York University
Dept. of Computer Science and Engineering

A laboratory Manual for Electric Circuits Lab
EECS2200

Prepared By

Prof. Mokhtar Aboelaze

September 2014

ACKNOWLEDGEMENT

PREFACE

This laboratory manual is intended for use in EECS2200 Linear circuit course. Every care was taken in preparing this manual, however no one is perfect. If you find any typos or errors in this manual, contact the course director.

To the student:

The objective of this lab is to get you familiar with the instruments used in electric and electronic circuits measurements and testing. It will introduce you to the concept of "lab book" and to how to design, implement and test simple electric circuits.

The lab will be done in groups of 2. Each lab consists of 2 parts. The prelab part will be done before you arrive to the lab. It will be submitted at the beginning of every lab. Then you have to do the experiment and take results. At the beginning of the next lab you should submit the lab report for the previous lab.

Each lab covers a specific topic in the course that will be clear from the lab title. It is your responsibility to read the theoretical part from the text book and the course notes before you go to the lab.

After you connect the circuit on the breadboard, check with the TA before connecting power. Please read the safety rules and troubleshooting hints before you start your first lab. Please be alert and use common sense during the experiment.

You have to maintain a laboratory book or journal, the TA must sign each page before you leave the lab. The journal will be checked once or twice during the term in order to be sure that you successfully did that part. Tips for maintaining a good journal are explained in this manual, please read the chapter titled lab notebook carefully.

The TA is there to help you, if you have any question ask the TA. A simple question might save you a lot of time and trouble later. Remember, you are dealing with expensive equipment.

To the TA:

Please read the experiment before you come to the lab. In the lab you have to approve the schematic diagram and the circuit connection before the students power up the experiment. Your job is to prevent any accidental mishaps that might injure students or destroy any equipment.

To the course Director:

The course director responsibility is to be sure that the lab is properly equipped, the TA is qualified to run the lab, the marked reports are returned to the students in a timely manner, and supervising the TA.

TABLE OF CONTENTS

Laboratory Notebook

Keeping a complete and accurate lab notebook is a very important part of your engineering education. Lab notebooks are a complete record of what you do in the lab. You record in it your thoughts, experiments you ran, assumptions, and results. Your lab notebook can play a vital role in protecting your intellectual property (IP). Although it is highly unlikely that you need IP protection for your experiments in this course, getting used to maintain a good lab notebook is a very important part of your engineering education).

If you ask an IP lawyer what to write in the notebook? The answer would be "everything". However, we will try to simplify this for you.

First a notebook must be bound not spiral, the idea is that in a bound notebook you can't remove (or add) pages without being clearly noticed.

Bound notebook
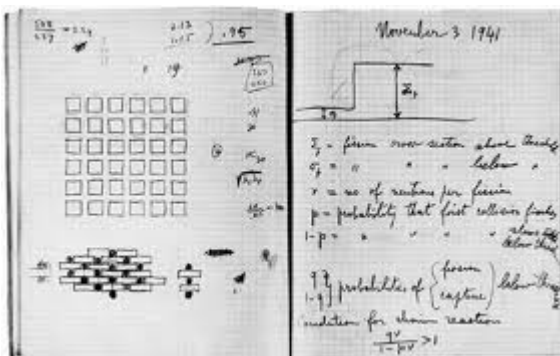
Enrico Fermi lab notebook (from University of Chicago library)

In your notebook you must include your thoughts, what do you want to achieve from the experiment together with the results.

A good rule of thumb (actually 2 rules) are the following:

- Can someone with your technical background read the notebook, understand what you did, and can reproduce the same results.
- If you come back 6 months from now, can you read your notebook and understand your thoughts at the time, and being able to modify or expand on your work

The notebook is not only used to protect your IP, but it is also used to document compliance with procedures especially testing procedures, safety standards, or environmental protection regulation and standards. Every page in the notebook must be sequentially numbered signed, and dated (use unambiguous dating, for example use May, 8, 2012 not 5/8/2012 since the later may means May 8th or August 5th depending on where you are in the world). Use pens not pencils in your notebook. Errors must be crossed with a single line not obliterated. Sometimes you can learn a lot from your mistakes. Any alteration to the notebook must be signed and dated. Also, it is a good idea to leave few blank pages at the beginning of the notebook; you might want to make a table of contents later. What to include in the notebook can be detailed by the 5 W's used by newspapers. The list is very exhaustive and is given only for information purpose; in this course we will use a simpler model. For more information about the lab notebook please take a look at the following reference.

 J. B. McCormack et al "The complementary roles of laboratory notebooks and laboratory reports" IEEE Transactions on Education. Vol. 34, No. 1 February 1991 pp 133-137

The 5 W's are

Who

Experimenters

Sponsors

Witnesses

What

Brainstorming possible solutions

Experimental design

Results: samples and raw data

Analysis of data

Difficulties encountered

When

Projected completion date

Explanation of delays

Progress

Where

Location of equipment

Location of models and samples

Address of author

Why

Statement of problem

Authorization and funding

Rational for engineering decisions

In this course, you are responsible for the following in your notebook

- Numbering, dating and signing every page
- Use X or Z for parts that you did not use in the page
- Start every experiment on a new page
- Include experiment title, objectives, data collected, and observation (any deviation from what you expected, any difficulties you faced, …)
- Describe your results.

In this course, you have to write in the notebook the values of the components you are using; for every component, write the nominal value and the value that you actually measured (there is a tolerance between these two values). Failing to do this, you will get ZERO in the experiment.

Resistor Color Code

There are many types of resistors, both fixed and variable. Axial-lead resistors (mainly used in labs using breadboard), surface mount resistors (used in printed circuit boards), variable resistor (potentiometer), and thermistor (thermistor is a resistor its value depends on the temperature and could be used to measure temperature). In this lab we will be dealing mainly with axial-lead carbon resistor.
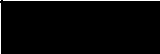

Figure 1. Different types of resistors

You can get the value of the axial-lead resistor using the color-code. The color code is a 4-bands of color (5 bands for precision resistors) that determine the value of the resistor.

For 4 colors resistors, the first two colors determine the digits, the third one is the multiplication factor, and the last one is the tolerance. For 5 colors, the first three colors determine the digits; the last two are for multiplication factor and tolerance. The value of every color is shown in the following table

Color		value	MULTIPLIER	Tolerance
BLACK		0	$X10^0$	
BROWN		1	$X10^1$	
RED		2	$X10^2$	
ORANGE		3	$X10^3$	
YELLOW		4	$X10^4$	
GREEN		5	$X10^5$	
BLUE		6	$X10^6$	
VIOLET		7	$X10^7$	
GRAY		8	$X10^8$	
WHITE		9	$X10^9$	
Gold		-	$X10^{-1}$	$\pm$ 5%
Silver		-	$X10^{-2}$	$\pm$ 10%
None		-	-	$\pm$ 20%

For example

For example: consider the image to the right, the bands are Brown (1), green (5), brown (1), gold (5%)

$15 \times 10^1 = 150$ ohm with 5% tolerance

15×10^1 5% tolerance

Figure 2. An axial lead resistor

Every resistor has a nominal power; you have to be sure that the resistor you are using is rated for the power generated in your circuit.

Capacitors

There are three basic types of capacitors, electrolytic capacitors, film capacitors, and ceramic disk capacitor

Disk ceramic   Electrolytic   Film capacitor

Figure 3. Different types of capacitors
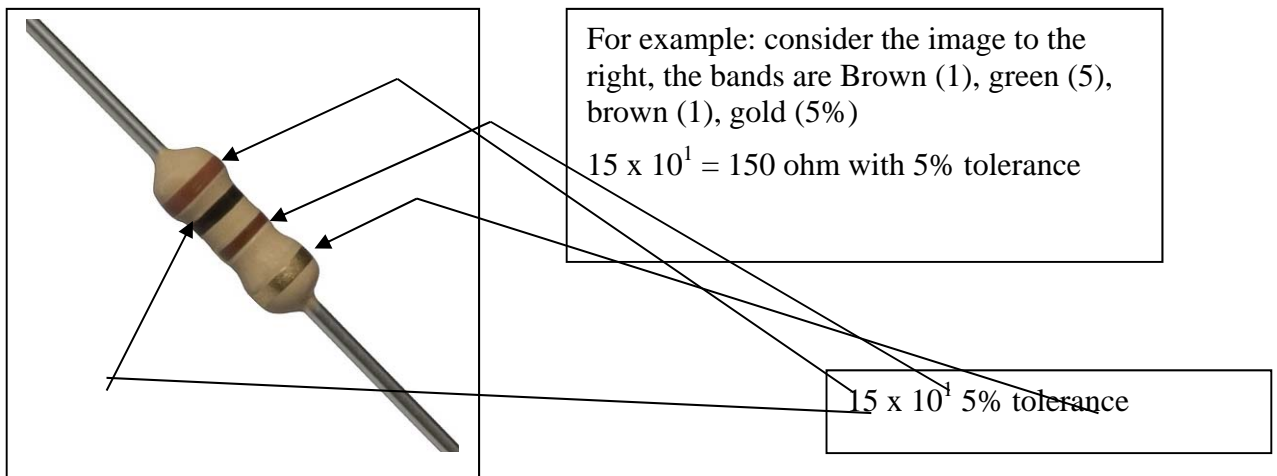
Electrolytic capacitors are made of aluminum or tantalum plates. A very thin layer of insulation is formed electrolytically (electrochemical reaction) on one of the plates. This layer could be very thin, so a large capacitors could be made. The tolerance of the capacitor value is not good though.

Another point with electrolyte capacitors is they have to be connected in a specific way (one plate to the positive voltage and the other to the negative voltage). If they were wrongly connected, the chemical process forming the insulation may be reversed. ***The long lead is connected to the positive voltage***.

Disk ceramic capacitors use ceramic for insulation, ceramic has a very high dielectric constant (100-200 times that of regular plastic). They are suitable for high frequency.

Film capacitors have a very good performance with good tolerance. They are bulky though.

Capacitors usually display their values. For large capacitors, the value may be written on the capacitor (22µF). Smaller capacitors display the values in a compact form. They use three digits and a letter (x y z) and a letter. The three digits display the value in pF as xy *

10^z pf (for example 234 = $23*10^4$ pF = 230 nF. The letter indicates the tolerance (J=5%, K=10%, M=20%). Also the operating voltage is written. For example

123K 330V means $12*10^3$pF=12 nF with10% tolerance and max. V=330V

Breadboards

Breadboard is a plastic board with holes in it that are used to prototype simple circuits. IC chips are inserted in these holes and wires are used to connect them according to the schematic. In this lab, you will be using breadboards to implement the circuit in every lab session.
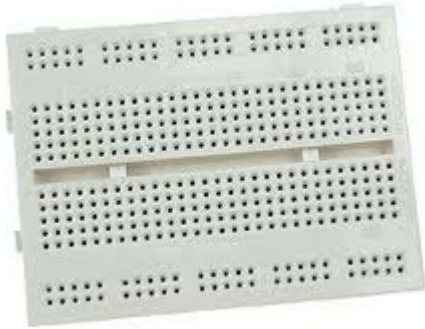
Figure 4. Breadboard

If you open the breadboard, you will find that these holes are connected with wires. For example, in Figure 5 below, all the holes in the top row are connected together. The same for the second top row, the bottom row and the second bottom row as shown in the figure below. These rows are usually used for power rails. For example the top row is connected to the power supply and the second top row to ground. If you want to connect power to any location in the breadboard, connect a wire to one of the holes in the top row and connect the other end of the line to where you want power to be

Usually used for power rail

Figure 5. B readboard showing the power rail

The middle matrix of holes are used for connection. Each column of holes in one side is connected internally by a wire as shown below (blue lines).

Figure 6. Breadboard showing the matrix connection

Figure 7. Breadboard with the connection

ERRORs and ERROR PROPAGATION

Introduction

When we measure something, there is always an inherent error in the measurement. Here, we would like to differentiate between accuracy and precision.

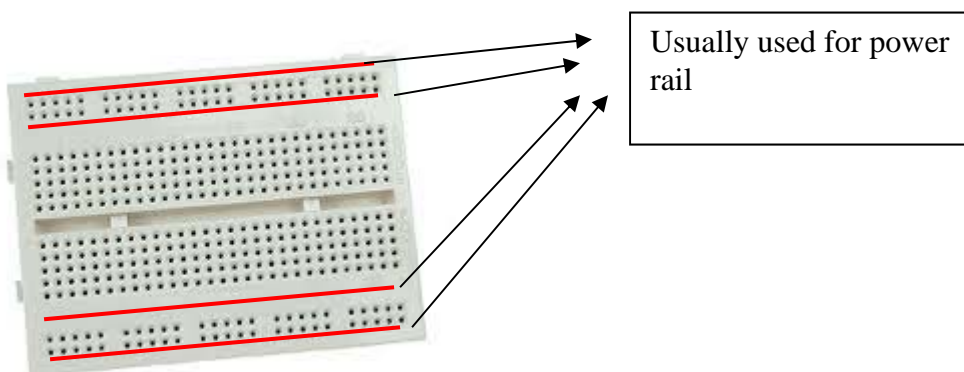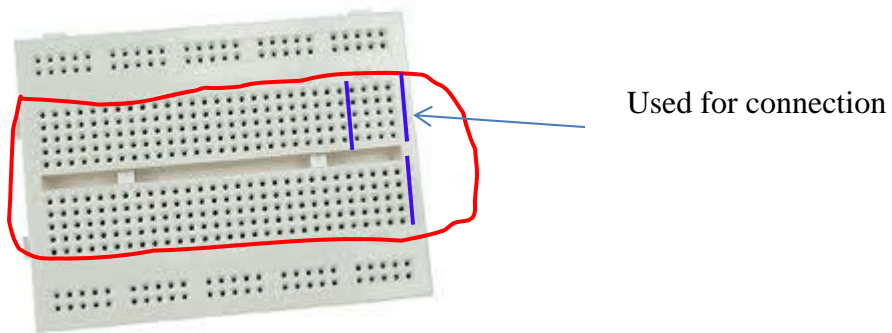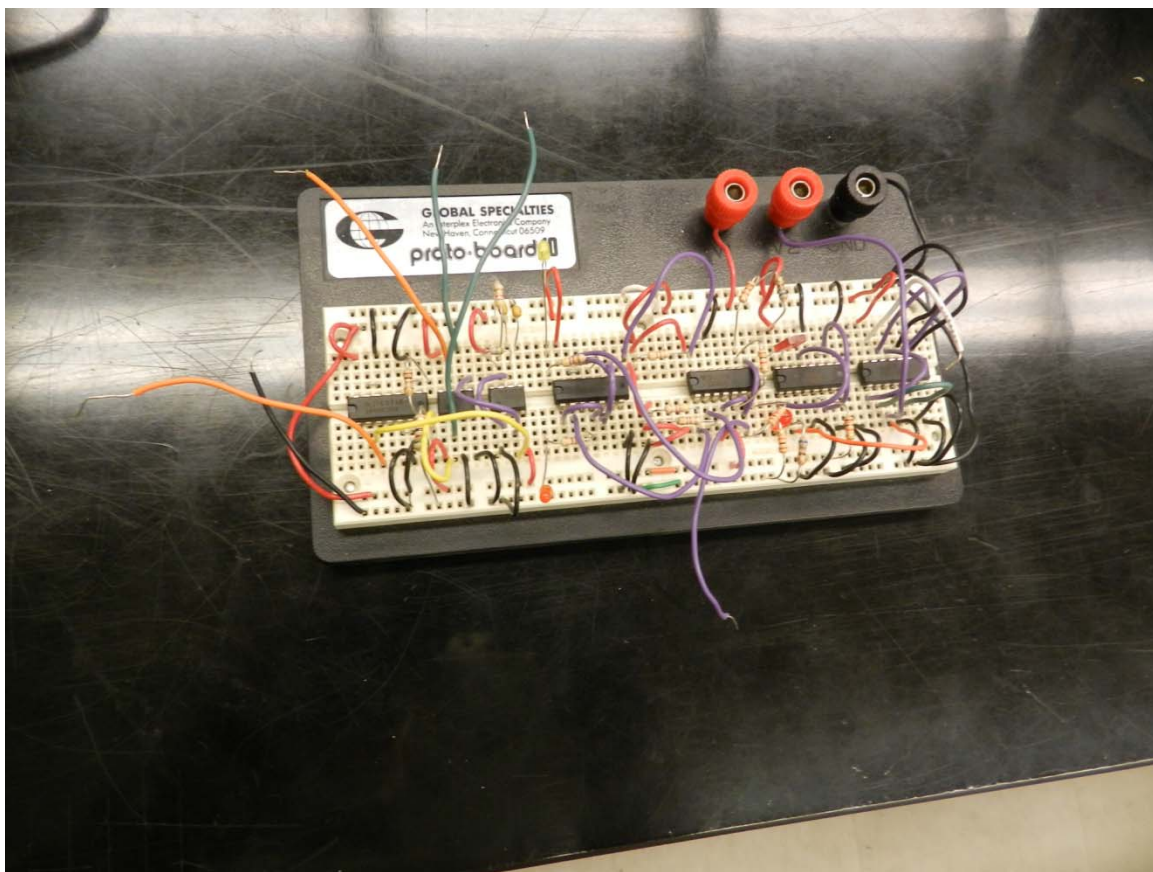Accuracy: accuracy refers to how close the measured value to the correct value.

Precision: precision refers to how close different measurements agree with each other. A classic example is used to differentiate between these 2 quantities is shown below.

Accurate not precise	Precise not accurate	Accurate and precise
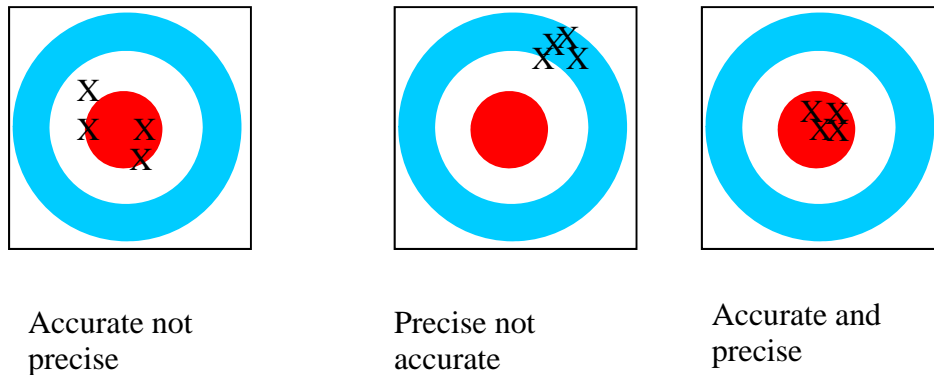
Figure 8. The difference between accuracy and precision

The error could be because of the inherent variability of the measured quantities (measuring pollution in the air by taking air samples will change from a place to a place). Or, if we measure the resistance of a resistor, there is always variations in the resistor produced by the same process and have the same nominal value. Another source of error is the measuring device. The measuring device may introduce some error into the measurements. Another source of error is visual interpolation, when you look at the hand of an analog device.

So, we assume that the measured quantity is

Measured value = true value ± error

Error propagation

Now we know the error associated with a measurement, but what if we use these measurements to calculate other values, what is the error in the calculated value?

For example, we measured two lines, the first line is $x_1 = 10 \pm 1$ and the second is $x_2 = 15 \pm 2$. We know for sure that the first line is between 9 and 11, the second line is between 13 and 17. What is the error in the variable $y = x_1 + x_2$

We know for sure that y is between 22 and 28. To make this more formally,

Assume that the quantities X,Y, and Z are measured, with error δX, δY, and δZ respectively, then

Addition:

$$R = X + Y - Z$$

$$\delta R = \delta X + \delta Y + \delta Z$$

That equation puts an upper bound on the error. If we assume that the error is independent and random, a better approximation is given as

$$\delta R = \sqrt{(\delta X)^2 + (\delta Y)^2 + (\delta Z)^2}$$

So, if we consider the above example, the error is

$$\delta R = \sqrt{(1)^2 + (2)^2} = \sqrt{5} = 2.24 \text{ and the length is } 25 \pm 2.24 \text{ i.e. between 22.76 and 27.24}$$

Multiplication or division

$$R = \frac{XY}{Z}$$

$$\frac{\delta R}{|R|} \approx \frac{\delta X}{|X|} + \frac{\delta Y}{|Y|} + \frac{\delta Z}{|Z|} \qquad \text{"approximation"}$$

$$\delta R = |R| \sqrt{\left(\frac{\delta X}{X}\right)^2 + \left(\frac{\delta Y}{Y}\right)^2 + \left(\frac{\delta Z}{Z}\right)^2}$$

Multiplication by a constant

$$R = CX$$

$$\delta R = |C| \delta X$$

Polynomial functions

$$R = X^n$$

$$\delta R = \left|n\right| \frac{\delta X}{|X|} \cdot |R|$$

General function

If the function we are calculating could be expressed as a general function of X, Y, Z, …
Then

$R = f(X, Y, Z, \cdots)$

$$\delta R = \sqrt{\left(\frac{\partial f}{\partial X} \cdot \delta X\right)^2 + \left(\frac{\partial f}{\partial Y} \cdot \delta Y\right)^2 + \left(\frac{\partial f}{\partial Z} \cdot \delta Z\right)^2 + \cdots}$$

The rules we mentioned above are a special case of this rule.

Sometimes engineers do not specify the precision of the quantity, instead they imply it by using a specific number of significant digits, for example 123400, or 123.4, or 0.001234 all of these implies 4 significant digits. The implied error here is one half the unit in the right most significant digit

123400 implies an error of 50, that is 123400 ± 50 i.e. between 123350 and 123450

123.4 implies 123.4 ± 0.05 i.e. 123.35 and 123.45

0 .001234 implies 0.001234 ± 0.0000005 i.e. between 0.0012335 and 0.0012345

When performing arithmetic operations on such numbers, the rule of thumb is for addition and subtraction we round the result to the last decimal place in the least precise number. While in multiplication or division, we round the result to a number of significant figures that is equal to the number of significant figures in the number with the least number of significant figures.

Fopr example

   12.012

+ 2.1

   7.23456

   21.34656   that is rounded to 21.3 (2.1 contains one decimal place)

While if we multiply

  6.1234

X

  2.83

17.329222   rounded to 17.3

if we multiply

1234.23

2.83

3492.8709 rounded to 3490

Random errors

Random errors result from many sources

Human errors: A good example is when a human is measuring time using a stop watch. His/her reflexes may not be very sharp, the measured period of time may vary from person to a person, and for the same person by repeating the experiment many times.

Variations in the quantities to be measured this is an inherent random variations in the productions of the same element

How to estimate these?

You should repeat the experiment many times; measure the quantity each time (x). A measure of the quantity is the average

$$\bar{x} = \frac{1}{N} \sum_{i=1}^{N} x_i$$

The sample variance is

$$\sigma_x = \sqrt{\frac{1}{N-1} \sum_{i=1}^{N} (x_i - \bar{x})^2}$$

For random normal, 68% of the measured values will be within the range $\bar{x} \pm \sigma_x$

The standard error of the mean is

$$s_{\bar{x}} = \frac{\sigma}{\sqrt{N}}$$

Safety rules and Operating Procedures

It is very important to follow the safety rules. These rules are to protect you personally and to avoid injuries (or may be even death). These rules are also important to protect the equipment in the lab; some of this equipment is very expensive.

General safety rules

- No eating, drinking, or smoking in the lab
- Perform the experiment you are supposed to, do not perform **unauthorized** experiments.
- Read the handout before you start.
- Follow the instructions by the TA or the lab engineer.
- Be always neat.

Electrical safety Rules

You are working in an electrical/Computer engineering lab. All the equipment in the lab is electrical. Electricity can harm or kill you. You have to be very careful in order to not hurt yourself.

Most people think that electric shocks can kill, and that depends on the severity of shocks (voltage). The higher the voltage, the more harm the shock can cause. However, that is not completely true. Current not voltage that cause harm. Surprisingly, very small amount of current can cause death. For example you can get 1000 Volts shock at home in winter when you touch a door knob or a faucet. While, 1 Ampere can easily (very easily) kill you.

The effect of current is shown below

Current	Effect
1-5mA	Sensation
5-20mA	Muscle contraction (you might not be able to let go)
20-100mA	Pain, difficulty in breathing
100-300mA	Ventricular fibrillation
>300mA	Possible death

Having said that, the actual harm depends on many factors. The point of entry and exit of the current plays a very important role. Current passing through the heart causes a lot more harm that current flowing between two fingers in the same hand. For example, if you touch a live wire with your hand, and the current will flow to ground through your shoes, it will most probably passes through your heart. Touching live wire and ground with 2 fingers in the same hand will probably not flow through the heart.

Another factor is the skin being wet or dry. The skin resistance drops dramatically if it is wet (less resistance means more current for the same voltage source).

Some safety tips for you

- Always power down all electrical equipment while connecting the circuit. After the circuit is connected, check it one more time and then you can power the equipment. If any doubt, check with the TA.
- Do not touch any wire without powering down the equipment.
- If you want to make changes to the circuit, power it down first, and then do any changes.
- Keep your hands dry (more resistance means less current).
- Avoid extension cords, if you have to, check with the TA or the lab engineer before using them.
- If you are going to leave your bench, even for a short time, power the circuit down.
- Report any faulty equipment (or you think they are faulty) to the TA or the lab engineer.
- If you smell any burning plastic smell, disconnect the circuit right away and report it to the TA or lab engineer.
- Voltage above 50V (RMS) AC, and a little higher than that for DC is considered dangerous, use extra care in such a situation.
- If someone suffers a serious electric shock, he/she may be knocked unconscious. If he/she is still in contact with a live wire, immediately turn off the power before touching that person.
- Call for emergency medical assistance, you may administer first-aid if you are familiar with that.
- Do not use water to extinguish electrical fire.
- If you are using soldering iron, it can get very hot, be careful.
- Use wrist strap if you will be touching chips, especially CMOS chips, it could be damaged very easily.

Trouble Shooting Hints

1. Be sure that the power is ON, as simple as this is it is a major reason of complaining "the circuit is not working"
2. All the ground connections are connected properly
3. Be sure that the measuring equipment is connected properly.
4. You should have a schematic diagram of the circuit before staring implementation. Go through the circuit and the schematic to be sure the circuit actually implements the schematic diagram.
5. All supply voltages are connected and are working properly. Sometimes you have to measure the supply voltage to be sure it is identical to what is shown on the display.
6. If everything up to this point is O.K. (very unlikely), you might have a defective component (breadboard, resistor …) or a device. In this case, you have to trace your circuits measuring voltages at different nodes and comparing them with what you expect.
7. The TA is always there to help you.

Report Format

You are required to submit a prelab and a lab report for every lab session (first lab has no lab report). All lab reports and prelabs should be neatly typed.

Prelab

Prelab is submitted at the beginning of the lab. You should prepare it before you arrive to the lab. No late prelabs will be accepted. If you do not have it before the lab, you get a zero for the prelab.

The first page of the prelab/lab report is as follows

York University

Dept. of Computer Science and Engineering

EECS 2200

Electric Circuits

Lab (Prelab) Report

Lab x

Lab name

Submitted by   :   name 1

                         name 2

Date

The work in this report is our own. We have read and understood York University academic dishonesty policy and we did not violate the senate dishonesty policy in writing this report.

Signature

_____   _____

For the prelab, just you have to answer the questions in the lab document.

For the lab report; it should consist of the following sections

Abstract: A brief statement (few sentences) mentioning what you did in the lab and your results.

Analysis: If you were asked to perform any theoretical calculations, that should be done in the Analysis section.

Experiment setup and results: That is the main part of your lab report. If you constructed any circuits, a schematic diagram should be drawn to represent the circuit. The components measurements and other measurements should be stated here. If you are required to draw any graphs or plots, that also should be stated here. Any tables should be here too.

Simulation results: If you were asked to perform any simulation, the simulation results should me mentioned in this section. Code is also presented here if you were asked to.

Discussion: If you were asked to compare experimental work to theoretical work, or to explain parts of your experiment, it should be here

Conclusion: State what parts of the lab objectives you achieved, any difficulties you met, you can even mention some suggestions to the labs in order to increase the students experience in your opinion.

SPICE Tutorial

Here we will present tutorial on using PSPICE and Orcad Capture.

The tutorial will cover few examples in increasing degree of difficulty. The examples will cover many details of SPICE.

Example I Q-Point Analysis

In this example we will construct a simple resistive circuit and measure the currents in, and voltages across, the different resistors.

Start All Programs → cadence →OrCAD 16.6 Lite → OrCAD capture CIS Lite

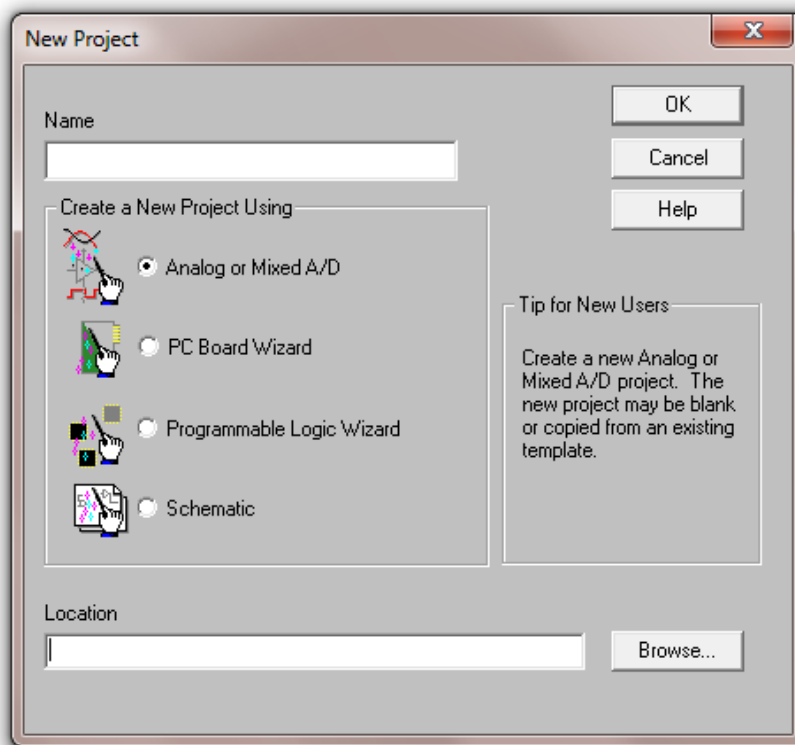You will be prompted with an error message saying License was not found.

Would you like to launch the lite version

Click "Yes". You can check don't ask me again if you do not want to be bothered again.

OrCAD capture CIS Lite windows appears

Click on the project New in the getting started window, or File → New → Project.

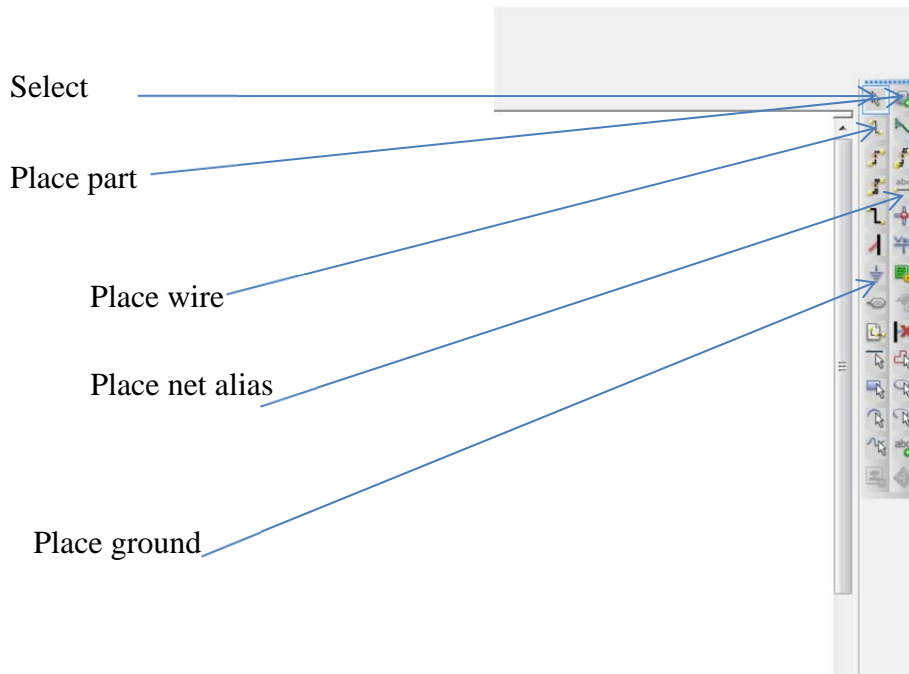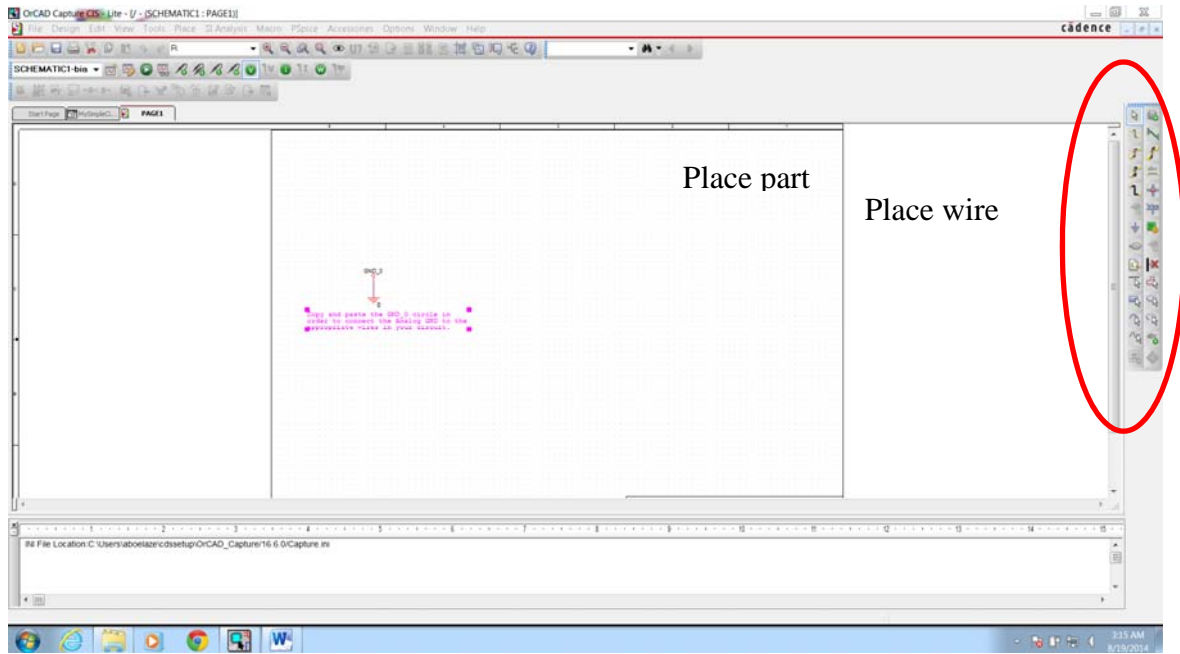The new project windows appear

Choose Analog or Mixed A/D

Browse to the appropriate directory location, and name the project MySimpleCircuit
Then OK.

Create PSpice Project Window appears, choose "Create a blank project", then OK. The
following screen appears

Expand mysimplecircuit.dsn then expand schematic and double click on page 1

You will get the schematic page, with a ground node in it and text to tell you to copy and paste the ground in your design.

Every circuit must contain a ground and the name of the node is "0" that is the number zero.

On the right there is a group of fast access keys, the most important is the place part (could also be accessed from the menu bar Place → part. The place ground button (ground is also a part you can access it from the place part button or menu. The place wire (also from the menu Place → Wire). Place net alias (also from the place menu, and the place ground as shown in the expanded figure

That schematic page is the window where you will build your circuit. The parts will be placed in the schematic one part at a time.

Go to Place → Part, the following window appears (or click on the place part in the side fast access buttons).
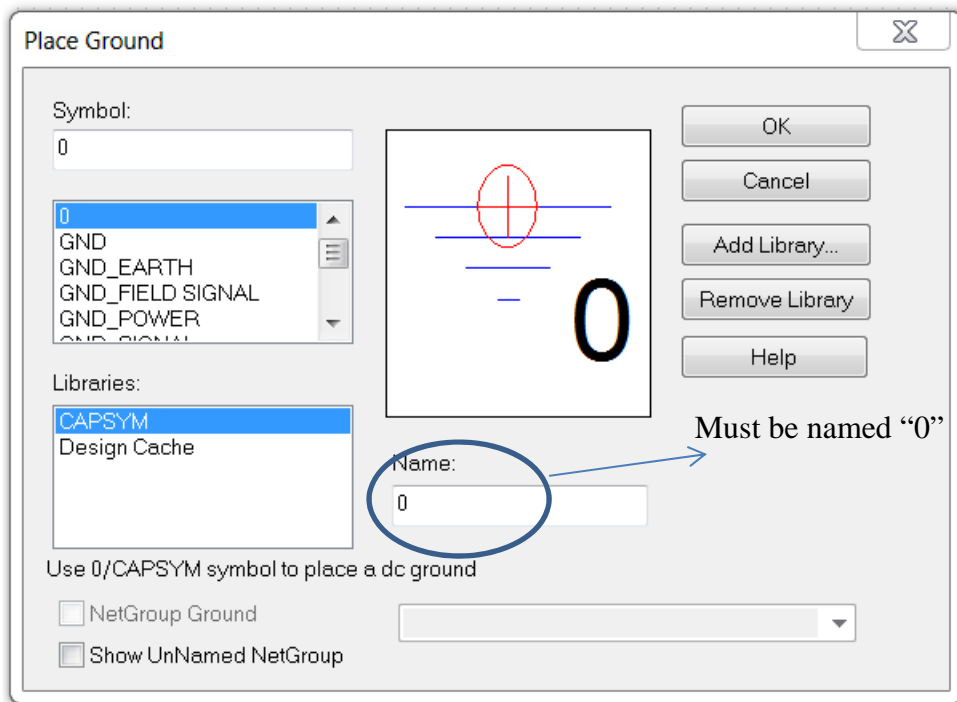
Probably it will not include all the libraries shown in the Figure.

The Figure here shows CAPSYM which include the node 0 (ground). You can also choose the node GND, but in this case you must right 0 in the field name.

Click on Add Library a new window appears, choose the PSpice directory.

Some libraries will appear, we need eval and source. A good idea is to add them all though, at least eval and source must be added.

The part windows may appear as a floating window or ducked between the schematic and the tool bar on the right.

Now we start constructing a simple circuit. Click on Place Part, the following window appears (again, the part window may be floating or ducked).

Choose the analog library

Type R in the part field,

The part list appear as shown, choose R and the resistor picture appear in the box below the library as shown in the window below.

Hit Enter and move the mouse to the schematic part, click to insert a resistor (you can click as many times as you want, when you finish his the Esc key).
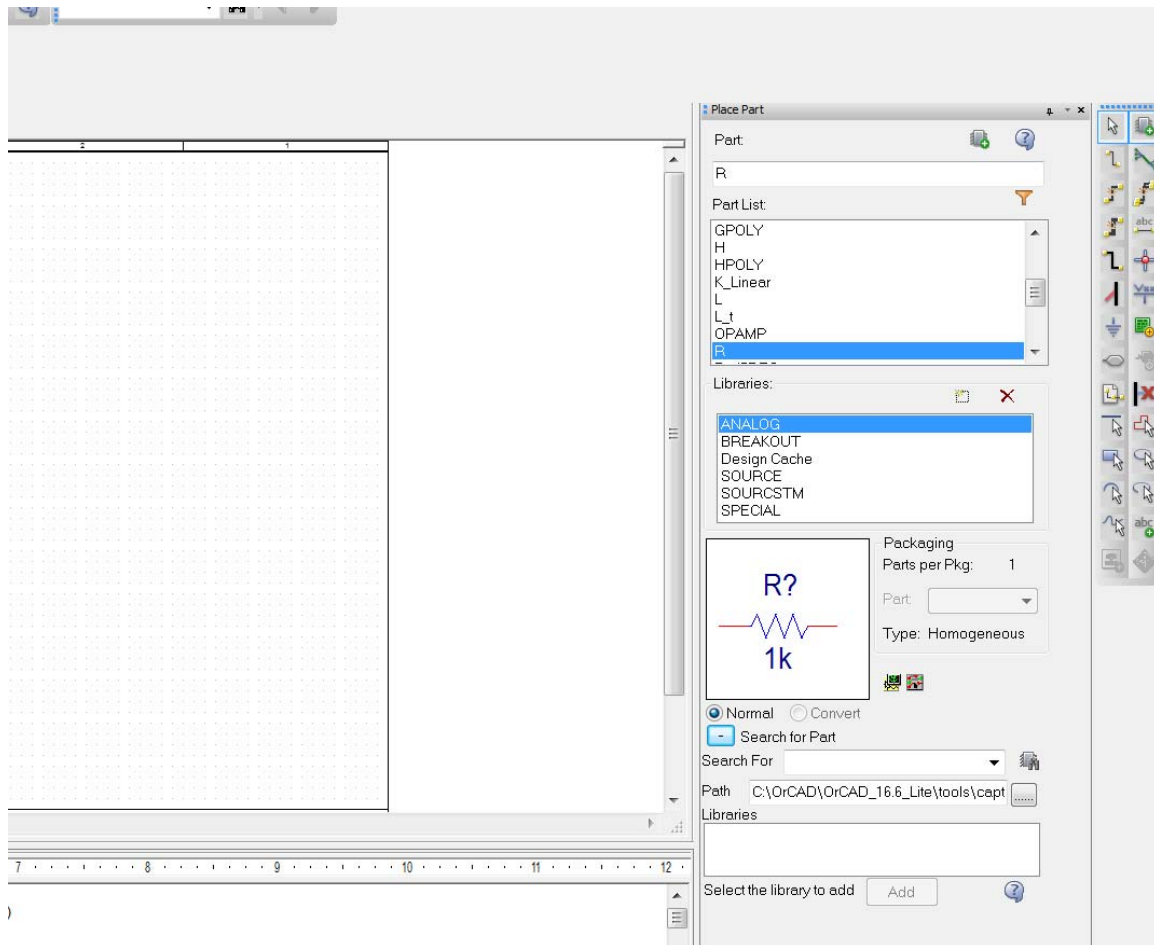
Put 3 resistors as shown below, the resistors are automatically named R1, R2, … with a value of 1K.

DoubleClick on the 1K value of R1, a window will appear with the value set to 1K, you can change this to any value you want. Change it to 5K. In the font part, click change and change it to size 10 (size 5 seems too small). Repeat the font change with the R1 name.

You can set the font in your design this way

Go to MySimpleCircuit tab.

Right click on mysimplecircuit.dsn file

Choose Design properties

A window appear

Choose Schematic Desig tab

Click on design properties

A window appears where you can choose the fonts type and size for all new parts (font Ariel 10 seems to work fine).

Highlight the second resistor, click R (rotate) each click will rotate it by 90 degrees counter clockwise (this is important, since when you place a resistor, nodes 1 and 2 are automatic to the left and right of the resistor respectively, the current is always measured as going from 1 to 2, if the current is actually from 2 to 1 the current will be negative.

R as placed   After one rotation "R"   After two rotations   After three rotations
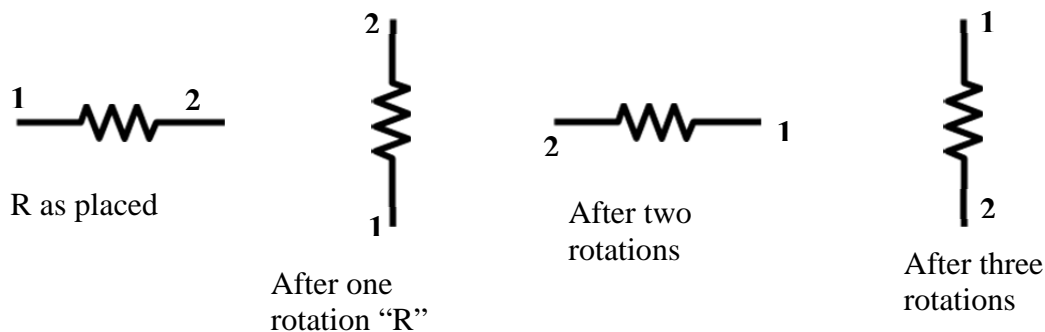
Highlight R3 and hit the R key twice to rotate it 180 degrees.

In the part window, choose the "source" library.

In the part type Vdc a batter symbol appears, highlight it and hit enter and place a battery (source) in the schematic. Double Click on the value (0Vdc) and change it to 10V.

Click on place wire (or Place → Wire)

Connect the parts as shown in the Figure below. Connect also the ground as shown below
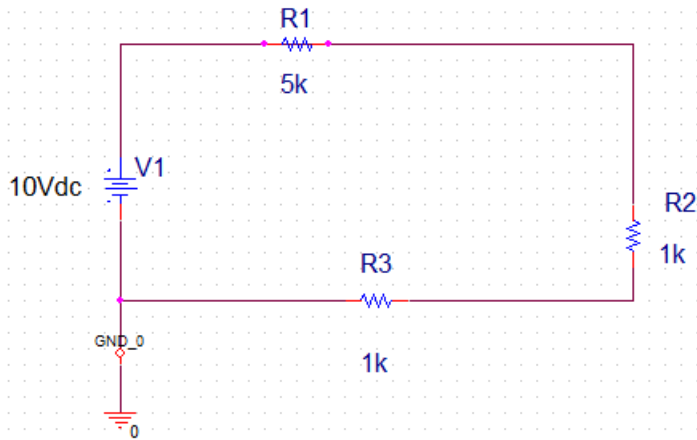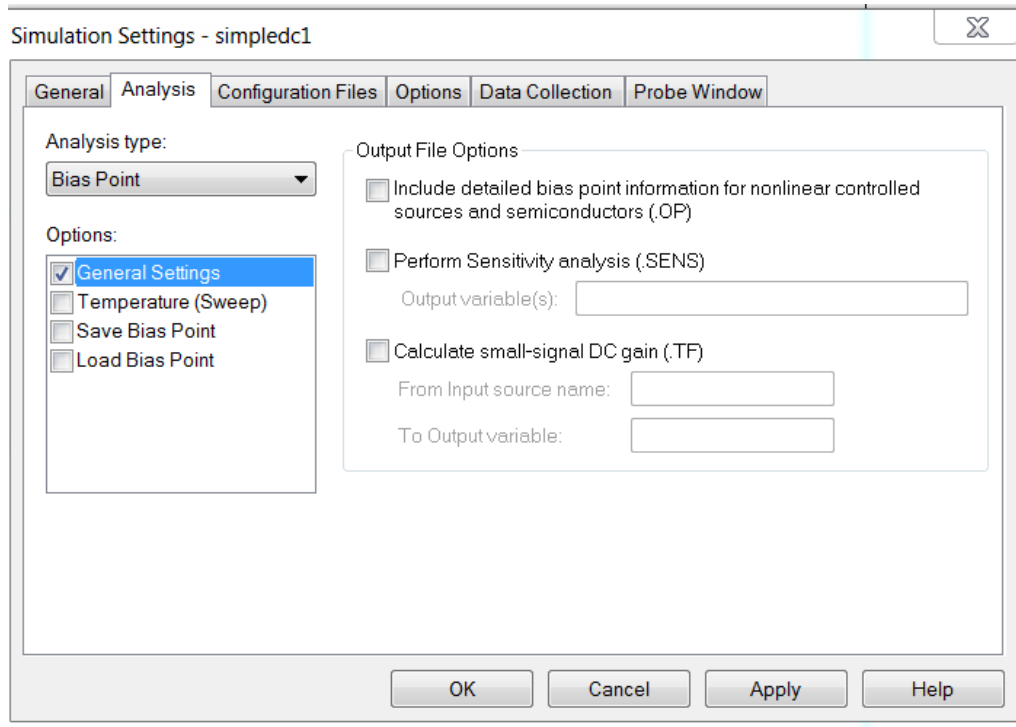
Now, we are ready to simulate the circuit

Go to the top menu, Pspice → New simulation profile.

Choose a suitable name, and inherit from "none"

The following window appears

In the Analysis choose "Bias Poin", then O.K.

Then click on the green arrow, or choose PSpice → Run

The simulation runs.

The simulation window appears, but since we are not using this window (we are only interested in the bias point analysis), we will not be using it.

In the schematic window, right below the main menu bar you find 3 buttons each one is a green circle with "V", "I", or "W" inside. These represent the voltage, current, and power respectively. If you click on any one of them, the corresponding value (voltage at every node, current in every circuit element, and power consumed in every circuit element) appears in the circuit schematic.
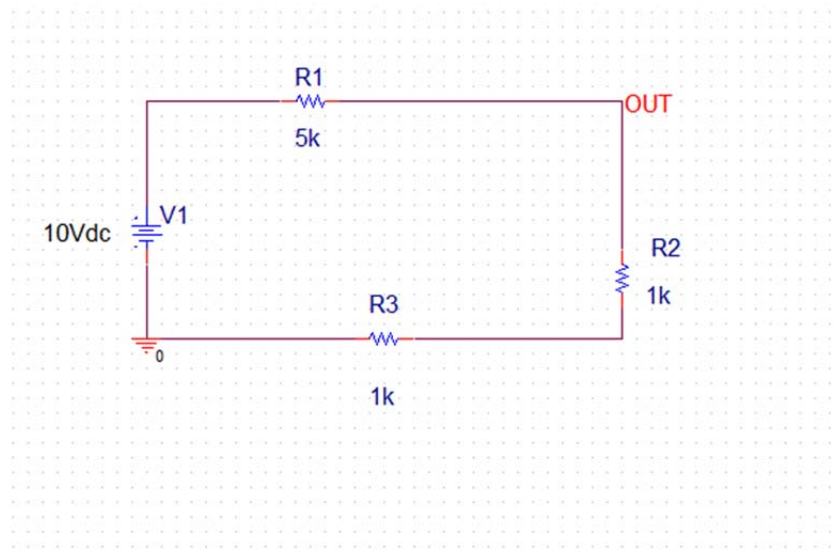
DC Sweep

Now, we will perform a DC sweep. We change one (or more) of the DC sources and check the voltage across, or current, in an element.

Go back to the schematic diagram. From the right hand fast access keys, choose net alias (a horizontal line with abc written above it), or Place → Net Alice. A windows pops up. In the filed "alias" write OUT. Then OK.

A small square appears attached to the mouse cursor. Move it to the intersection of the horizontal line connecting R1 to the vertical line connecting R1, then click.

The label (net alias) "OUT" is shown there as shown below. It is just a handy way of describing this node.

From the top menu. Choose PSpice → edit simulation profile, you get the following window
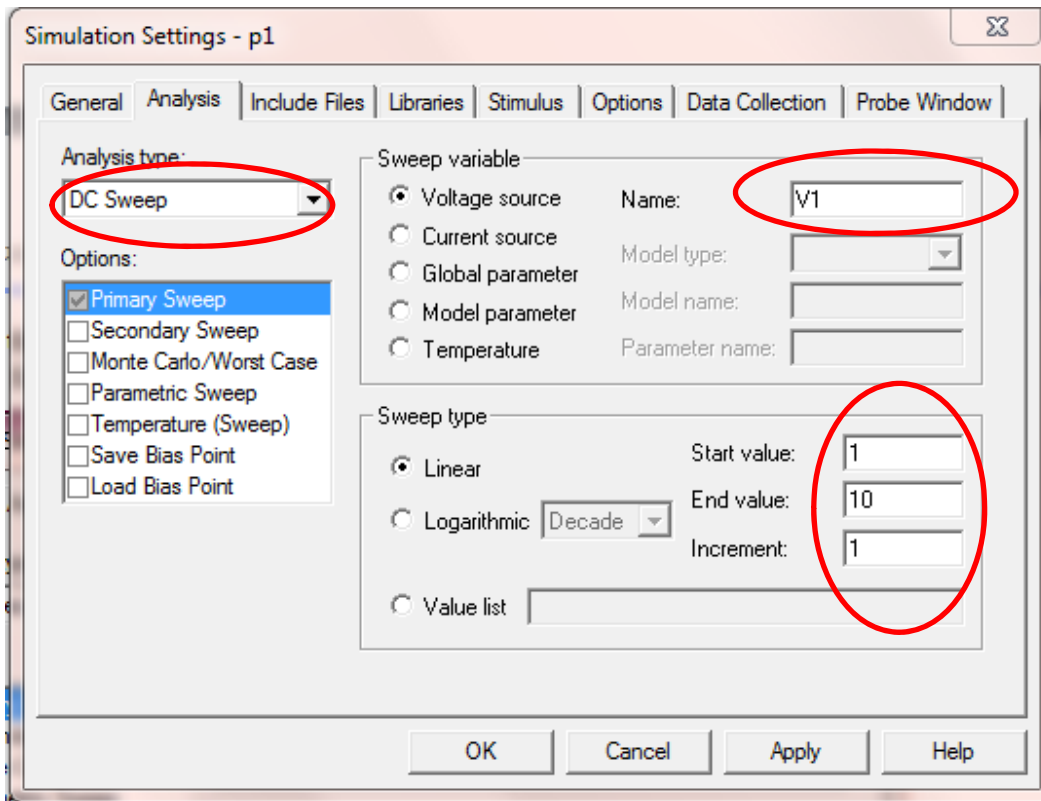
In the Analysis tab set the "Analysis type" to be DC Sweep. Choose primary sweep from "Options".

In Sweep variable, choose Voltage source

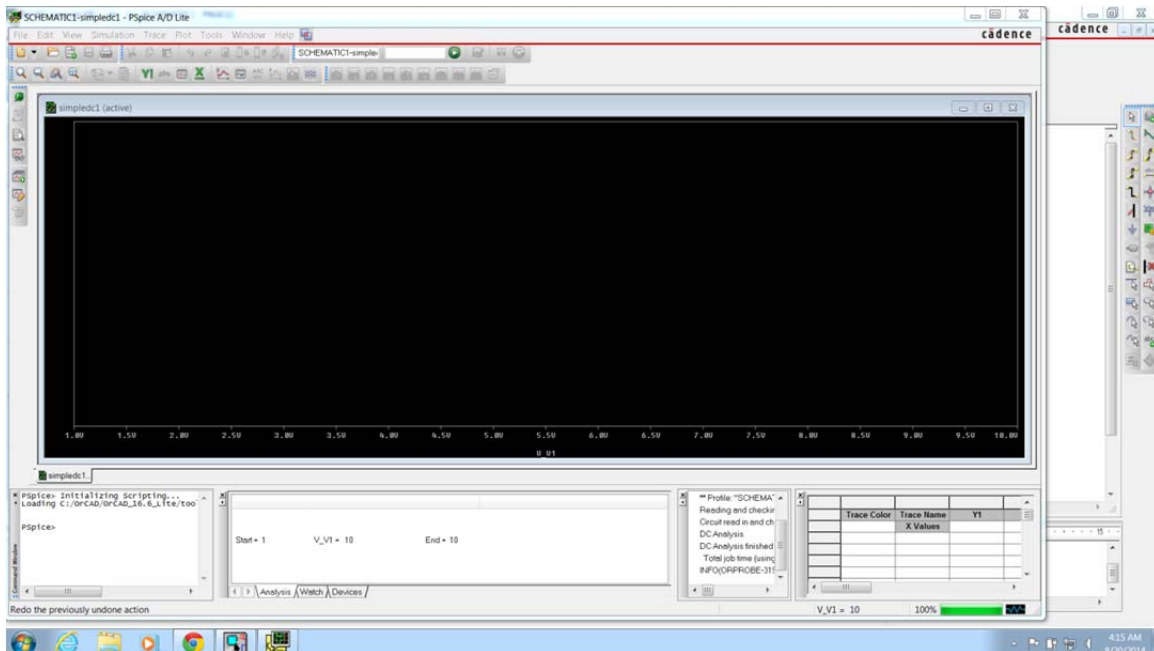Set name to V1 (The name of the voltage source in your schematic).

In Sweep type, choose linear, start value = 1, End value = 10, increment = 1.

PSpice will change the voltage source V1 from 1 to 10 in linear increment of 1.

Now click Apply and O.K..

In Schematic choose PSpice → run

A window appear

That window indicates that the simulation is completed; a linear sweep was done on the voltage source V1 from 1 to 10.

Note that the upper window contains the X-axis with V1 varying from 1 to 10 as we set it. Nothing appears in the window, we have to choose what we want to see.
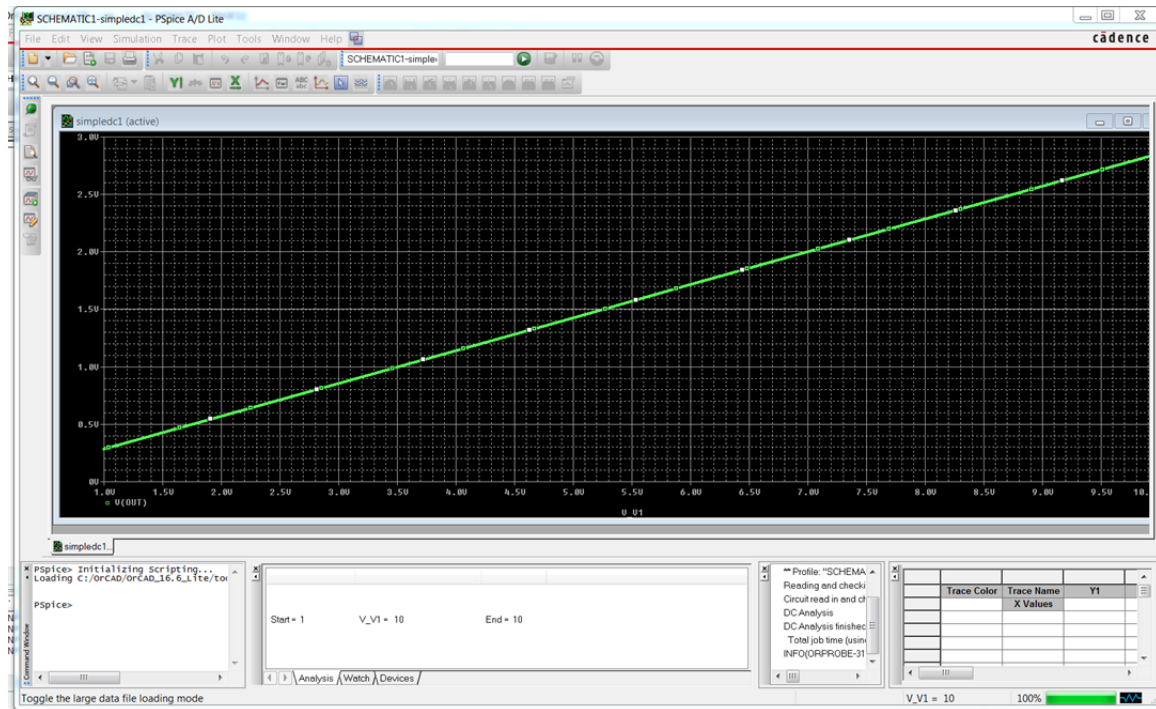
Go to Trace → add trace, a window appears

The right panel include all signals that you can trace. The left panel includes macros and operations.

From the right panel, choose V(OUT), note that is the net label we choose for our output voltage V(OUT) appears in the trace expression field, click O.K.

Now the diagram showing the relation between Vout and V1 appears.

In the graph below I made the line thicker to make is easier to see. You can do that by placing the mouse on the line, right click, and choose "Trace property" where you can change the line type, thickness and color.

Again, go to top menu Trace → add trace

The previous window appears.

Click on "-" from functions or Macros(or type - , minus).

Click on I(V1) from simulation output variables
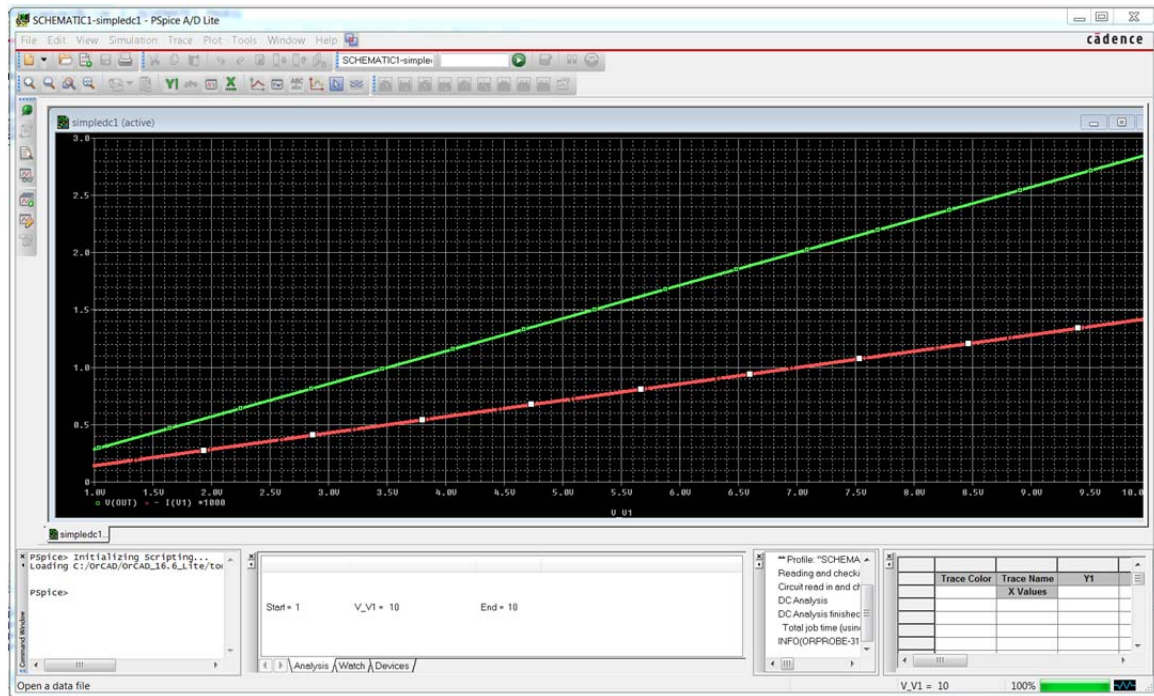
Click on "*" from functions or macros

type 1000 in the trace expression box.

Now the trace expression box contains –I(V1)*1000

Click O.K.

The following graph appears

The plot shows two graphs, the green one is V(OUT) and the red one is I(V1).

Now, why did we do –I(V1)*1000?

I(V1) is the current flowing in the v1 source, by definition the current in a voltage source is flowing from the positive terminal to the negative terminal. That is a negative quantity (since the actual current is leaving the positive terminal of V1).

That should be O.K., however the value of the current will be negative, and the graph will be resized to show both positive and negative values. We prefer to see the current in the positive direction as we imagine the current will be flowing in the circuit.

Also, the value of the current will be in the milli ampere range. If we leave it as is it will be a line almost on the X-axis. If we multiply it by 1000 the value will be in the ampere range and will show as a decent size graph as we see in the Figure above.

That concludes our first circuit.

Transient analysis

Now start a new project called RC1 to simulate the step response of a simple RC Cicuit. Draw the following circuit. Do not forget to change the name of the ground from GND to 0 or SPICE will not work. Save the file.
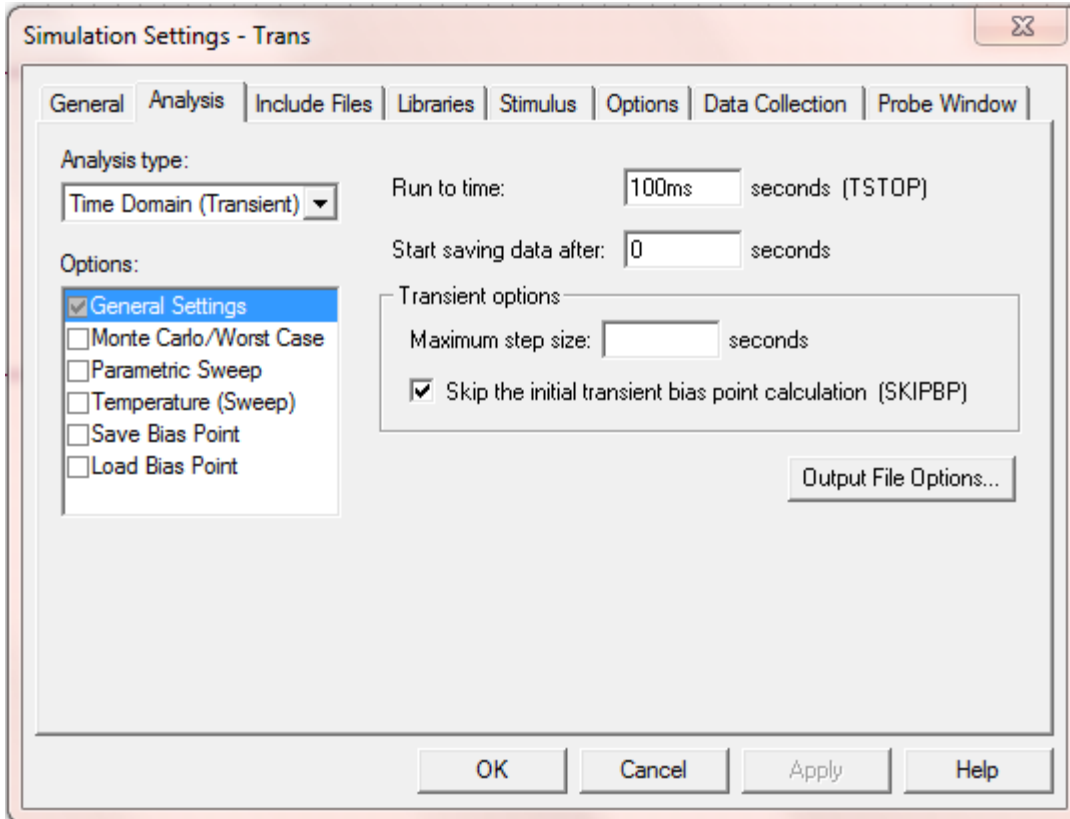
Change the R value to 2K and the capacitance to "10uF"
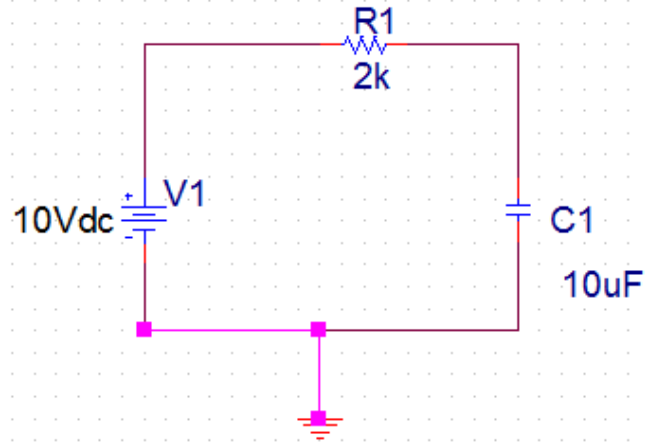
Now, we have to set the simulation profile.

Top menu PSpice → new simulation profile.

Call it tran, no inheritance from other profiles, then O.K.

In the Analysis tab choose analysis type to be Time domain (transient), general setting

Choose the run to time 100 msec. (the time constant in the circuit is 20 ms, we will run it five times that). Also start saving data after set it to 0.

Click on "Skip the initial transient bias point calculation". This is very important if you want to see the transient response. Without it SPICE will calculate the bias point (Capacitor voltage is 10V) and use it to start simulation. In this case you will not see the transient period and the voltage will be a constant 10V. Click Apply and OK.

Go to the top menu PSpice → run

The output window appears as before without any traces.

Go to Trace → add trace

The add the following traces

V[C1:2] Note this is the upper terminal of the capacitor, if you choose V[C1:1} that is connected to ground and always 0.

Choose also trace –I[V1]*1000 as before

To make the graph lines thicker, right click on the graph curve, choose property and set the thickness as you wish. Here is the output
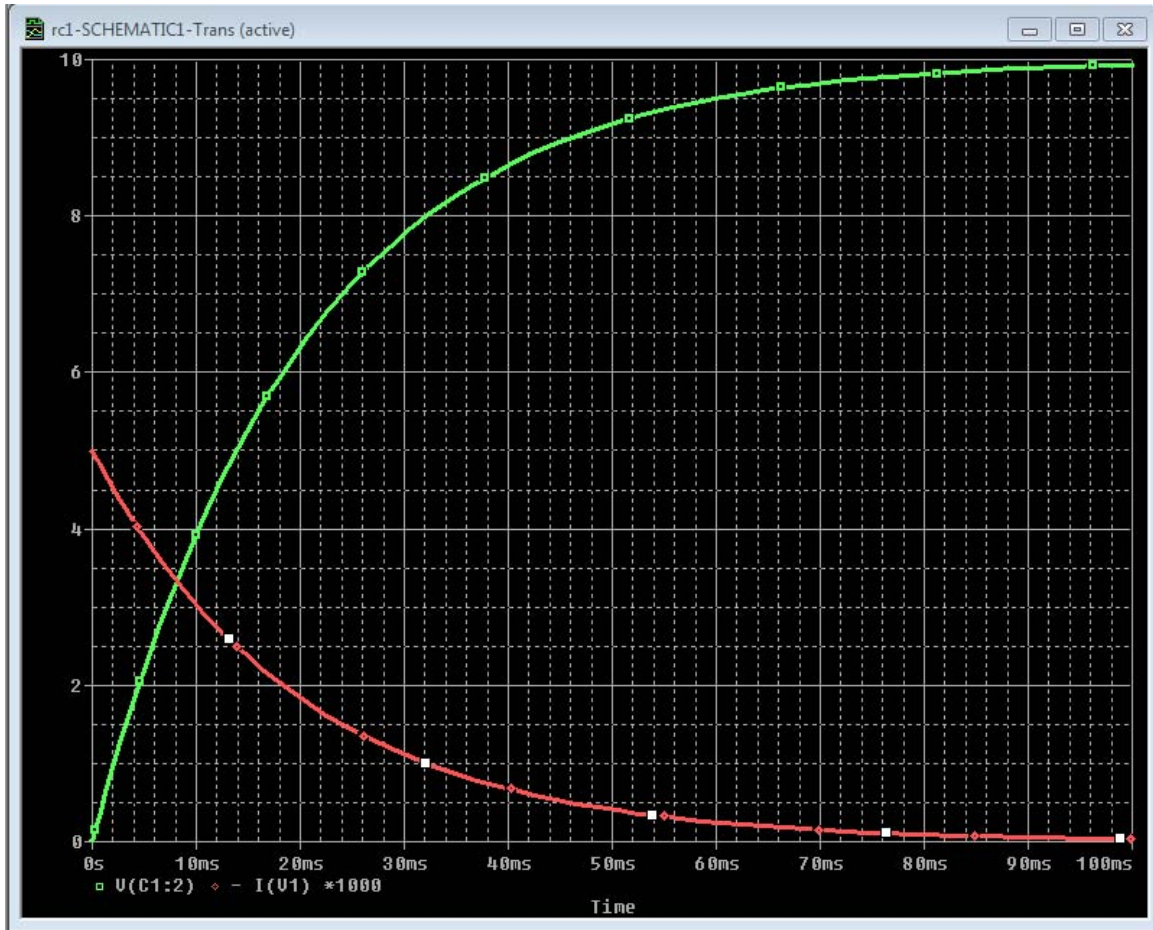
Now for a little bit more complicated input source. Instead of a DC voltage source, use PWL (piece wise linear) source. A piece wise linear source is defined by the voltage values at different times. For example consider the following source.
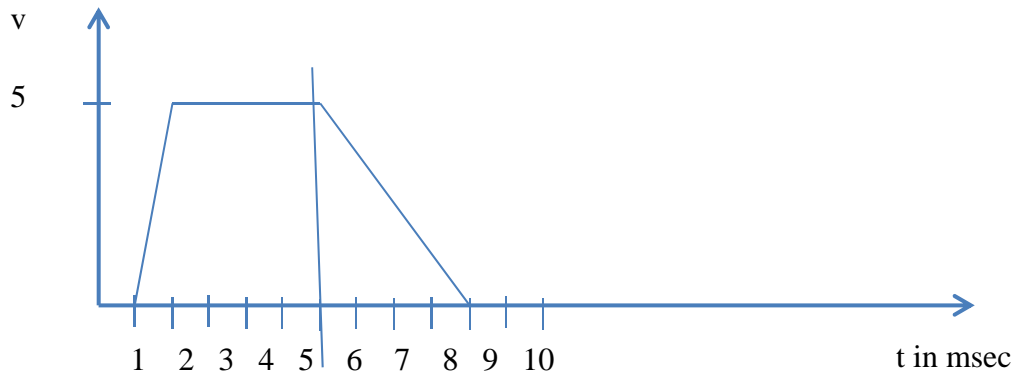
The above waveform is piece linear waveform. It stays at 0 V till 1 msec. Then it, goes linearly from 0 to 5 V for another millisecond. Stays at 5 V for 4 msec. then it goes linearly to 0 in 4 msec.

Replace the DC source in your circuit with a PWL source.
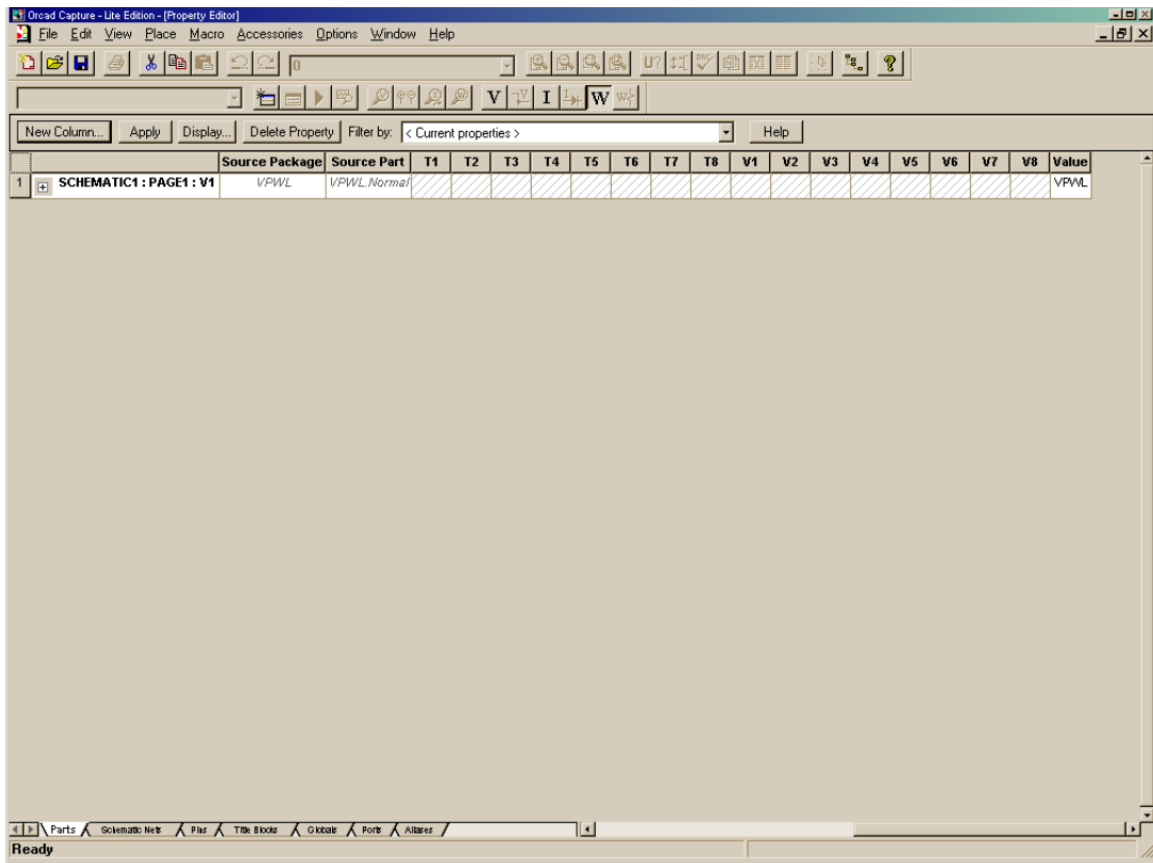
Highlight V1 in the schematic and delete it

Insert part, from library source choose VPWL

Insert it in the schematic

Connect it to the circuit

Now double click on the source to open the property editor

In the Parts tab (bottom part of the window) start inserting columns. Insert columns with

Name	T1	T2	T3	T4	T5	T6	T7	T8	T9	T10
Value	0e-3	10e-3	100e-3	101e-3	200e-3	210e-3	220e-3	230e-3	240e-3	250e-3
Name	V1	V2	V3	V4	V5	V6	V7	V8	V9	V10
Value	0	1	1	0	0	0	0	0	0	0

This creates a piecewise voltage signal with value of 1V from 10ms-100ms and zero elsewhere.

Apply and exit.

Now, the circuit looks like this
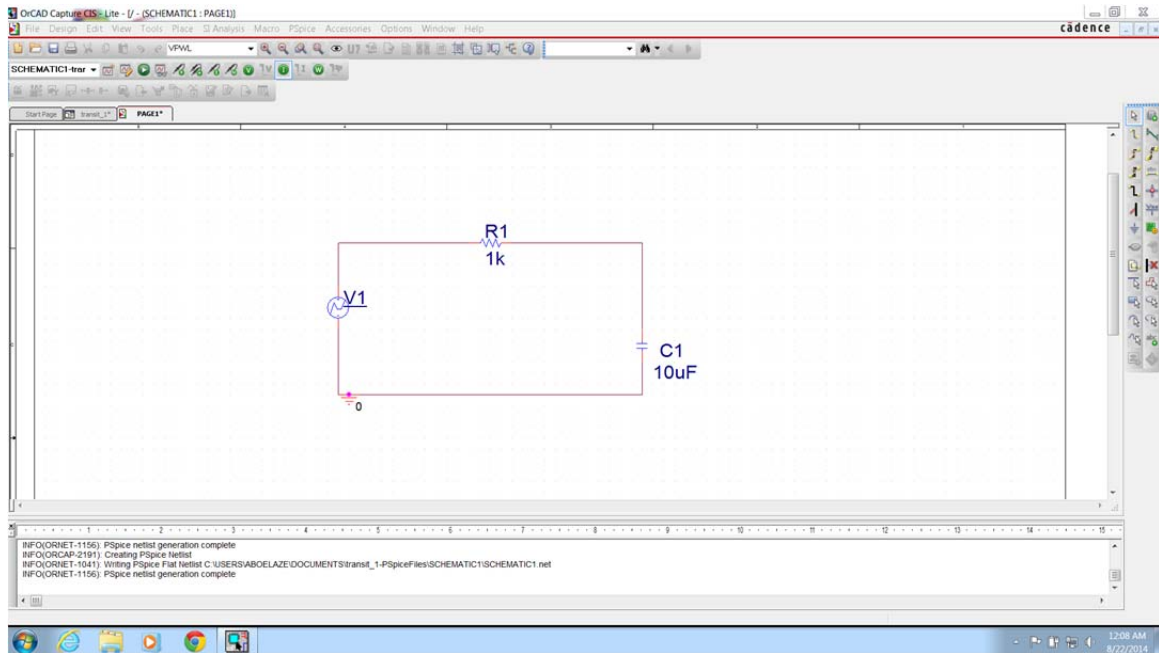
Edit the simulation profile to make the simulator runs for 300 msec. Run the simulator.

Using Trace → add trace add V[C1:2] and V[V1:+]. Note that V6 is the name of the PWL source in my simulation. The result should look like that
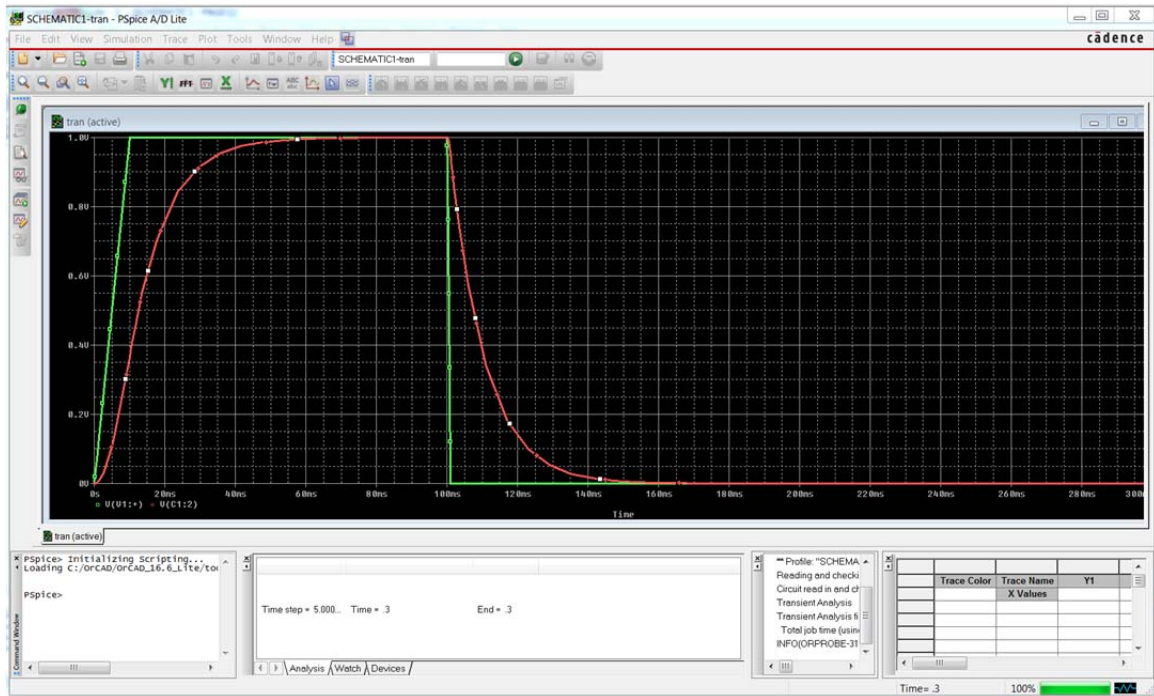
Exercise

Simulate the following circuit
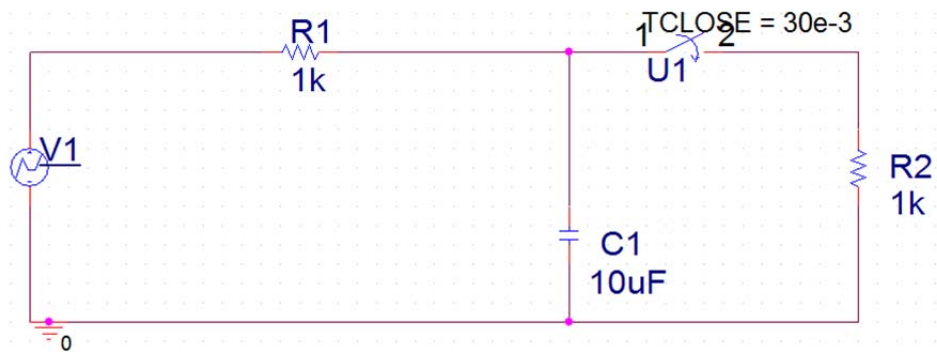
The answer should look like that, not ethat the switch is sw_tClose in the library EVAL and set it to close at 30msec.

L=20 mH

R1 = 5K

C=20 uF

R2=10K

V1=10 V
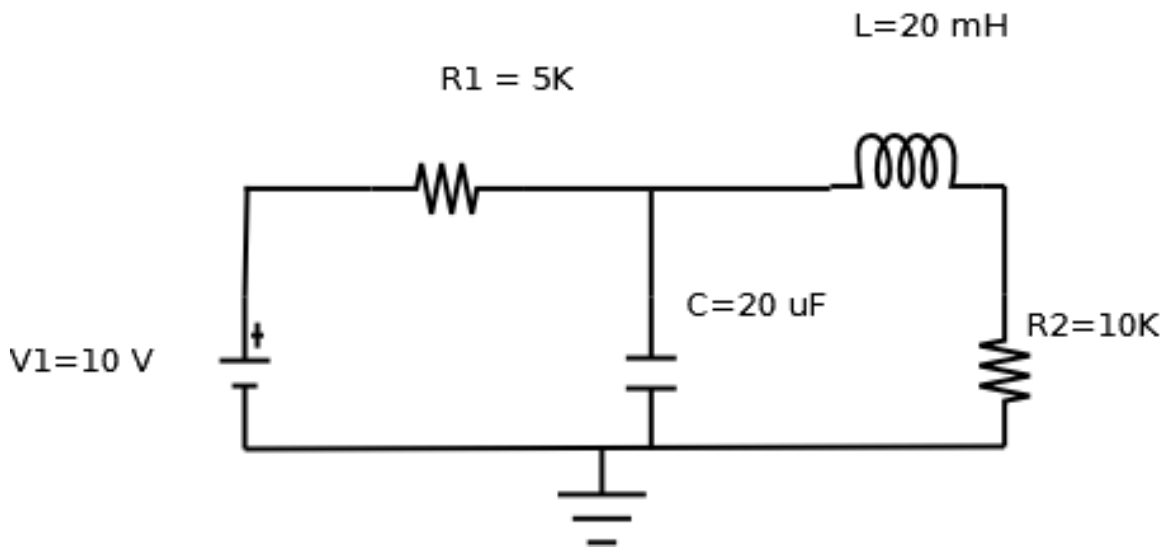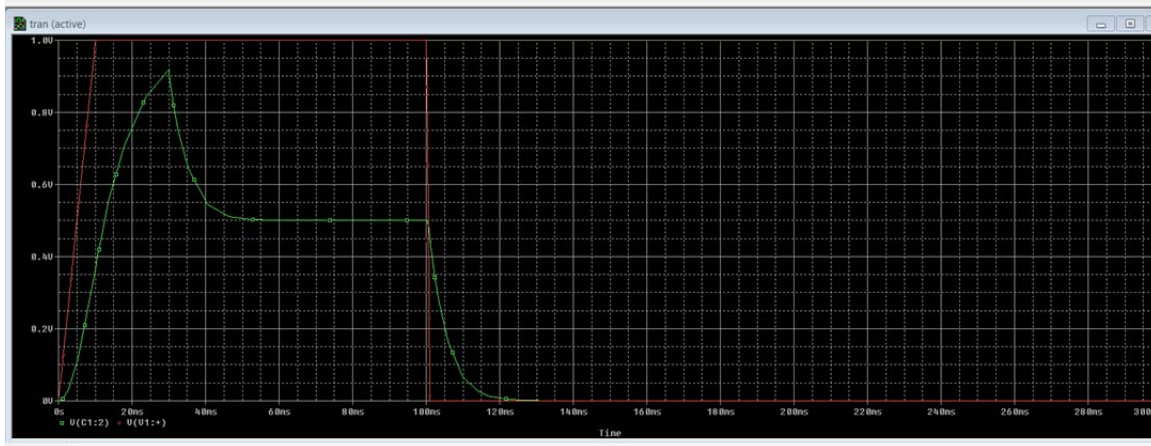
INSERT p10
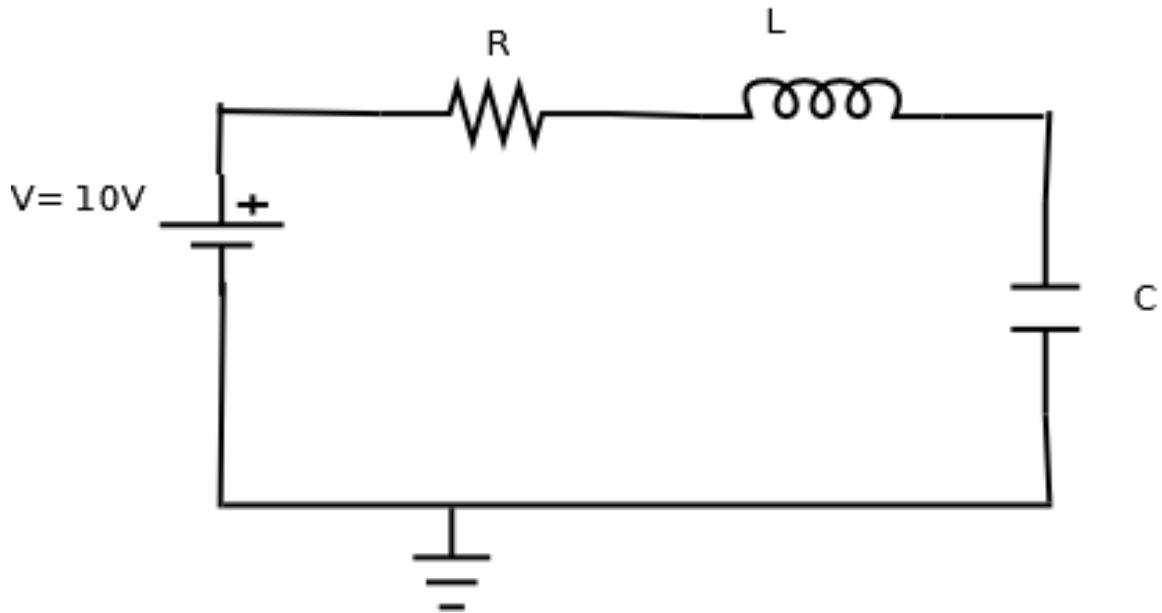
Simulate three cases as follows

1.   R=10 Ω C= 10 µF L = 0.25 mH
2.   R=2 Ω same C and L
3.   R = 25 Ω same L and C

Submit the following

The schematic of the circuit (one schematic is for the three cases)

The SPICE code (one code is enough)

Graph showing the voltage across the capacitor and the coil **for every case**.

AC Analysis

In this section, we xxxxxx

Start a new project and construct the following circuit
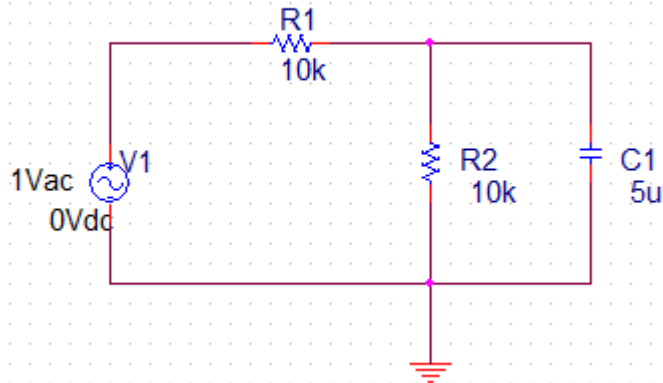
Choose the parameters as shown.

in the Vac source, double click on 1Vac and choose a value of 1 (there is no change here, just to know how to set it).

Save the file,

In capture, top menu PSice → New simulation profile, choose a suitable name, no inheritance.

Set the parameters as shown. Click OK
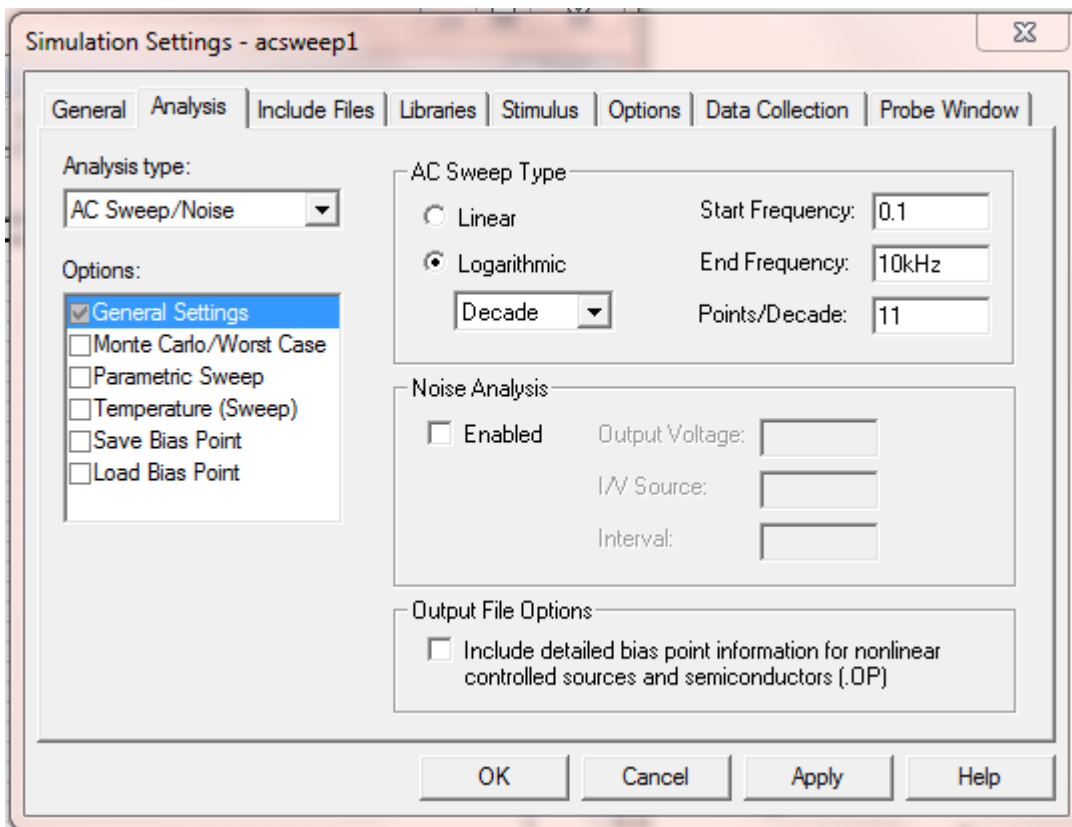
PSpice → Run, the probe widow appears.

Now, top menu Trace → add trace add V(C1:2)

add P!11
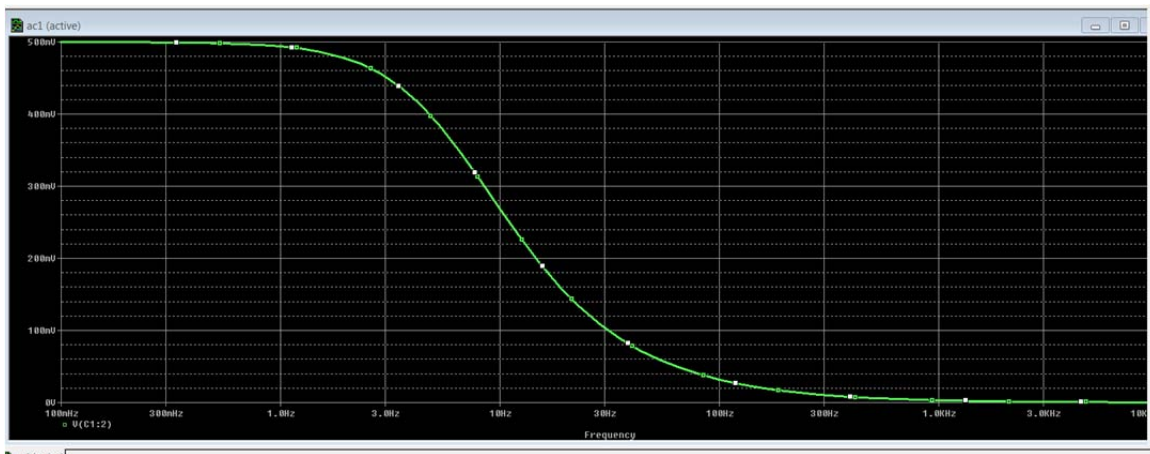
Now we can see Vout vs frequency.

For Bode plot, we would like to plot Vout in dB. In that case Trace → Delete all traces.

Then add a new trace. From the output variable choose Vdb(C1:2) (choose V(out) then insert db after V). Another way to do this is to choose DB() from Functions or macros from the right hand side of the Add Trace window, then click on V(C1:2) to get DB(V(C1:2)).

Add another trace Vp(C1:2) (again insert p after V in V(C1:2). This is to show the phase of V_{C1}. Or you may use the Functions/macros P(V(C1:2)).

You can also divide the display into 2 plots. From the top menu choose Plot → add plot to window. Another plot will be inserted. The active plot will have "SEL >>" to its left. If you click in the inactive plot, then it becomes active and "SEL >>" moves next to it.
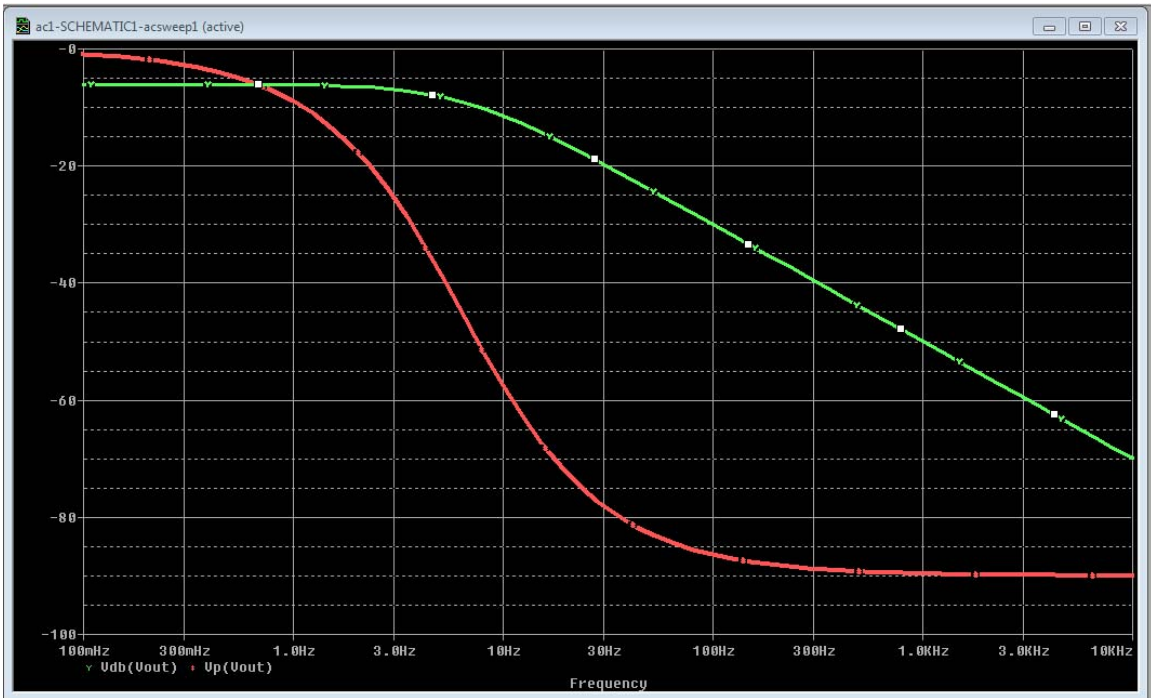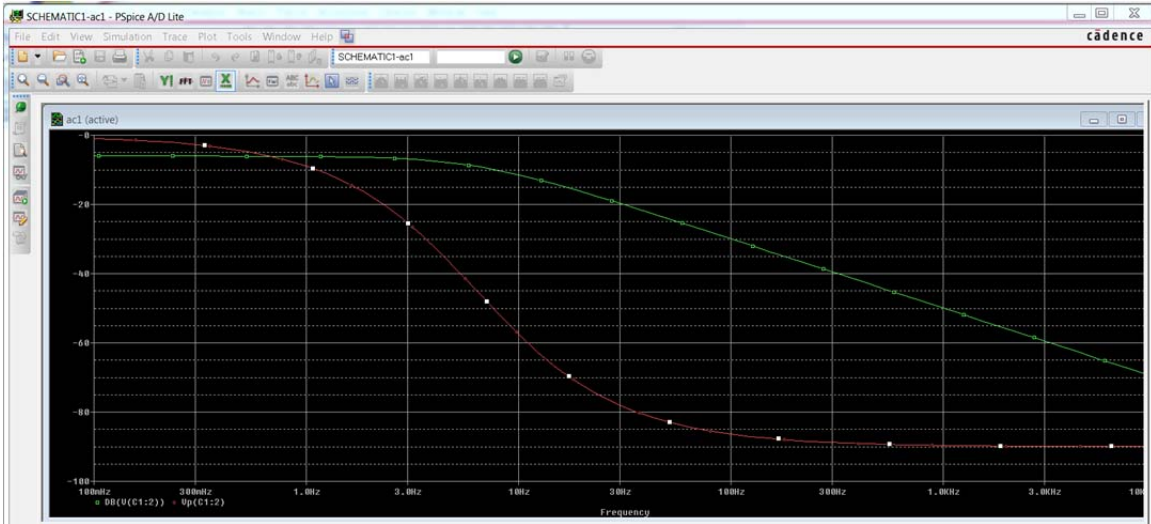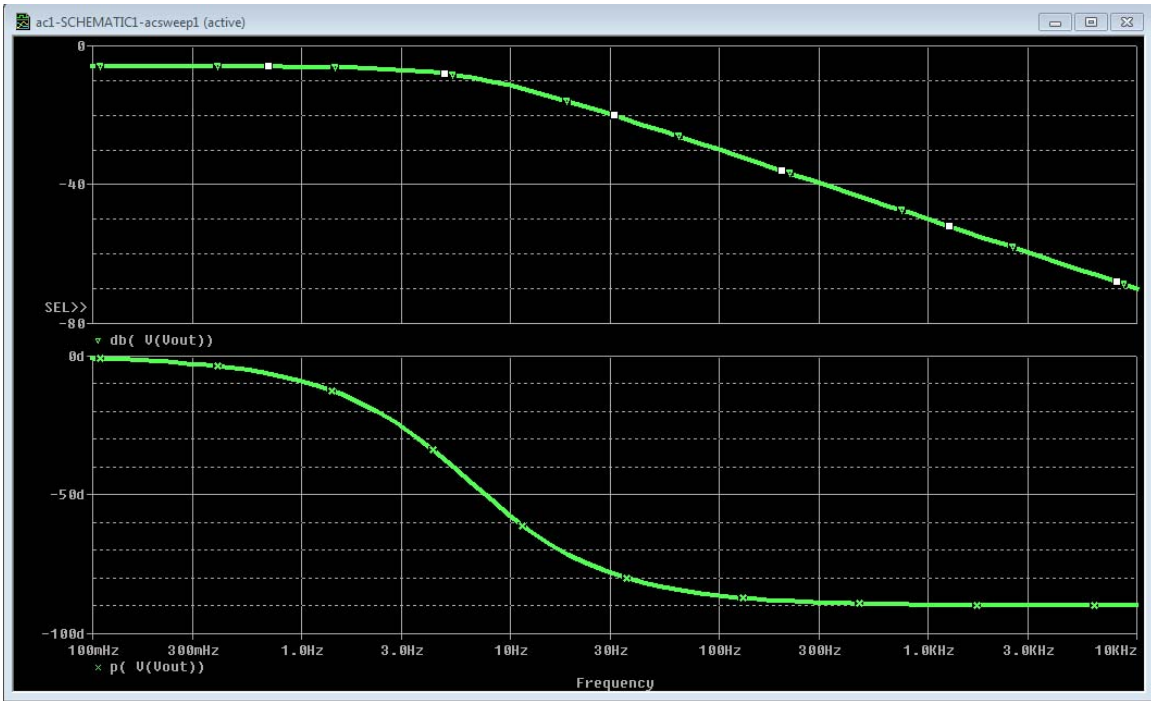
LAB 0: READ BEFORE YOU START

Please read carefully what you have to do in every lab

- Prepare the perlab and submit it in the first 15 min of the lab.
- Experiments are performed in groups of 2.
- The lab part describes the circuit you have to implement. The nominal values for every component will be given. However, in reality a 10 KΩ resistor is not actually a 10 KΩ resistor, it is 10 KΩ ±tolerance. The tolerance usually is 5% or 10%. So, for every circuit you implement you have to measure the actual values.
- Record the values that you measured, and everything that you measured, in your lab notebook. The TA must sign every page of the notebook BEFORE you leave the lab.
- Submit the lab report for the previous week at the beginning of every lab.

LAB 1: Basic DC Measurements

Part I: Introduction to the instruments

* DO NOT LET THE LEADS FROM THE FUNCTIONAL GENERATOR OR THE POWER SUPPLY TOUCH EACH OTHER, IF YOU DO THE INTERNAL FUSE WILL BLOW (OR THE FUNCTIONAL GENERATOR WILL BE GONE).

Objective:

- The objective of this lab is to introduce you to the equipment you will be using in this course; they are (a) The power supply, (b) the multimeter, (c) the function generator, and (d) the oscilloscope.
- Get familiar with how to read a schematic and to wire a circuit on the breadboard

This was never intended to be a complete description and a user guide for the equipment; this is only a quick introduction to get you started. In the different labs, you may be introduced to other features of the devices as needed. On the computers in the lab, you can find the user guides for all the devices, together with 'quick starts" and tutorials, read these if you want to know more.

Function generator:

On each station in the lab you can find Agilent 33120A functional and arbitrary waveform generator. You will need this to generate square waves, sine waves, and ramps. A front view of the device is shown below

Turn it ON by pushing the power switch

At power-on, the function generator outputs a sine wave with 1 KHz frequency and amplitude of 100 mV.

You can change the frequency or amplitude in 2 different ways.

Push the "Freq" button in the second row to control frequency

Use the knob to decrease or increase frequencies.

Change the frequency to 960 Hz.

Change it to 1.4 KHz.

Return it back to 1.00 KHz.

Push the "Ampl" button (next to 'freq" button)

Again, use the knob to change the amplitude to 120 mV, 80 mV, 100 mV).

Another way to do the same thing is as follows

Push "Freq" to enable frequeny modification as before

Use the > and < buttons to move among the different digits

Use the up and down arrow (^ , ∨) to increase or decrease the chosen digit

Use this method to change the frequency to 4.41 KHz.

The same thing could be done to change the amplitude of the sine wave after pushing the "Ampl" button.

This is a slow way to change the frequency especially if you will be changing it a lot; there is another way to set the frequency you want.

Push "Freq" button

Push 'Enter Number" ENTER NUM will be displayed on the screen

Enter the required digits (Enter 1.2)

Then choose the units using the arrow keys, push the ^ arrow key (Mhz).

1.200000 MHz will be displayed on the display.

To cancel the number mode press "Shift" "Cancel" (shift Enter Number).

The same procedure could be used to set the amplitude value. **Note** that to use mV you have to press "Shift" then the up or down arrow.

To set a dc offset voltage:

Press "Offset", the enter numbers as shown above.

To stop the number from flashing, move the cursor to the left of the left most digit (be sure that you are not in number entering mode by pressing "Shift" "cancel.

To generate a square wave:

By pressing the square wave symbol "FM" and using the previous procedure you can generate a square wave.

You can set the duty cycle of the square wave (the percentage of the time the signal is HIGH)

Press "Shift" "%Duty" Then you can use the knob or the 'Enter Number" procedure to set it between 20 and 80.

The Digital Multimeter:

Start by pressing the ON OFF button

You can perform a self-test by pressing the 'Shift" key then press "ON" to turn the device on and continue pressing the "Shift" key for more than 5 seconds, you hear a beep, then the self-test starts after you release the 'Shift". If the self-test is successful, you see "PASS" on the display.

The multimeter automatically selects the range for you; you can select the range buttons. The range buttons are 4,5,6 ($\vee$, ^ and Auto/man) you press the "Shift" followed by one of these keys to choose the required range.

You can choose the multimeter function by pressing one of the keys "DC V", "AC V", "Ω 2W", "Freq", or "Shift" "Freq" for period.

Now, you will generate a signal and measure it.

Use the function generator to generate a sinusoidal signal with 500 Hz, and 1 V PP (Peak-to-peak).

Note: When you set it to 1 V PP you get a sinusoidal signal with amplitude of 1 (peak to peak is 2 V not 1).

The function generator has a BNC splitter. One cable is a BNC-BNC that usually will be connected to the scope; the other is BNC-Alligator that Is used to supply the signal to the breadboard. In this case, connect the clips to the probe pins of the multimeter.
IMPORTANT avoid the contact between the red and black alligator clips by proper spacing them apart from each other.

Set the multimeter to measure "AC V"

What is the displayed value? Is it the same as what you set the functional generator output to be? Discuss in you report any discrepancies.

The DC Power Supply:

That is the easiest device to use.

Turn it ON using the "ON" "OFF" switch.

Choose the range +6V, +20V, -20V

Use the knobs to adjust the required voltage.

Use the banana clips to take the output to the circuit you want to power.

Note that the red is the active wire and the black is the ground.

IMPORTANT avoid contact between the red and black wires.

The Oscilloscope:

You will use Tektronix TDS 210 DIGITAL REAL-TIME OSCILLISCOPE

The oscilloscope consists mainly of a CRT (Cathode Ray Tube). There is an electronic beam that is generated and directed towards the screen. The beam illuminates the screen at the point it hits it. The beam goes through two sets of metal plates that are charged to deflect (bend) the beam as Shown in Fig. L1.1.

The first deflection plates can move the beam up and down (vertical), while the second set moves it left and right (horizontal).
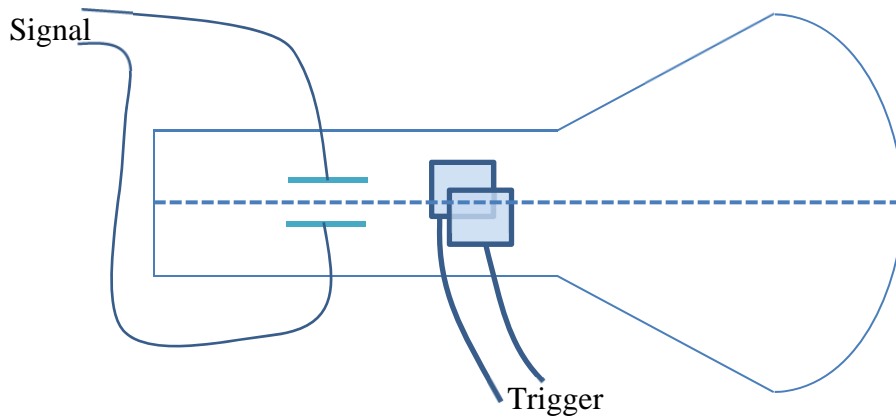
Fig. L1.1 Oscilloscope

The signal to be displayed is connected to the vertical plates. According to the signal strength the beam moves up and down (the higher the signal value, the more up the beam moves). The horizontal plates are used to make the beam sweep the screen from left to right many times per second so we can see a picture of the signal on the screen. This is a very simplified description of the scope, we will get back to it when we talk about the trigger source. Now start using the scope.

Push the ON OFF switch

A self-test will start, after few seconds it will be ready for use.

First test.

Connect the alligator clips of the BNC cable connected to the scope channel 2 to the PROBE COMP reference leads on the scope front panel (black clip to ground and red clip to the test signal lead).

Push the AUTOSET button, within few seconds you will see a 1KHz square wave with 1 V peak-to-peak.

You can push CH 1 MENUE and CH 2 MENUE to alternate between turning each channel ON and OFF.

Now, use the function generator to generate a 1 V P-P sine wave with a frequency of 500 Hz, connect it to CH 1 and press AUTOSET.

You will see a display of a sine wave on the CRT display.

Let us get familiar with some of the buttons on the front panel of the scope.

Push CH 1 MENUE to get the menu for CH 1.

Under vertical and CH 1 rotate the "POSITION" knob, what do you see?

Rotate VOLTS/DIV knob, what you see?

Center the signal in the middle of the display, and change the VOLTS/DIV until you see on the bottom left on the screen "CH 1 500mV". That means every square on the screen represents 0.5 V. What is the amplitude of the signal? Is that the same amplitude you generated from the function generator?

You can take a snapshot of the scope screen, ask the TA how to do this

Rotate the SEC/DIV knob under HORIZONTAL, what do you see

Use the SEC/DIV knob to set the scale to 1.00ms as shown in the bottom middle of the screen (again, that means a square is 1.00ms).

If you look at the peak-to-peak you notice it takes 2 squares, that means the peak to peak is 2 squares * 500 mV per division=1V that is the P-P of the sine wave.

Also look at the cycle time, it takes 2 squares (2 divisions), each division is 1 msec, that means the frequency is 1 KHz.

Look at the menu for CH 1 you see the following

Coupling (DC AC Ground): This control allows the AC coupling (basically a capacitor is used to block the DC component of the signal and you will see the AC component only). DC coupling (the DC component will be displayed together with the AC component). Ground means no signal (ground) is displayed.

BW Limit (ON and OFF): If ON, it limits the BW of the signal displayed (uses a low pass filter to filter noise and high frequency component).

Volts/Div (Coarse Fine): Self explanatory

Probe (1X 10X ...): This allows magnification of the signal by the set factor

Invert: Inverts the signal

Press the TRIGGER MENU button, now you see the menus for the trigger displayed on the right side of the display. The fields are

Edge-Video: It will you choose between an edge trigger or video. In this lab it will always be an edge.

Slope: You can choose rising or falling edge.

Source: As we mentioned before, a trigger signal will be applied to the horizontal plates to start sweeping from left to right. The question is when to start the scan. There are three different sources to trigger the scope.

- CH1 or CH2. The signal itself. The trigger will start when the signal reaches a specific threshold (usually zero, but it could be set to any number).
- AC The power line (50 or 60 Hz depednding on your location in the world)
- External or EXT5: In this case an external source is connected to the scope to provide the required trigger. EXT5 divides the signal by 5 so you can use a larger signal for the external trigger.

Mode: How the scope behaves in the absence of a trigger, usually we use NORMAL

- Auto: This mode allows the scope to acquire the signal even if there is no trigger
- Normal: Only acquire the signal if there is a trigger
- Single: It acquires the signal only one wave when you push RUN

Coupling: Coupling determines what part of the signal will be passed to the trigger circuit

- DC: Passes both AC and DC components.
- AC: Blocks the DC components.
- Noise Rejection: Lowers the trigger sensitivity to reduce the chance of falsely triggering on noise.
- High Frequency Rejection: Blocks the high frequency portion and passes only the low frequency components.
- Low Frequency Rejection: Blocks the low frequency portion and passes only the high frequency components.

Part II: basic DC Measurements
PRELAB:

Find the voltage across each resistor (V_{Rx}) and the current in each resistor (I_{RX}) in Fig. L1.1 using both mesh and nodal analysis.

Assume the following values for the different R

R_1 = 1 KΩ, R_2 = 4.7 KΩ, R_3 = 10 KΩ, R_4 = 4.7 KΩ, R_5= 1 KΩ, V_1= 5V, V_2= 12V
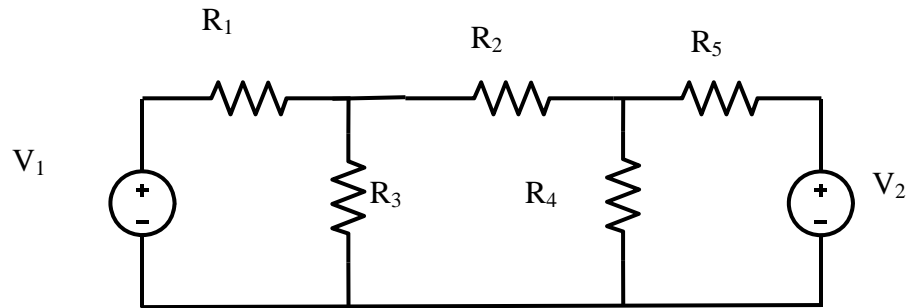
Fig. L1.1 Series parallel circuit

LAB:

Construct the circuit in Fig. L1.1. Set V_1 to 5 V (always measure it with a voltmeter, do not depend on the displayed value). Set V_2 to 7V.

Chose the values of R given in the prelab.

Measure the actual values of the resistors.

Measure the voltages across the resistors

Calculate the voltage across the three resistors using the values measured instead of nominal values.

Show the values you measured and the calculated values in a table like the following

	Prelab	Measured	Calculated using measured values of R
VR1			
VR2			
VR3			
VR4			
VR5			

Repeat the last four steps but with measuring currents instead of voltages in the three resistors and compare them to the calculated values using a similar table.

LAB 2: Thevenin and Norton Theorems

OBJECTIVE:

The objectives of this lab are

- To learn how to construct Thevenin and Norton equivalent circuits for any circuit.
- To verify the results using the constructed circuit in the lab.

PRELAB

Calculate the Thevenin's and Norton's equivalent of the circuit shown in Figure L2.2 as seen by R_2.

LAB

Part I: Voltage Dividers

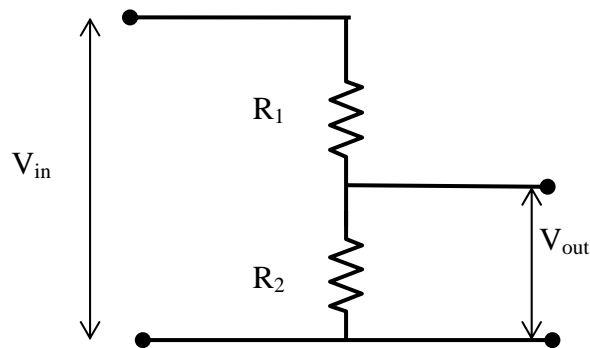Construct a voltage divider as shown in Fig. L2.1

Fig. L2.1 Voltage divider

Set V_{in} to 10V

Choose R1 = 4.7 KΩ, R2 = 6.8 KΩ.

Calculate V_{out} / V_{in} using the nominal values above.

Measure V_{out} and compare it to V_{in}

Measure the actual values of R_1 and R_2.

Calculate V_{out} using the measured values instead of the nominal values.

Compare the measured value with the value you calculated in the prelab and the calculated values from measuring the actual resistors.

Now, load the output with a 5.6 KΩ resistor and repeat the previous steps

What effect the load value has on the voltage ratio? To understand this part, assume that a load is using V_{out}. The value of the load resistance is R. Draw a graph showing the relation between R and the V_{out}.

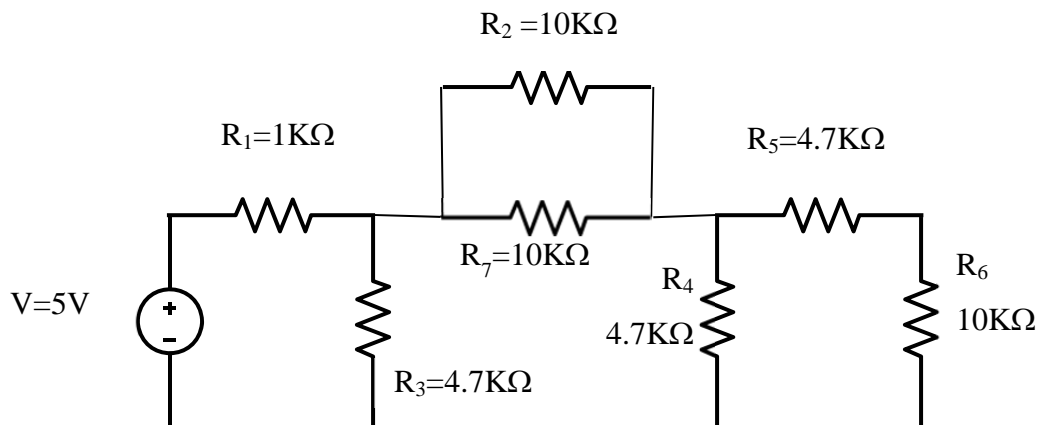## *Part II Thevenin and Norton's Equivalent Circuits*

Fig. L2.2 A circuit for Lab 4

Construct the circuit shown in the Figure L2.2. Choose. We want to construct the Thevenin and Norton equivalent circuit.

Measure the actual values of the resistors.

Thevenin equivalent circuit

Thevenin equivalent circuit consists of a voltage source in series with a resistor. The voltage source equal to the open circuit voltage seen across the load resistor (R2 in the figure above). The series resistor is the resistance seen at the load after short circuiting the voltage source.

Disconnect the load resistor (R2) and measure the voltage seen at R2. This is the Thevenin voltage source.

Disconnect the load resistor, short circuit the voltage source and measure the current seen at the load output.

Construct the thevenin equivalent circuit and calculate the current in the load, and the voltage across the load. Compare that with the measured values.

Norton's equivalent circuit

Repeat to construct Norton's equivalent circuit.