domain. The following mathematical transforms allow us to move between the time and frequency domains.

The phasor transform (from the time domain to the frequency domain):

$V = V_m e^{j\varphi} = P\{V_m\cos(\omega t + \varphi)\}$.

The inverse phasor transform (from the frequency domain to the time domain):

$$\mathcal{P}^{-1}\{V_m e^{j\phi} = \Re\{V_m e^{j\phi} e^{j\omega t}\}$$

The V-I Relationship for a Resistor:

From Ohm's law, if the current in a resistor varies sinusoidally with time that is, if i = I_m cos (ωt + θ_i) the voltage at the terminals of the resistor, as shown in Figure 8.1 is

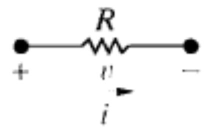


Figure 8.1: A resistive element carrying a sinusoidal current

v = R [I_m cos (ωt +θ_i)]

v = RI_m[cos (ωt + θ_i)]

Where I_m is the maximum amplitude of the current in amperes and θ_i is the phase angle of the current. The phasor transform of this voltage is

$$\mathbf{V} = RI_m e^{j\theta_i} = RI_m\angle\theta_i.$$

But $I_m\angle\theta_i$ is the phasor representation of the sinusoidal current, so we can write as V = RI which states that the phasor voltage at the terminals of a resistor is simply the resistance times the phasor current. There is no phase shift between the current and voltage. Figure 8.2 depicts this phase relationship, where the phase angle of both the voltage and the current waveforms is 60°. The signals are said to be in phase because they both reach corresponding values on their respective curves at the same time.

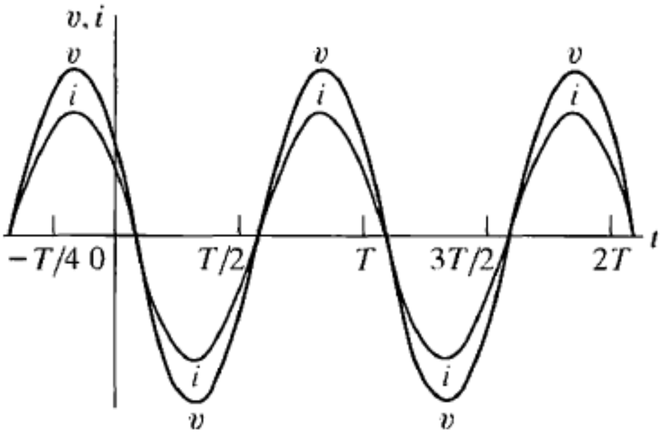


Figure 8.2: Voltage and current at the terminals of a resistor are in phase.

The V-I Relationship for an Inductor:

We derive the relationship between the phasor current and phasor voltage at the terminals of an inductor by assuming a sinusoidal current and using Ldi/dt to establish the corresponding voltage. Thus, for i =I_m cos (ωt+θ_i), the expression for the voltage is

v = L di/dt = -ω L I_m sin (ωt + θ_i).

The phasor representation of the voltage given: **v= jωLI**

The equation states that the phasor voltage at the terminals of an inductor equals jwL times the phasor current. Figure 8.3 shows the frequency domain equivalent circuit for the inductor.

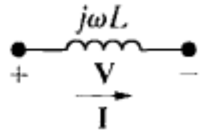


Figure 8.3: The frequency-domain equivalent circuit for an inductor

$v = (\omega L\angle 90)I_m\angle\theta_i$

$v = \omega L\ I_m\ \angle(90+\theta_i)$

Which indicates that the voltage and current are out of phase by exactly 90°. In particular, the voltage leads the current by 90°, or, equivalently, the current lags behind the voltage by 90°. Figure 8.4 illustrates this concept of voltage leading current or current lagging voltage.

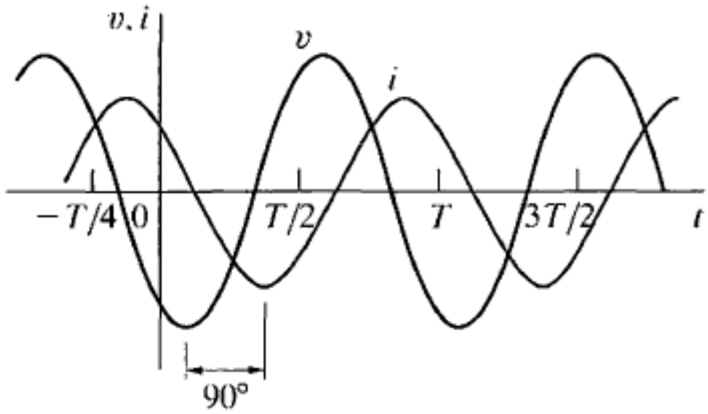


Figure 8.4: A plot showing the phase relationship between the current and voltage at the terminals of an inductor (θ_i = 60).

The V-I Relationship for a Capacitor:

The relationship between the phasor current and phasor voltage at the terminals of a capacitor

i = c dv/dt ,and assume that

$v = V_m \cos(\omega t+ \theta_v)$, then

I = jωCv

The voltage as a function of the current

V=1/jωc I

The equivalent circuit for the capacitor in the phasor domain is as shown in Figure 8.5

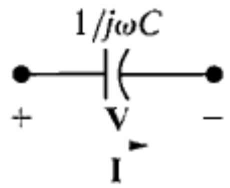


Figure 8.5: The frequency domain equivalent circuit of a capacitor.

The voltage across the terminals of a capacitor lags behind the current by exactly 90°.

$V=1/\omega c\angle -90\ I_m\angle\theta_i$

$V= I_m /\omega c\ \angle(\theta i-90)$

The current leads the voltage by 90°. Figure 8.6 shows the phase relationship between the current and voltage at the terminals of a capacitor.

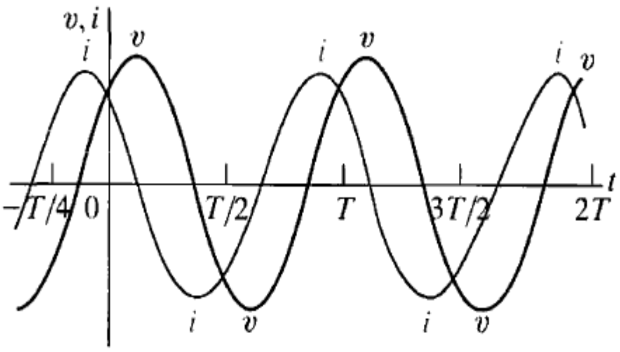


Figure 8.6: A plot showing the phase relationship between the current and voltage at the terminals of a capacitor (θ_i = 60°).

Impedance and Reactance:

V = ZI

Where Z represents the impedance of the circuit element. Solving for Z in equation, you can see that impedance is the ratio of a circuit element's voltage phasor to its current phasor. Thus the impedance of a resistor is R, the impedance of an inductor is jωL, the impedance of mutual inductance is jωM, and the impedance of a capacitor is 1/ωC. In all cases, impedance is measured in ohms.

Note that, although impedance is a complex number, it is not a phasor. Remember, a phasor is a complex number that shows up as the coefficient of e^{jt}. Thus, although all phasors are complex numbers, not all complex numbers are phasors. Impedance in the frequency domain is the quantity analogous to resistance, inductance, and capacitance in the time domain. The imaginary part of the impedance is called reactance. The values of impedance and reactance for each of the component values are summarized in Table 8.1. And finally, a reminder. If the reference direction for the current in a passive circuit element is in the direction of the voltage rise across the element, you must insert a minus sign into the equation that relates the voltage to the current.

TABLE 8.1*:* Impedance and Reactance Values

Circuit Element	Impedance	Reactance
Resistor	R	—
Inductor	$j\omega L$	ωL
Capacitor	$j(-1/\omega C)$	$-1/\omega C$

- **Prelab**:

  - **Part A: Impedance Measurement**

1. For the circuits shown in Figures 8.7 - 8.10 **calculate the magnitude** of the impedances Z_R, Z_C, Z_L, and Z_{RC} respectively, for the following frequencies: 250, 500, 1000 and 2000 Hz.

  - **Part B: Phase Measurement**

1. For the circuit shown in Figure 8.11:
   a. Use PSPICE to do transient analysis of the circuit, show $V_{in}(t)$ and $V_R(t)$ on one plot (you may need to use different Y-axes).
   b. Use cursors to measure the time difference between the peaks of the two signals, then use the following relationship to calculate the phase shift using the measured time $\{\Delta\theta = 360° \text{ x f x } \Delta t\}$.
2. Repeat the same procedure in step 1 above for the circuit shown in Figure 8.12.

- **<u>Procedure</u>**:

- **Part A: Impedance Measurement**

1. Connect the circuit in Figure 8.7, set V_{in} to sinusoidal voltage at 8 V_{PP}, f = 250 Hz

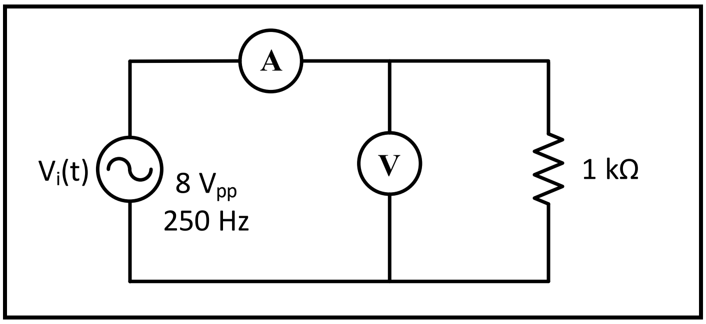


Figure 8.7

   a. Measure |I| and |V| **using DMM**, record results in Table 8.1.

   b. Repeat step (a) for f = 500 Hz, 1000 Hz and 2000 Hz, record results in Table 8.1.

   Question 1: Using measured values of |I| and |V| calculate $|Z_R|$ for each value of (f) recording results in Table 8.1. How does changing frequency affects the impedance Z_R?

2. Connect the circuit in Figure 8.8, set V_{in} to sinusoidal voltage at 8 V_{PP}, f = 250 Hz.

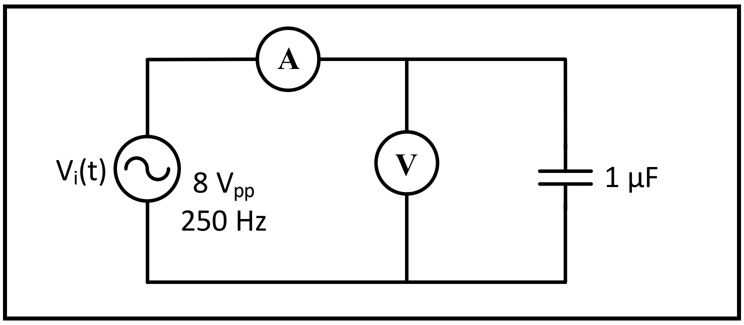


Figure 8.8

   a. Repeat steps (a) and (b) in (1) for the circuit in Figure 8.8 record results in Table 8.2.

   Question 2: Using measured values of |I| and |V| calculate $|Z_C|$ for each value of (f) recording results in Table 8.2. How does changing frequency affects the impedance Z_C?

3. Connect the circuit in Figure 8.9, set V_{in} to sinusoidal voltage at 8 V_{PP}, f = 250 Hz.

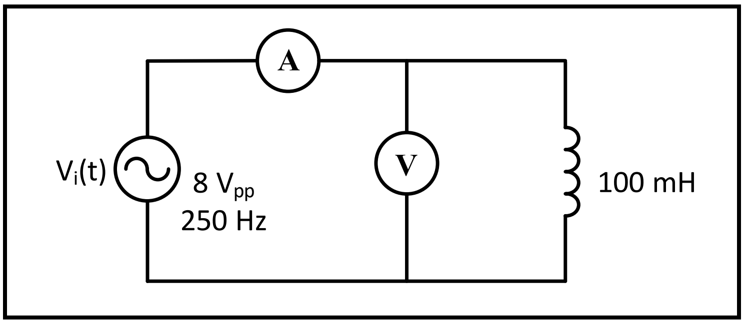


Figure 8.9

a. Repeat steps (a) and (b) in (1) for the circuit of Figure 8.9 record results in Table 8.3.

Question 3: Using measured values of |I| and |V| calculate $|Z_L|$ for each value of (f) recording results in Table 8.3. How does changing frequency affects the impedance Z_L?

4. Connect the circuit in Figure 8.10, set V_{in} to sinusoidal voltage at 8 V_{PP}, and f = 250 Hz.

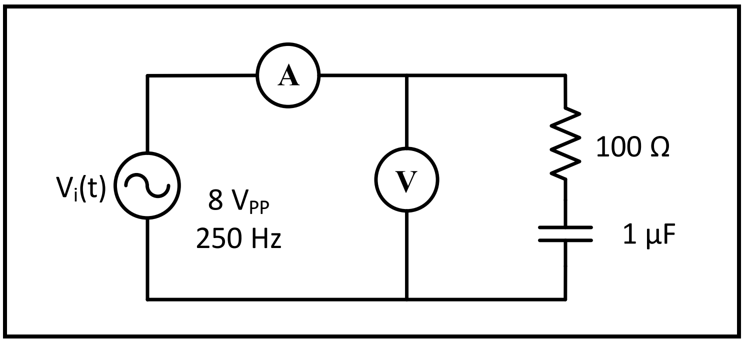


Figure 8.10

a. Repeat steps (a) and (b) in (1) for the circuit in Figure 8.10 record results in Table 8.4.

Question 4: Using measured values of |I| and |V| calculate $|Z_{RC}|$ for each value of (f) recording results in Table 8.4. How does changing frequency affects the impedance Z_{RC} compared to Z_C in step 2?

➢ Part B: Phase shift measurement

1. Connect the circuit in Figure 8.11, set V_{in} to sinusoidal voltage at 8 V_{PP} and f = 100 Hz.

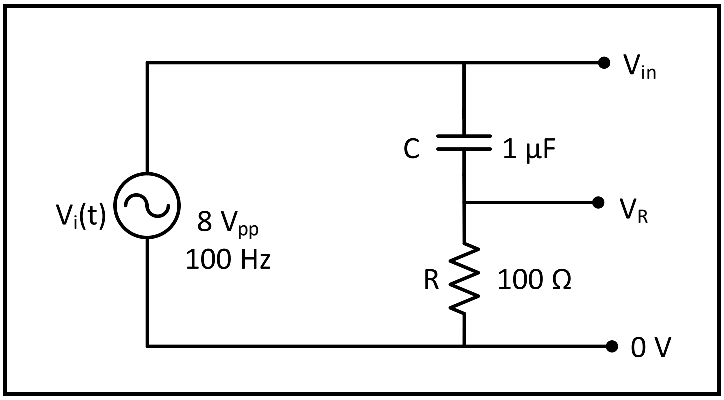


Figure 8.11

a. Connect CH1 of oscilloscope across Vin and CH2 across V_R. Measure the time difference (Δt) between V_{in}(t) and V_R(t) using cursors, record result in Table 8.5.

Note that the voltage V_R has the same waveform as the current in the circuit, and V_{in} will approximately equal ≈ V_C since Z_C is much higher than Z_R.

Question 5: Using measured value of (Δt) calculate the phase shift (Δθ), record result in Table 8.5. If (V_{in}) represents (V_C), and (V_R) represents current (I_C), which one of the voltage and the current lags the other?

2. Connect the circuit in Figure 8.12, set V_{in} to sinusoidal voltage at 8 V_{PP} and f = 1 KHz.

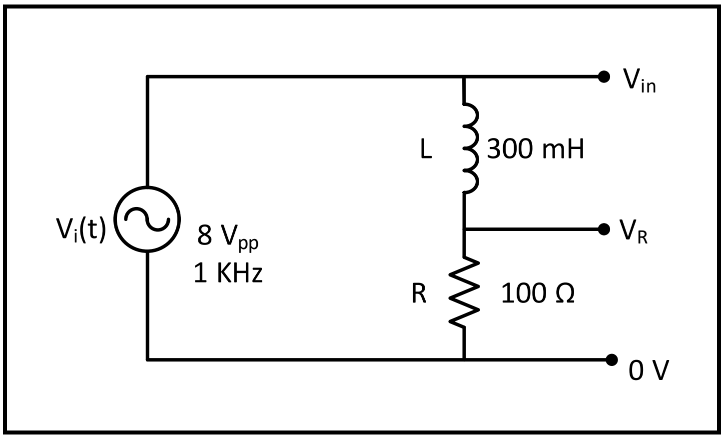


Figure 8.12

a. Connect CH1 of oscilloscope across Vin and CH2 across V_R. Measure the time difference between $V_{in}(t)$ and $V_R(t)$ using cursors, record results in Table 8.6.

Note that the voltage V_R has the same waveform as the current in the circuit, and V_{in} will approximately equal ≈ V_L since Z_L is much higher than Z_R.

Question 6: Using measured value of (Δt) calculate the phase shift (Δθ), record result in Table 8.6. If (V_{in}) represents (V_L), and (V_R) represents current (I_L), which one of the voltage and the current lags the other?

➢ Part C: Inductance and Capacitance Measurement

1. Connect the circuit of Figure 8.13, set V_{in} to sinusoidal voltage at 6 V_{PP}, and f = 1 kHz.

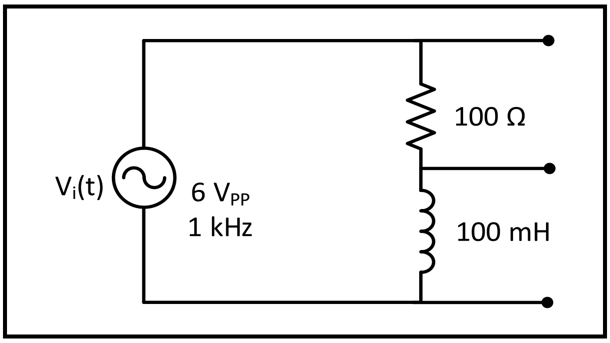


Figure 8.13

a. Measure |I|, $|V_R|$, and $|V_L|$ using DMM, record results in Table 8.7.

Question 7: Using measured values of |I| and |V|, calculate the self-inductance of the coil, record result in Table 8.7.

2. Connect the circuit of Figure 8.14, set V_{in} to sinusoidal voltage at 6 V_{PP}, f = 1 kHz.

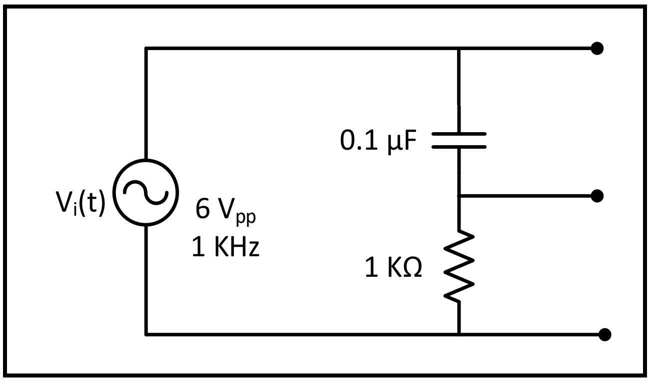


Figure 8.14

a. Measure $|V_R|$, $|V_C|$, and |I| using DMM, record results in Table 8.8.

Question 8: Using measured values of |I| and |V| calculate the capacitance of the capacitor record results in Table 8.8.

Before Leaving the Laboratory

Be sure the following is completed before you leave the laboratory.

(d) Check to be sure that you have all the required measured values.

(e) Have the laboratory instructor check your laboratory readings.

(f) Restore your laboratory station (equipment and chairs) to the condition they were in when you arrived.

Thank you for your cooperation.

Experiment 9

AC & DC Power Analysis and Design

➢ Objectives:

1. Examine AC and DC circuit power analysis and maximum power transfer design.

➢ Equipment Required:

1. Digital Multimeter.
2. Function Generator.
3. Oscilloscope.
4. Resistors.
5. Inductor 100mH.

➢ Introduction:

To understand power factor, we'll first start with the definition of some basic terms:

KW is Working Power (also called Actual Power or Active Power or Real Power).

It is the power that actually powers the equipment and performs useful work.

KVAR is Reactive Power.

It is the power that magnetic equipment (transformer, motor and relay) needs to produce the magnetizing flux.

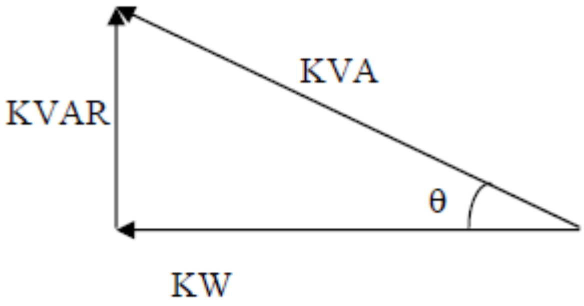
KVA is Apparent Power

Power Factor (P.F.) is the ratio of Working Power to Apparent Power.

$$PF = KW/\left((KW)^2 + (KVAR)^2\right)^{0.5} = P_{av}/P_a$$

$$PF = \cos(\theta v - \varphi i)$$

The "Power Triangle" Figure 9.1 illustrates this relationship between KW, KVA, KVAR, and Power Factor:



$$P.F. = \frac{KW}{KVA} = COS\ \theta \quad \Big| \quad \frac{KVAR}{KVA} = SIN\ \theta$$

$$KVA = \sqrt{KW^2 + KVAR^2}$$

Figure 9.1

Note:

- KVAR would be very small.
- KW and KVA would be almost equal
- The angle θ (formed between KW and KVA) would approach zero
- Cosine θ would then approach one
- Power Factor would approach one

Since power factor is defined as the ratio of KW to KVA, we see that low power factor results when KW is small in relation to KVA. What causes a large KVAR in a system? The answer is…inductive loads.

Inductive loads (which are sources of Reactive Power) include:

- Transformers
- Induction motors
- Induction generators (wind mill generators)
- High intensity discharge (HID) lighting

Power Factor correction:

You want to improve your power factor for several different reasons. Some of the benefits of improving your power factor include:

1. Lower utility fees
2. Increased system capacity and reduced system losses in your electrical system
3. Increased voltage level in your electrical system and cooler, more efficient motors

Thus, it comes as no surprise that one way to increase power factor is to add capacitors to the system. This and other ways of increasing power factor are listed below:

1. Installing capacitors (KVAR Generators) Minimizing operation of idling or lightly loaded motors.
2. Avoiding operation of equipment above its rated voltage.

The Maximum Power Transfer Theorem is not so much a means of analysis as it is an aid to system design. Simply stated, the maximum amount of power will be dissipated by a load resistance when that load resistance is equal to the Thevenin/Norton resistance of the network supplying the power. If the load resistance is lower or higher than the Thevenin/Norton resistance of the source network, its dissipated power will be less than the maximum.
Maximum power is transferred from the source to the load when the resistance of the circuit is equal to the resistance of the load.

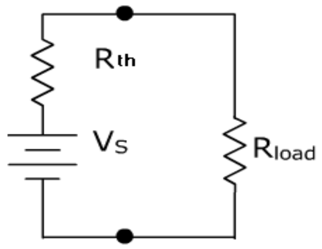


Figure 9.2: $R_{Load} = R_{Th}$

The efficiency factor is based on the percentage of total power generated by the source that is delivered to the load.

$$\text{Efficiency}(\%) = \frac{P_{out}}{P_{in}} \times 100$$

Maximum power is delivered to the load when Rload = Rint, therefore, the efficiency would be equal to 50 percent.

For the sinusoidal steady state:

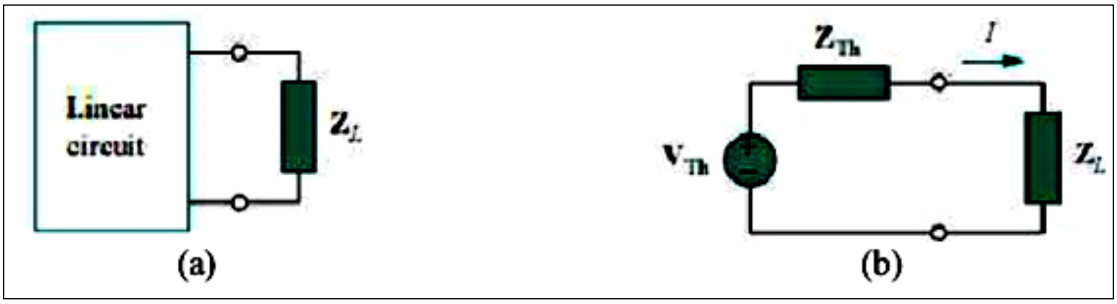


Figure 9.3: Finding maximum average power transfer: (a) circuit with a load, (b) the Thevenin equivalent

Consider the circuit in Figure 9.3, where an ac circuit is connected to a load ZL and is represented by its Thevenin equivalent. The load is usually represented by an impedance, which may model an electric motor, an antenna, a TV, and so forth. In rectangular form, the Thevenin impedance Z_{TH} and the load impedance Z_L are

$$Z_{TH} = R_{TH} + jX_{TH}$$

$$Z_L = R_L + jX_L$$

The current through the load is

$$\boldsymbol{I} = \frac{\boldsymbol{V_{TH}}}{\boldsymbol{Z_{TH}} + \boldsymbol{Z_L}} = \frac{\boldsymbol{V_{TH}}}{\left((\boldsymbol{R_{TH}} + \boldsymbol{jX_{TH}}) + (\boldsymbol{R_L} + \boldsymbol{jX_L})\right)}$$

The average power delivered to the load is

$$\boldsymbol{P} = \frac{\boldsymbol{1}}{\boldsymbol{2}}|\boldsymbol{I}|^2\boldsymbol{R_L} = \frac{\boldsymbol{V_{TH}}}{\boldsymbol{Z_{TH}} + \boldsymbol{Z_L}} = \frac{|\boldsymbol{V_{TH}}|^2\, \boldsymbol{R_L}/\boldsymbol{2}}{((\boldsymbol{R_{TH}} + \boldsymbol{R_L})^2 + (\boldsymbol{X_{TH}} + \boldsymbol{X_L})^2)}$$

Our objective is to adjust the load parameters R_L and X_L so that P is maximum. To do this we set $\partial P/\partial R_L$ and $\partial P/\partial X_L$ equal to zero. From the previous equation, we obtain Setting $\partial P/\partial X_L$ to zero gives $X_L = -X_{TH}$

and setting $\partial P/\partial R_L$ to zero results in $R_L = \sqrt{(\boldsymbol{R_{TH}})^2 + (\boldsymbol{X_{TH}} + \boldsymbol{X_L})^2}$

Combining the two previous equations leads to the conclusion that for maximum average power transfer, ZL must be selected so that $X_L = -X_{TH}$ and $R_L = R_{TH}$

$$Z_L = \boldsymbol{R_L} + \boldsymbol{jX_L} = \boldsymbol{R_{TH}} - \boldsymbol{jX_{TH}} = \boldsymbol{Z_{TH}}^*$$

For maximum average power transfer, the load impedance Z_L must be equal to the complex conjugate of the Thevenin impedance Z_{TH}

➢ Prelab:

1. Make a DC sweep for R_L in the circuit of Figure 9.5 to produce a plot of (P_{LOAD} vs. R_L) for the range of values of R_L shown in Table 9.2, use cursors to find the value of R_L that results in maximum power transfer to R_L.
2. For the circuit of Figure 9.6:
   - Use phasor analysis to calculate V_L and I_L, assume the input voltage $V_{in} = 8\ V_{PP}\angle 0°$.
   - Calculate the complex power of the load" S_L.
   - Calculate the parallel capacitance needed to correct the load power factor to unity.

     Note: Remember to use **RMS** values for these calculations. All phase angles must be referenced to V_s whose angle is zero.
   - Use PSPICE to do transient analysis of the circuit in Figure 9.6, show $I_L(t)$ and $V_L(t)$ on one graph, you will need different Y-axis, and measure the power factor (from the time difference between the two waveforms).
   - Repeat the previous step with the added capacitor to show power factor improvement.
3. For the circuit of Figure 9.8, design a load that is when connected to the output terminals of the circuit will extract maximum average power, then calculate magnitude of P_{Max}.

➢ Procedure:

➢ Part A: DC power measurement

1. Connect the circuit of Figure 9.4

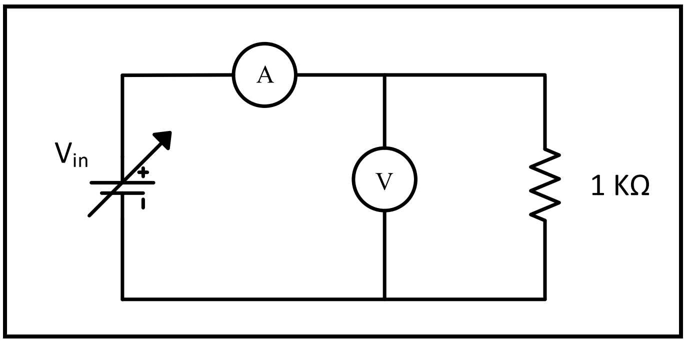


Figure 9.4

2. Measure the resistor current as input voltage is varied according to Table 9.1. Record results in the same table.

Question 1: Calculate the power of R using data in Table 9.1. Record results in Table 9.1.

Question 2: Use excel to draw a graph of (power vs. current). Deduce the relationship between the power and the current through a resistor.

➢ Part B: Maximum DC power transfer

1. Connect the circuit of Figure 9.5, with the output of the power supply set to 15 V.

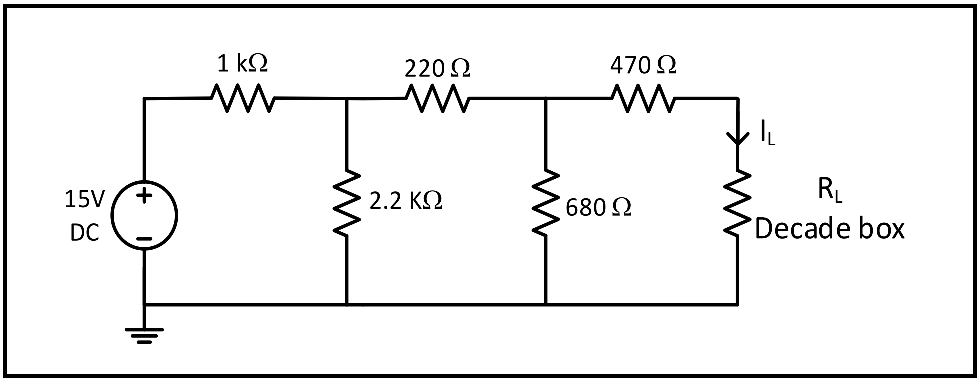


Figure 9.5

2. Measure the current I_L for each value of the load resistor R_L shown in Table 9.2, record results in the same table.
3. Measure the equivalent resistor to the left of variable resistor after killing the source, record result below Table 9.2.

Question 3: Calculate the power dissipated by R_L, record results in Table 9.2.

Question 4: Use excel to plot a graph of power dissipated by the load against corresponding value of load. From the graph determine the load resistance which dissipated maximum power, compare it with the value measured in step 3, and discuss the behavior of the graph.

➢ Part C: Power factor measurement

1. Connect the circuit of Figure 9.6, set the function generator to produce a sinusoidal voltage at $8V_{PP}$ and f = 1 KHz.

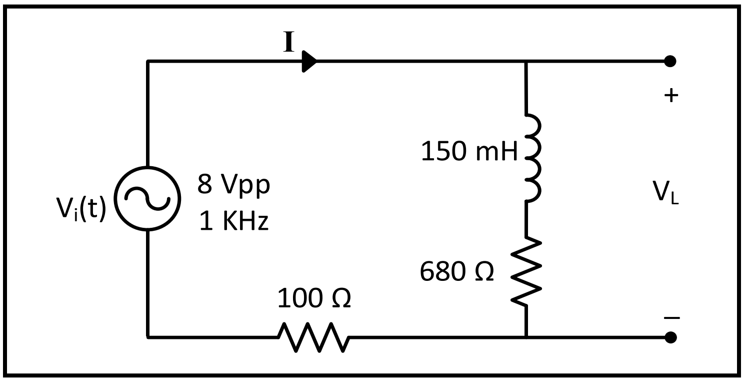


Figure 9.6

2. Connect CH1 of the oscilloscope across Vin and CH2 across the 100 Ω resistor (the voltage across the resistor has the same waveform and angle as the current since V = I x R).
3. Use cursors to measure the time difference between V_{in} and $V_{100\Omega}$, record result in Table 9.3, take a picture of the oscilloscope screen with cursors shown, to be added to the report.

4. Measure the RMS value of: V_L and I **using DMM,** record results in Table 9.3.

Question 5: Calculate the phase shift $\Delta\theta$ between V_{in} and $V_{100\Omega}$ signals using $\{\Delta\theta = 360° \text{ x f x } \Delta t\}$, ($\Delta\theta$ here represents the angle of the current, as the input voltage is assumed to be at zero angle), record results in Table 9.3.

Question 6: Calculate the power factor of the circuit using measured data in Table 9.3, record results in Table 9.3. Indicate whether the load power factor is leading or lagging,

Question 7: Calculate the average power P and the reactive power Q delivered to the load using measured data in Table 9.3, record results in Table 9.3.

➢ PART D: Power factor correction

1. For the circuit of figure 9.6: connect the capacitor value that you calculated in your prelab to correct the power factor to unity, in parallel with the load as shown in figure 9.7.

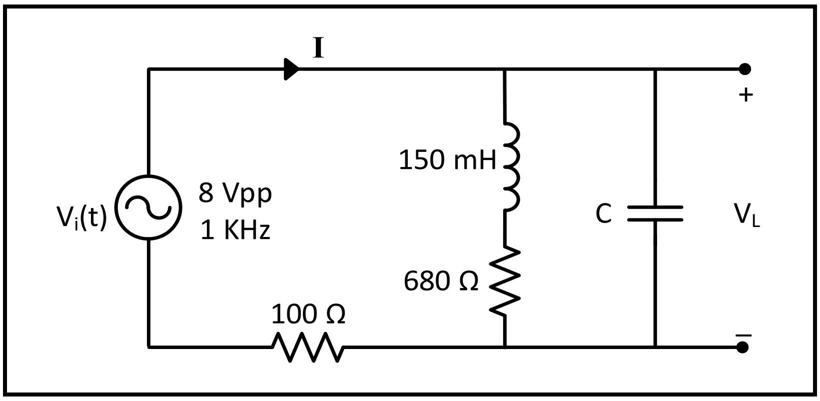


Figure 9.7

2. Connect CH1 of the oscilloscope across Vin and CH2 across the 100 **Ω** resistor, take a picture of the oscilloscope screen to be added to the report.
3. Measure the RMS value of: V_L and I **using DMM,** record results in Table 9.4.

Question 8: Calculate the new: power factor, average power P, and reactive power Q of the load using the data in Table 9.4, record results in the same table. Discuss the effect of adding the capacitor on the measured and calculated values of Table 9.4.

➢ PART E: Maximum average power transfer

1. For the circuit shown in Figure 9.8: connect the load that you designed in your prelab in order to get the maximum power transfer to the load, to the circuit terminals.

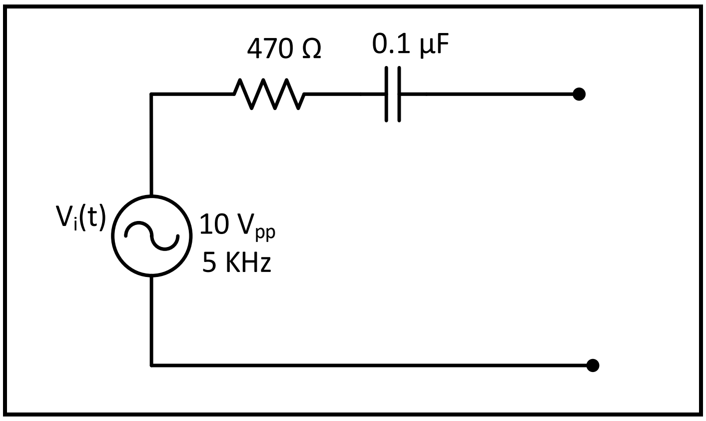


Figure 9.8

2. Measure the load current, record result in the indicated place in data sheet.

Question 9: Calculate the average power that is delivered to the load ($P = I^2 \times R$), compare it with the value of P_{Max} calculated in prelab.

Before Leaving the Laboratory

Be sure the following is completed before you leave the laboratory.

(g) Check to be sure that you have all the required measured values.

(h) Have the laboratory instructor check your laboratory readings.

(i) Restore your laboratory station (equipment and chairs) to the condition they were in when you arrived.

Thank you for your cooperation.

Experiment 10

Three Phase Circuits

➢ Objectives:

To investigate the power, voltages, current, and phase angle of a 3-phase balanced/unbalanced load which is supplied from a 3 phase supply.

➢ Equipment Required:

- Connectors and safety Leads
- Multimeter
- Oscilloscope
- Differential Probe

➢ Introduction:

The three-phase (3ph) system is generated by a special coil arrangement within the alternating current generator (3ph synchronous generator) at the power plant. The generated waveforms, given in figure 10.1, are shifted by 120° from each other's.

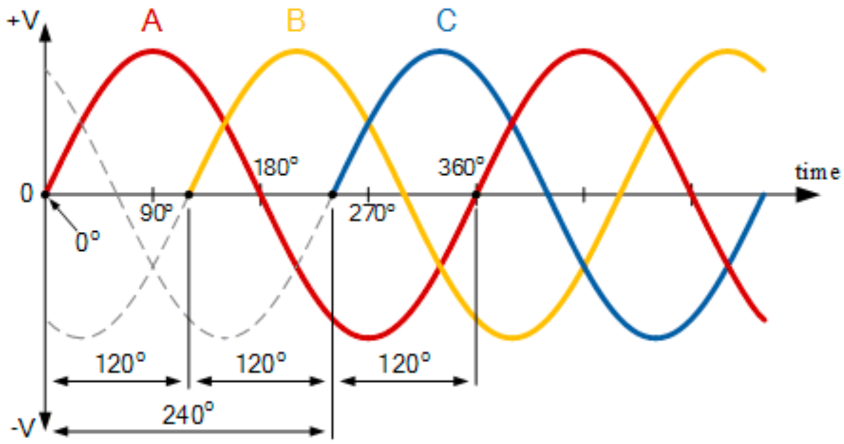


Figure 10.1: three phase voltages

Two connection types are common for the coils of AC generators, the star connection, and the delta connection, see figure 10.2. The phase (ph) and the line-to-line (L) relations in Star and Delta connections (considering ABC sequence) are given in Table 10.A

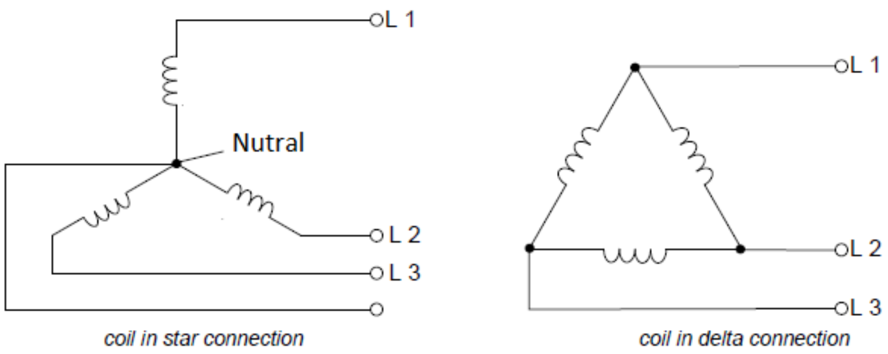


Figure 10.2: Star and Delta connections

Table 10.A: phase and line to line voltages in Star (Y) and Delta connections

Star (Y) connection	Delta connection
$\boldsymbol{V}_L = \sqrt{3}\boldsymbol{V}_{ph}\angle 30^o$	$\boldsymbol{V}_L = \boldsymbol{V}_{ph}$
$\boldsymbol{I}_L = \boldsymbol{I}_{ph}$	$\boldsymbol{I}_L = \sqrt{3}\boldsymbol{I}_{ph}\angle -30^o$

The standard mains supply phase voltage is 230 Vrms (between each phase L1, L2, L3, and the neutral line). However, the line value is 400 Vrms (between L1, L2, and L3).
In this experiment, for safety reasons, a lab test system with low voltage 3-phase generator is used: 230V is replaced by 7V and 400V is replaced by 12V respectively

- **THREE-PHASE POWER:**

The power in a three-phase system is given by:

$$P_{3phase} = P_{phase1} + P_{phase2} + P_{phase3}$$

There are two ways to compute the real power in a balanced three phase system:

1) If line values of voltage and current are known, the consumed real (active) power in a purely resistive load can be computed using the formula:

$$P_{3ph} = \sqrt{3}\, V_{Line} I_{Line}$$

2) If the phase values of voltage and current are known, the consumed real (active) power in a purely resistive load can be computed using the formula:

$$P_{3ph} = 3\, V_{Phase} I_{Phase}$$

Notice that in the first formula, the line values of voltage and current are multiplied by $\sqrt{3}$. In the second formula, the phase values of voltage and current are multiplied by 3.

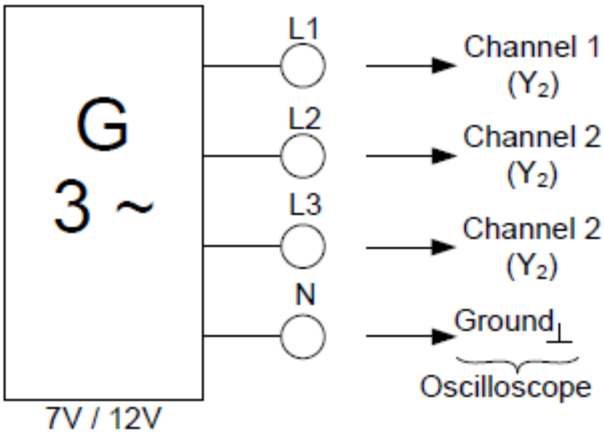
The first formula is used more often because it is generally more convenient to obtain line values of voltage and current, which can be measured with a voltmeter and ammeter.

➢ Procedures:

✓ Part A: **Source**

- Measure the phase voltages in a 3ph system with your oscilloscope and draw the voltage curves.
- Connect channel 1 of your oscilloscope to L1-N
- Connect channel 2 to L2-N and observe the phase shift.

Calculate/fill table 10.1:



- Swap channel 2 to L3-N and determine the sequence (abc or acb) in table 10.2.
- Use **math. Function** ($\boldsymbol{V}_{L1,L2} = \boldsymbol{V}_{L1,N} - \boldsymbol{V}_{L2,N}$) in the oscilloscope to observe and calculate the relation between line-to-line voltage $\boldsymbol{V}_{L1,L2}$ with phase value $\boldsymbol{V}_{L1,N}$.
- Measure with your multimeter the phase voltages and line values and fill them in table 10.3.

- ✓ **Part B: Star-Star system** (balanced and unbalanced load)

  Required Materials:

  - Connectors and Safety Leads
  - 1×Resisitor 330 Ω
  - 1×Resisitor 680 Ω
  - 3×Resisitor 1k Ω
  - 1×Multimeter

- Connect a symmetric load to the circuit as shown in figure 10.3.
- Measure the voltages and currents with your multimeter and calculate the powers. Use table 10.4.
- Respect your measurements and calculations for the unbalanced load and complete table 10.4.

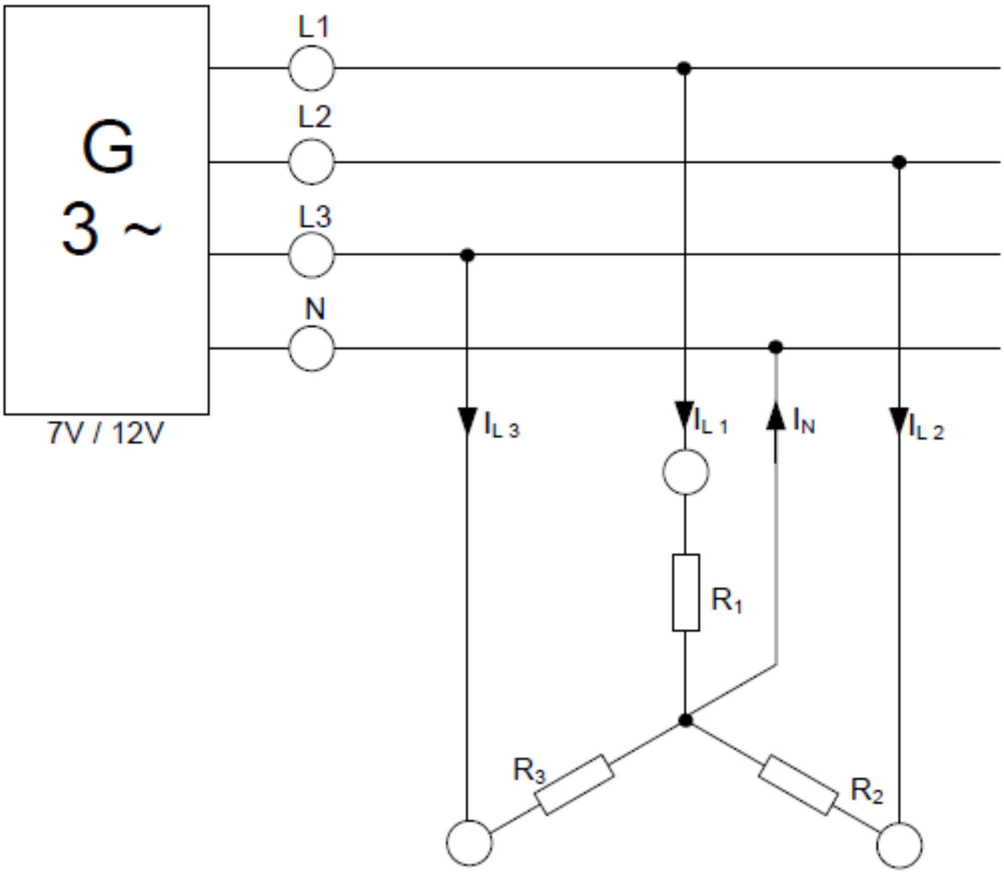


Figure 10.3 Balanced Y-Y system

✓ **Part C: Star-Delta system** (balanced and unbalanced load)

- Connect a symmetric load to the circuit as shown in figure 10.4.
- Measure the voltages and currents with your multimeter and calculate the powers. Use table 10.5.
- Respect your measurements and calculations for the unbalanced load and complete table 10.5.

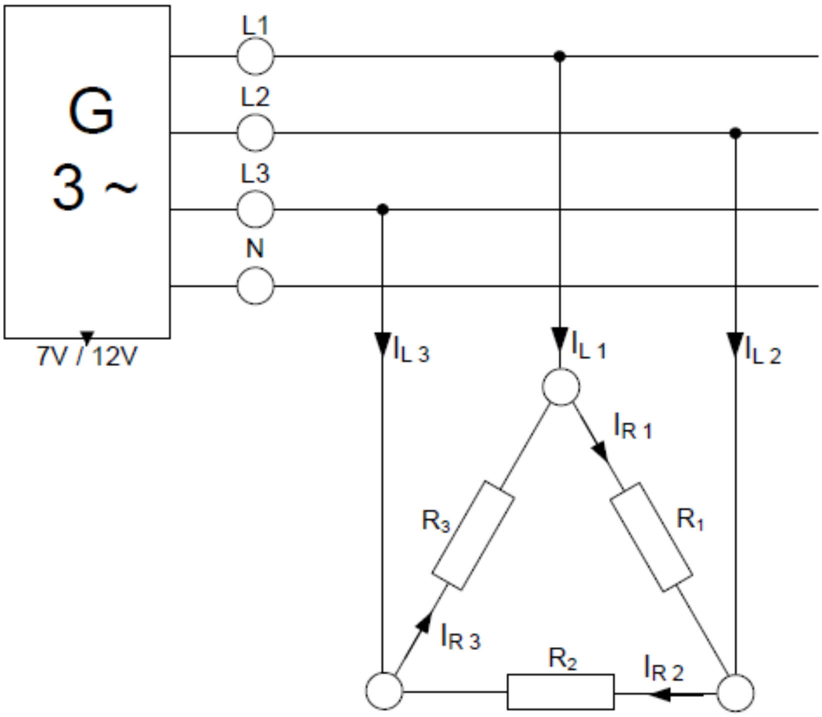
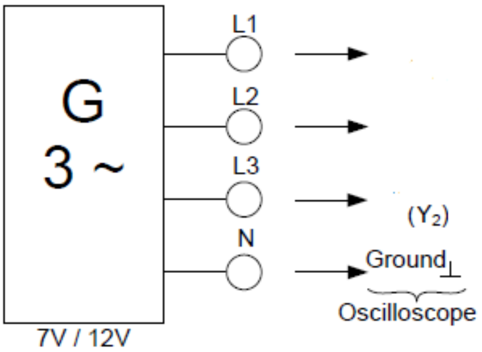


Figure 10.4 Balanced Y-Δ system

In-Lab Task: Use the Differential probe along an oscilloscope to investigate the phase shift between the line-to-line (L1-L2) and line-to-phase (L1-N) voltages of the source.



Experiment # 11

Series and Parallel Resonant Circuits

➢ Objectives:

1. To investigate the characteristic of the Series Resonant Circuits.
2. To investigate the characteristic of the Parallel Resonant Circuits.

➢ Equipment's Required:

1. Digital Multimeter GDM-8135.
2. Oscilloscope TDS 2002B.
3. Function Wave Generator GFG-8215a.
4. Resistors, capacitors, inductors, and wires.

➢ Introduction:

Series Resonance:

Series Resonance:

Resonance in AC circuits implies a special frequency determined by the values of the resistance, capacitance, and inductance. For series resonance, the condition of resonance is straightforward and it is characterized by minimum impedance and zero phase. The resonance of a series RLC circuit occurs when the inductive and capacitive reactances are equal in magnitude but cancel each other because they are 180 degrees apart in phase. The sharp minimum in impedance that occurs is useful in tuning applications. The sharpness of the minimum depends on the value of R and is characterized by the "Q" of the circuit.

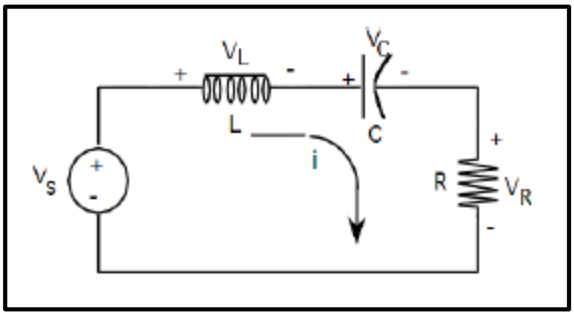


Figure 11.1: Series RLC circuit.

If Vi = V_m.cos(wt), then i = I_m.cos(wt + θ)

The frequency at which the reactances of the inductance and the capacitance cancel each other is the resonant frequency (or the unity power factor frequency) of this circuit. This occurs at:

$$\omega_o = \frac{1}{\sqrt{LC}}$$

Since i = VR /R, then the current i can be studied by studying the voltage across the resistor. The current i has the expression

$$i = I_m \cos(\omega t + \theta)$$

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

$$\theta = -\tan^{-1}\left(\frac{\omega L - \frac{1}{\omega C}}{R}\right)$$

The bandwidth of the series circuit is defined as the range of frequencies in which the amplitude of the current is equal to or greater than (1 / $2^{0.5}$) times its maximum amplitude, as shown in fig. 2. This yields the bandwidth B = w2-w1= R/L where

$$\omega_{2,1} = \sqrt{\left(\frac{R}{2L}\right)^2 + \frac{1}{LC}} \pm \frac{R}{2L}$$

w2,1 are called the half power frequencies or the 3 dB frequencies, i.e the frequencies at which the value of Im equals the maximum possible value divided by $\sqrt{2}$ =1.414 .

The quality factor

$$Q = \frac{\omega_o}{B} = \frac{1}{R}\sqrt{\frac{L}{C}}$$

Then the maximum value of :

1- VR occurs at w= wo
2- VL occurs at $\dfrac{\omega_o}{\sqrt{1 - \frac{R^2 C}{2L}}}$
3- VC occurs at $\omega_o \sqrt{1 - \frac{R^2 C}{2L}}$

The bandwidth from the cutoff frequencies:

$$\beta = \omega_{c2} - \omega_{c1} = \frac{R}{L}$$

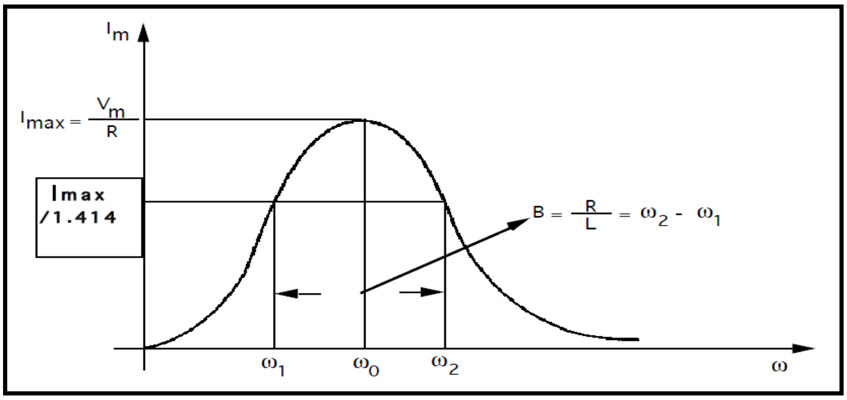


Figure 11.2 Frequency Response of a Series - Resonant Circuit

Parallel Resonance:

The basic parallel-resonant circuit is shown in Figure 11.3. The resonance of a parallel RLC circuit is a bit more involved than the series resonance. The resonant frequency can be defined in three different ways, which converge on the same expression as the series resonant frequency if the resistance of the circuit is small.

One of the ways to define resonance for a parallel RLC circuit is the frequency at which the impedance is maximum. The general case is rather complex, but the special case where the resistances of the inductor and capacitor are negligible can be handled readily by using the concept of admittance.

Defining the parallel resonant frequency as the frequency at which the voltage and current are in phase, unity power factor.

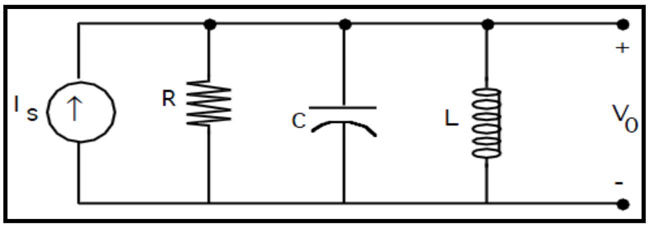


Figure 11.3: The Parallel Resonant circuit.

If Is = Im cos(wt), then Vo = Vm cos(wt+θ) where

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

The resonant frequency is

$$\omega_o = \frac{1}{\sqrt{LC}}$$

The cutoff frequencies are:

$$\omega_{2,1} = \sqrt{\left(\frac{1}{2RC}\right)^2 + \frac{1}{LC}} \pm \frac{1}{2RC}$$

The bandwidth B = w2 - w1 = 1/RC.

The quality factor:

$$Q = \frac{\omega_o}{B} = R\sqrt{\frac{C}{L}}$$

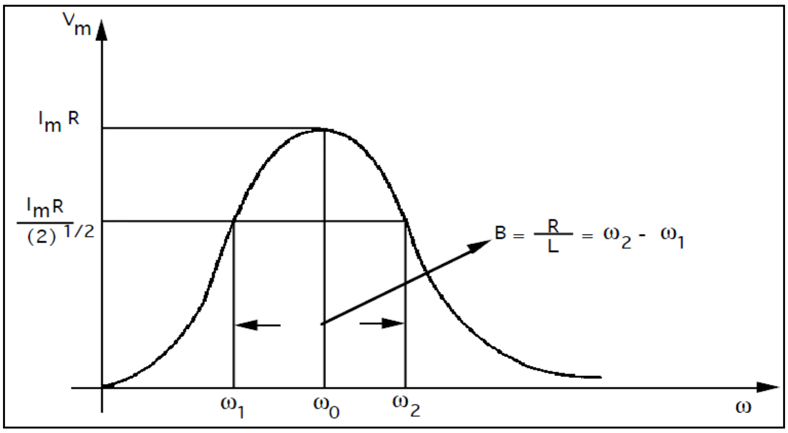


Figure 11.4: Frequency Response of the Parallel - Resonant Circuit

Pre-Lab:

1. Calculate the center frequency (f_0), two cut-off frequencies (f_{C1} and f_{C2}), and bandwidth (β) **in Hertz**, for the circuit in Figure 11.6.
2. Simulate the circuit of figure 11.6 using ac sweep with a Vac source at 1 V amplitude; plot the magnitude of the output voltage (Vo)

(Refer to pages 24-25 in experiment 0, there is an example of this simulation setting).

3. Repeat Step 2 for the circuit in Figure 11.6, using R = 7.1 kΩ.
4. Repeat steps 1 and 2 for the circuit of figure 11.7.
5. Repeat Step 2 for the circuit of figure 11.7, using R = 1.6 kΩ.
6. Repeat Step 2 for the circuit of figure 11.7, using L = 60 mH.

➢ Procedure:

➢ Part A: Impedance and resonance

1. Connect the circuit of Figure 11.5, with a sinusoidal input voltage at 1 V_{RMS}, use decade box for R.

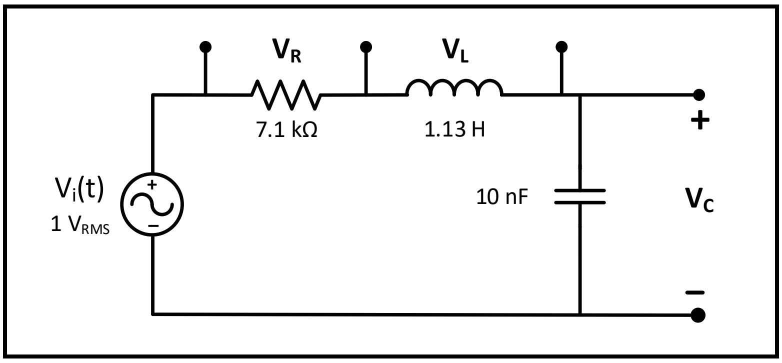


Figure 11.5

2. Change the frequency of the input voltage according to the values listed in Table 11.1, for each value of frequency measure the voltages V_C, V_L, and V_R **using DMM**, record results in the same table.

Question 1: Discuss how the voltages measured in step 2 change compared to each other as the frequency changes.

➢ Part B: Series Resonant Circuits

1. Connect the circuit of Figure 11.6, with a sinusoidal input voltage at 1 V_{RMS}, use decade box for R.

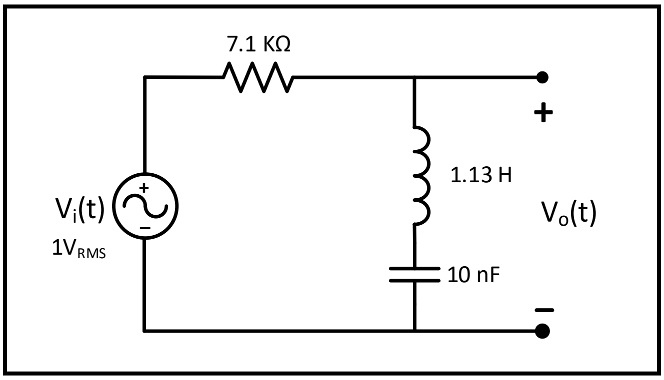


Figure 11.6

2. Connect DMM across V_O, increase frequency starting from a low value (100 Hz) to a high value (10 kHz) while monitoring V_O, notice the range of frequencies where V_O is minimum (near to 0 V), and the range where V_O is maximum (near to 1 V).
3. To determine resonant frequency (f_0) **experimentally**, change frequency until V_O is around $V_{Min} = 0$, at this point (f) is resonant frequency (f_0) record its value in Table 11.2.

4. To determine cutoff frequencies (f_{C1} and f_{C2}) **experimentally**, change the frequency until V_O is around $(V_{max}/\sqrt{2})$, there will be two points, record two values of (f) in Table 11.2.
5. Connect **DMM** across V_O, then change the frequency in steps so that V_O is equal to the values listed in Table 11.2, record the corresponding values of frequency in Table 11.2.

Question 2: In your report use data in Table 11.2 to plot $\{(V_O) \text{ vs. } (f)\}$ using excel. From data calculate the bandwidth $(\beta = f_{C1} - f_{C2})$ and discuss the plot's behavior comparing experimental values of (f_0), (f_{C1} and f_{C2}), and (β) to their theoretical value.

Question 3: If we change the value of R (increasing or decreasing) discuss the effect of changing R on the values of (f_0), (f_{C1} and f_{C2}), and (β).

Part C: Parallel Resonant Circuits

1. Connect the circuit of Figure 11.7, with a sinusoidal input voltage at 1 V_{RMS}, use decade box for R.

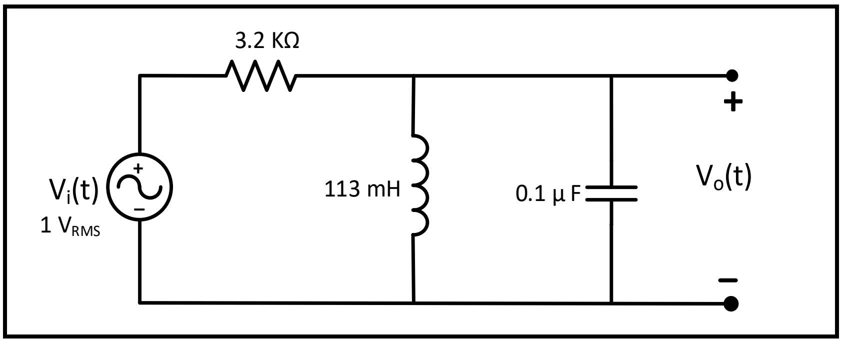


Figure 11.7

2. Connect **DMM** across V_O, increase frequency starting from a low value (100 Hz) to a high value (30 kHz) while monitoring V_O **using DMM**, notice the range of frequencies where V_O is minimum (near to 0), and the range where V_O is maximum (near to 1 V).
3. To determine resonant frequency (f_0) **experimentally**, change frequency until V_O is around V_{Max} = 1 V, at this point (f) is resonant frequency (f_0) record its value in Table 11.4.
4. To determine cutoff frequencies (f_{C1} and f_{C2}) **experimentally**, change the frequency until V_O is around $(V_{max}/\sqrt{2})$, there will be two points, record two values of (f) in Table 11.4.
5. Connect the **DMM** across V_O, then change the frequency in steps so that V_O is equal to the values listed in Table 11.4, record the corresponding values of frequency in Table 11.4.

Question 4: Repeat same procedure in question 2 for data in Table 11.4.

Question 5: If we change the value of R (increasing or decreasing), discuss the effect of changing R on the values of (f_0), (f_{C1} and f_{C2}), and (β).

Question 6: If we change the value of L (increasing or decreasing), discuss the effect of changing L on the values of (f_0), (f_{C1} and f_{C2}), and (β).

Before Leaving the Laboratory

Be sure the following is completed before you leave the laboratory.

(j) Check to be sure that you have all the required measured values.

(k) Have the laboratory instructor check your laboratory readings.

(l) Restore your laboratory station (equipment and chairs) to the condition they were in when you arrived.

Thank you for your cooperation.

Experiment 3 - Data Tables:

Part A: Kirchhoff's Laws

Table 3.1: Voltages and currents for circuit of figure 3.4

	2.2 KΩ	680 Ω	4.7 KΩ	1 KΩ	470 Ω	220 Ω
Voltage [V]						
Current [mA]						

Part B: Voltage Divider

Table 3.2

Voltage [V]	V_{1-2}	V_{2-3}	V_{3-0}

Table 3.3: potentiometer values

R_{AB} [KΩ]	R_{BC} [KΩ]

Table 3.4: Voltage measurements as the resistance change from 1k to ∞

R_L	Open circuit	500 KΩ	100 KΩ	10 KΩ	1 KΩ
V_O [V]					

Part C: Current Divider

Table 3.5

I_s [mA]	I_1 [mA]	I_2 [mA]	I_3 [mA]	I_{23} [mA]

Part D: Short-and-Open Circuited Resistor in Series-Parallel Circuits

Table 3.6: Short Circuit with R_1, R_2 and R_3

	Normal	R_1 S C	R_2 S C
I_1 [mA]			
I_2 [mA]			
I_3 [mA]			
V_{1-2} [V]			
V_{2-3} [V]			
V_{3-0} [V]			

Table 3.7: Open Circuit with R_1, R_2 and R_3

	R_1 O C	R_2 O C
V_{1-2} [V]		
V_{2-3} [V]		
V_{3-0} [V]		
I_1 [mA]		

Experiment 4 - Data Tables:

Part A: The Proportionality Theorem

Table 4.1

Vi	Vo [V]	$(K = V_O / V_{in})$
5 V	0.415	K=0.415/5 ⇒ K=0.083
10 V	0.836	K=0.836/10 ⇒ K=0.0836

Mohammad
9/10/2024

Part B: The Superposition Theorem

Table 4.2

	V (4.7 kΩ) [V]	I (4.7 kΩ) [mA]
Total Response $(V_1 = 20\text{ V}, V_2 = 15\text{ V})$	14.31	3.07
Response 1 $(V_1 = 20\text{ V}, V_2 = 0\text{ V})$	3.477	0.75
Response 2 $(V_1 = 0\text{ V}, V_2 = 15\text{ V})$	10.825	2.33
Response 1 + Response 2	14.302	3.08

Part C: Thevenin Theorem

Table 4.3

V (330 Ω) [V]	I (330 Ω) [mA]
2.388	7.25

Table 4.4

V (open circuit) [V]	I (Short circuit) [mA]	R_{TH} (Calculate) [Ω]
7.692	10.47	$R_{th} = \frac{V}{I}$ ⇒ R_{th} = 735 Ω

Table 4.5

V_{Test} (SET)	I_{Test} (Measure) [mA]	R_{TH} (Calculate) [Ω]
4 V	5.56	$Rth = \frac{V}{I} \Rightarrow Rth = 719\ \Omega$

Table 4.6

R_{Th} [Ω]	0.724K ⇒ 724 Ω

Table 4.7

V (330 Ω) [V]	I (330 Ω) [mA]
2.372	7.21

Mohammad

9/10/2024

Part D: The Reciprocity Theorem

Table 4.8

I_1 [mA]	I_2 [mA]
2.99	2.98

Experiment 5 - Data Tables:

Part A: Capacitance

Table 5.1

Time (s)	Current (μA)	Voltage on C_1 (V)	Charge (μC)
0	10	0	
2			
4			
6			
8			
10			
12			
15			
18			
21			
24			
27			
30			
35			
40			
60			
80			
90			

Part B: Capacitors in series

Table 5.2

Time (s)	Current (μA)	Charge (μC)
0	10	
2		
4		
6		
8		
10		
12		
14		
16		
18		
20		
24		
28		
35		
40		

Table 5.3

At t = 40 s

V_{C2}	V_{C1+C2}	V_{C1}

Part C: Capacitors in parallel

Table 5.4

Time (s)	Current (μA)	Charge (μC)
0	10	
3		
6		
9		
12		
15		
20		
25		
30		
35		
40		
50		
60		
70		
90		
120		

Table 5.5

V_{PC}	

Experiment 6 - Data Tables:

Part A: Step response of first-order RC circuit

Table 6.1

R_{Actual}	

Part B: Step response of first-order RL circuit

Table 6.2

R_{Actual}	$R_{Inductor}$

Experiment 7 - Data Tables:

Part A: Step response of second-order Series RLC circuit

Case C: under damped response

$R_{Inductor}$ [Ω]	

Table 7.1

V_a	t_a	V_b	t_b	$V(\infty)$

Part B: Step response of second-order parallel RLC circuit

Case A: under damped response

$R_{Inductor}$ [Ω]	

Table 7.2

V_a	t_a	V_b	t_b	$V(\infty)$

Experiment 8 - Data Tables:

Part A: Impedance Measurement

1.

Table 8.1

f [Hz]	250	500	1000	2000
|V|				
|I|				
$\|Z_R\|$				

2.

Table 8.2

f [Hz]	250	500	1000	2000
|V|				
|I|				
$\|Z_C\|$				

3.

Table 8.3

f [Hz]	250	500	1000	2000
|V|				
|I|				
$\|Z_L\|$				

4.

Table 8.4

f [Hz]	250	500	1000	2000
|V|				
|I|				
$\|Z_{RC}\|$				

Part B: Phase Measurement

1.

Table 8.5

Δt	$\Delta\theta$

2.

Table 8.6

Δt	$\Delta\theta$

Part C: Inductance and Capacitance Measurement

1.

Table 8.7

f [Hz]	$\lvert V_R \rvert$	$\lvert V_L \rvert$	$\lvert I \rvert$	L
1 kHz				

2.

Table 8.8

F	$\lvert V_R \rvert$	$\lvert V_C \rvert$	$\lvert I \rvert$	C
1 kHz				

Experiment 9 - Data Tables:

Part A: DC power measurement

Table 9.1

V_{in} [V]	0	2	4	6	8	10
I [mA]						
P [mW]						

Part B: Maximum DC power transfer

Table 9.2

R_L	0	100	400	700	800	850	900	1k	1.1k	1.3k	1.5k
I [mA]											
P [mW]											
Req											

Part C: Power factor measurement

Table 9.3

Measure			Calculate			
V_L	I	Δt	θ_L	Pf	P [mW]	Q [mVAR]

Part D: Power factor correction

Table 9.4

Measure			Calculate			
V_L	I	Δt	θ_L	Pf	P [mW]	Q [mVAR]

PART E: Maximum average power transfer:

I_L=	

Experiment 10 - Data Tables:

Table 10.1: Three Phase Data

Time-period T	
Time shift between the two waveforms Δt	
Phase shifting angle	
Source frequency	

Table 10.2: Sequence

Sequence abc or acb?	

Table 10.3: Line and Phase Voltages

$\boldsymbol{V}_{L1,N} =$	$\boldsymbol{V}_{L1,L2} =$
$\boldsymbol{V}_{L2,N} =$	$\boldsymbol{V}_{L2,L3} =$
$\boldsymbol{V}_{L3,N} =$	$\boldsymbol{V}_{L3,L1} =$
Calculate the linkage factor defined as $\boldsymbol{V}_{L,L}/\boldsymbol{V}_{L,N} =$	
Calculate the peak value of $\boldsymbol{V}_{L,N} =$	
Calculate the peak value of $\boldsymbol{V}_{L,L} =$	

Table 10.4: Balance and Unbalanced loads for Y-Y Connection

Y-Y connection		Balanced Load	Unbalanced Load
Resistor	R_1	1 kΩ	1 kΩ
	R_2	1 kΩ	680 Ω
	R_3	1 kΩ	330 Ω
Phase Currents	I_{L1}		
	I_{L2}		
	I_{L3}		
	I_N		
Line Voltages	$V_{L1,L2}$		
	$V_{L2,L3}$		
	$V_{L3,L1}$		
Phase Voltages	$V_{L1,N}$		
	$V_{L2,N}$		
	$V_{L3,N}$		
Phase Power	P_{R1}		
	P_{R2}		
	P_{R3}		
Total Power	P_{3ph}		

Table 10.5: Balance and Unbalanced loads for Y-Δ Connection

Y-Δ Connection		Balanced Load	Unbalanced Load
Resistor	R_1	1 kΩ	1 kΩ
	R_2	1 kΩ	680 Ω
	R_3	1 kΩ	330 Ω
Line Currents	I_{L1}		
	I_{L2}		
	I_{L3}		
Phase Currents	I_{R1}		
	I_{R2}		
	I_{R3}		
Line Voltages =phase voltages	$V_{L1,L2}$		
	$V_{L2,L3}$		
	$V_{L3,L1}$		
Phase Power	P_{R1}		
	P_{R2}		
	P_{R3}		
Total Power	P_{3ph}		

Experiment 11 - Data Tables:

Part A: Impedance and resonance

For the circuit of figure 11.5

Table 11.1

f [Hz]	30	400	700	1 k	1.2 k	1.5 k	2 k	2.5 k	3.5 k	6 k	80 k
V_R [V_{RMS}]											
V_L [V_{RMS}]											
V_C [V_{RMS}]											

Part B: Series Resonant Circuits

For the circuit of figure 11.6

For R = 7.1 kΩ

Table 11.2

Vo [V_{RMS}]	V_{max}	$V_{max}/\sqrt{2}$	0.5	0.3	0.15	V_{min}	0.15	0.3	0.5	$V_{max}/\sqrt{2}$	V_{max}
f [kHz]		f_{c1}				f_0				f_{c2}	

Part C: Parallel Resonant Circuits

For the circuit of figure 11.7

For R = 3.2 kΩ

Table 11.4

Vo [V_{RMS}]	0.03	0.15	0.3	0.5	$V_{max}/\sqrt{2}$	V_{max}	$V_{max}/\sqrt{2}$	0.5	0.3	0.15	0.03
f [kHz]					f_{c1}	f_0	f_{c2}				