YORK UNIVERSITY
DEPT. OF COMPUTER SCIENCE AND ENGINEERING

A laboratory Manual for Electronic Circuits Lab
ENG2210

Prepared By
Prof. Mokhtar Aboelaze
January 2015

PREFACE

This laboratory manual is intended for use in ENG2200 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 performing 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 is 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 you 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.

ACKNOWLEDGEMENT

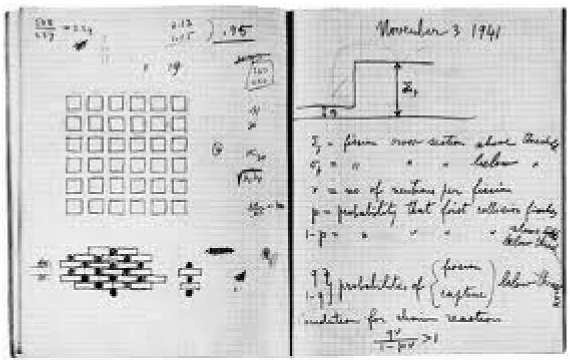
I would like to thank Konstantin Bolshakov for proof reading this manual. He took this course in 2013, and then he put a lot of time and energy to make my life, as well as, the students' life, a lot easier.

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 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. You can 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 breadboards, surface mount resistors (used in printed circuit boards), variable resistor (potentiometer), and thermistor (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.

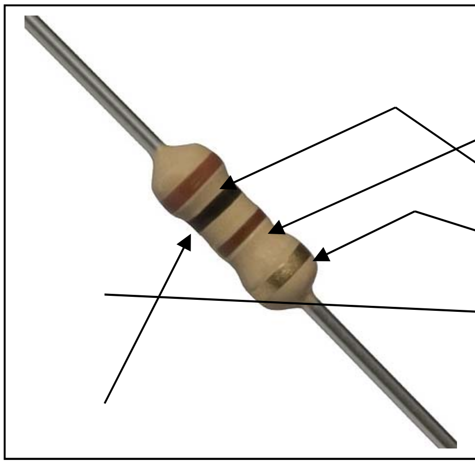
Figure 1. Different types of resistors

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		-	$X10^{-1}$	± 5%
Silver		-	$X10^{-2}$	± 10%
None		-	-	± 20%

For example



For example: consider the image to the right, the bands are Brown (1), green (5), brown (1), gold (5%)
15×10^{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. Also they have a relatively high series conductance which makes them unsuitable for high voltage application (why?). **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*103pF=12 nF with max. V=330V

Breadboards

Breadboards 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.

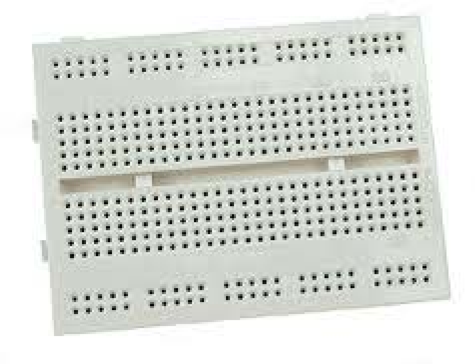


Figure 4. Breadboard

If you open the breadboard, you will find that these holes are connected with wires. For example, in the Figure above, 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

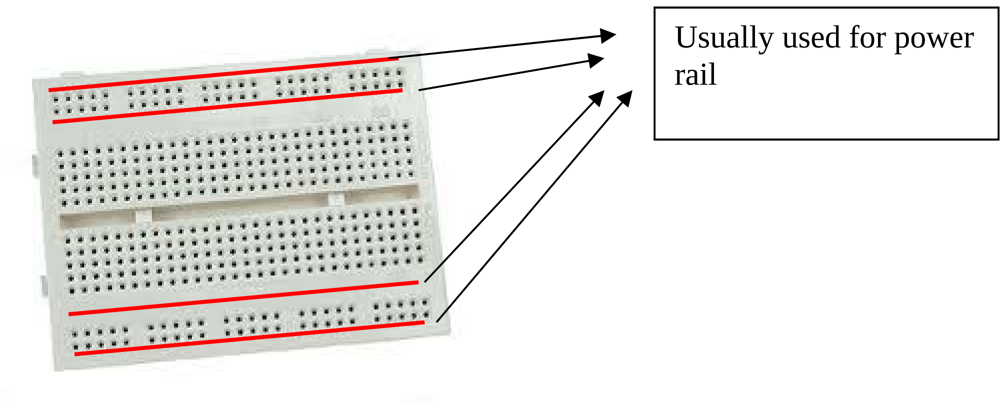


Figure 5. B readboard showing the power rail

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

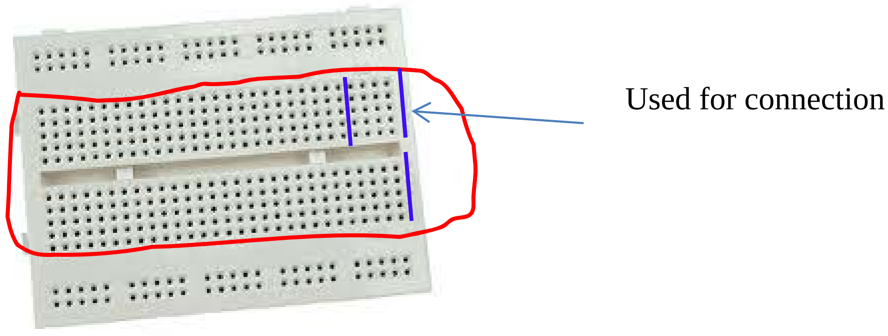


Figure 6. Breadboard showing the matrix connection

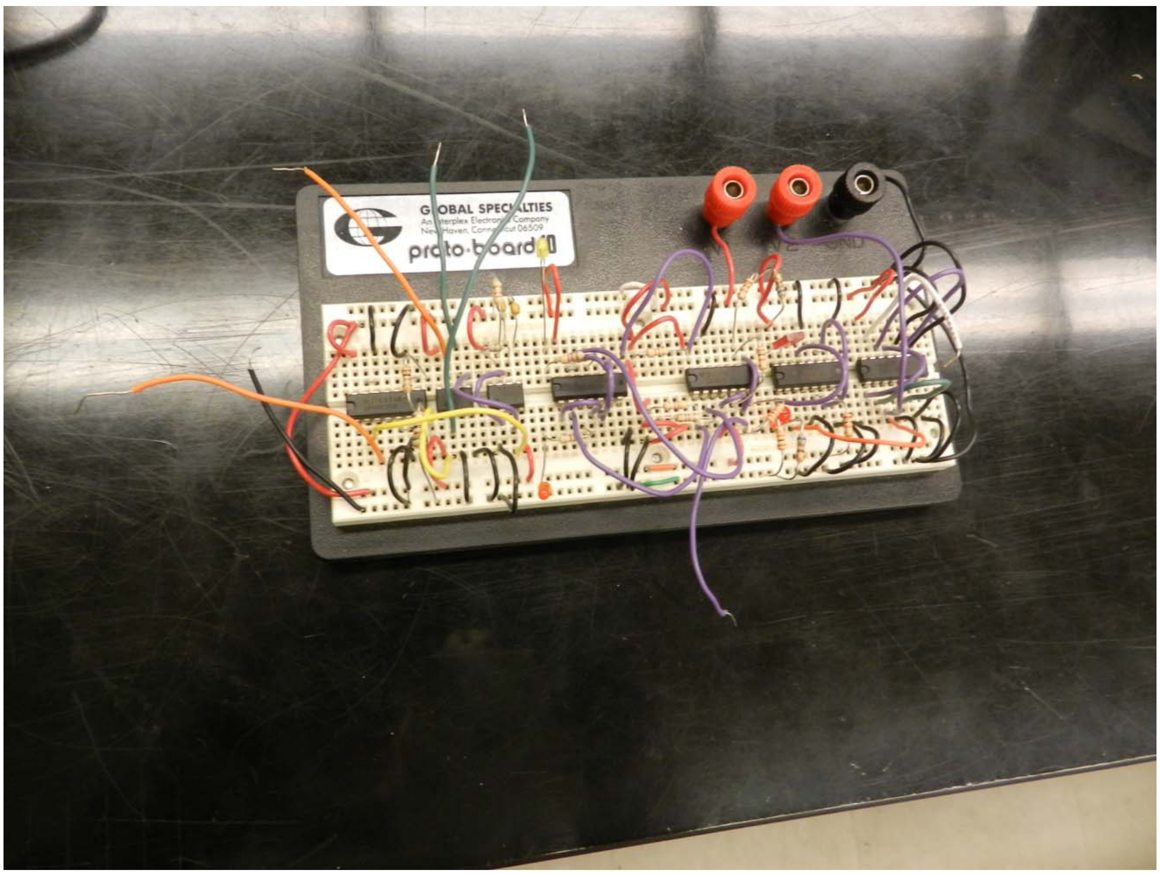


Figure 7. Breadboard with the connection

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's in the lab; some of these equipment are 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 are electrical. Electricity can harm or kill you. You have to be very careful in order not to 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 series 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 are 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

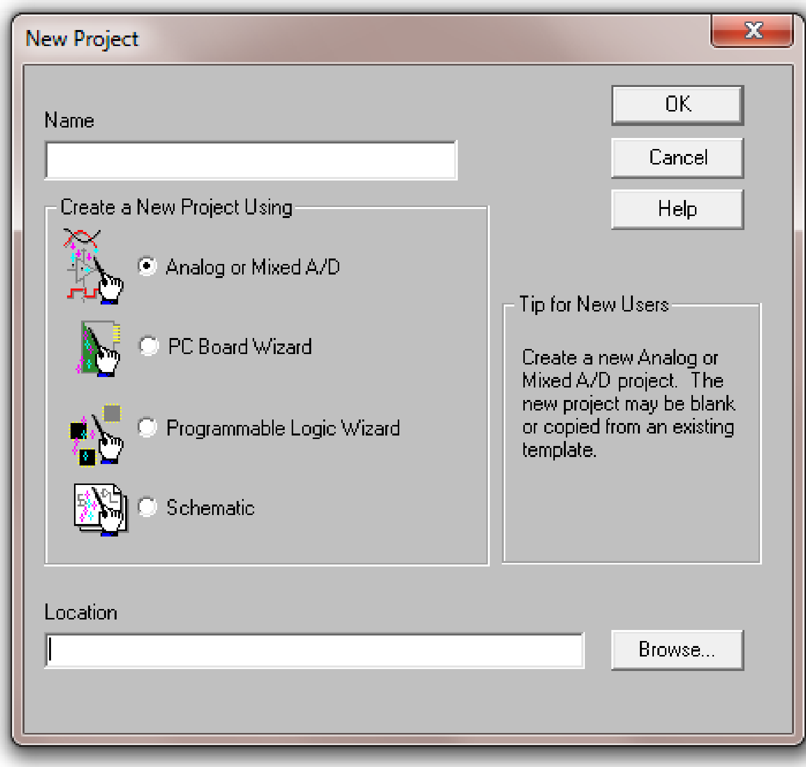
For the prelab, just you have to answer the questions in the lab document.

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

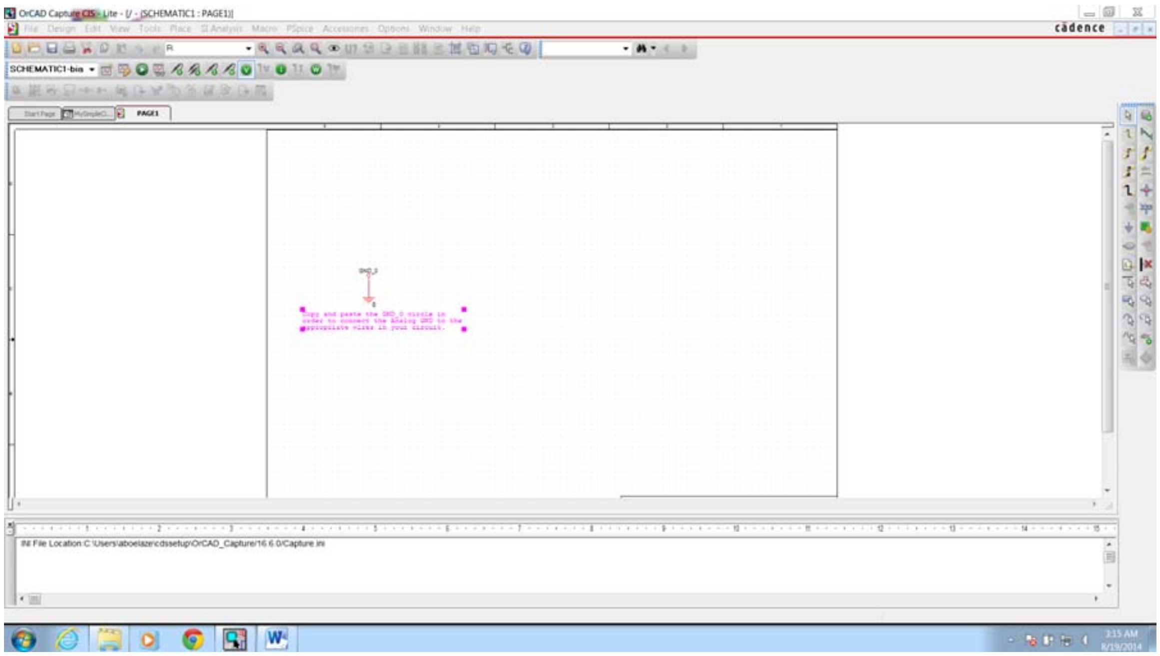
Abstract: A brief statement (few sentences) mentioning what you did in the lab and your results.
Analysis: If you were asked to perform any theoretical calculations, that should be done in the Analysis section.
Experiment setup and results: That is the main part of your lab report. If you constructed 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.

LAB 0_A: SPICE Tutorial

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

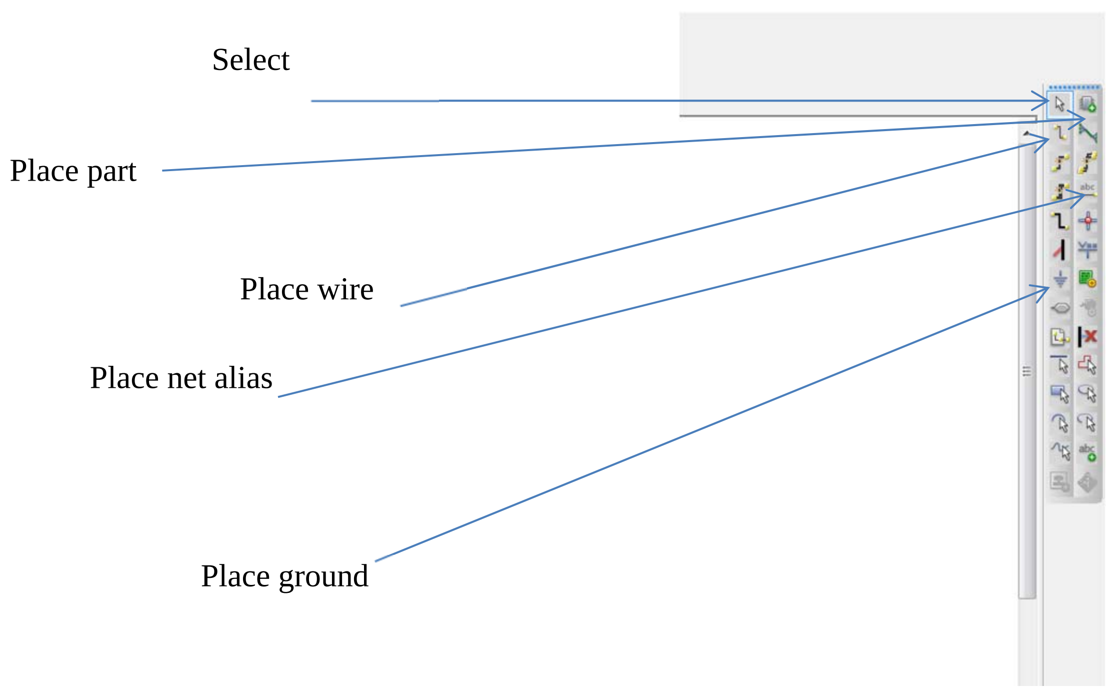


Choose Analog or Mixed A/D
Browse to the appropriate directory location, and name the project MySimpleCircuit
Then OK.
A new project Window appears, choose “Create Blank Project”. The following screen appears



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. If you did not get the ground node do not panic, you can place it from the "part" menu.

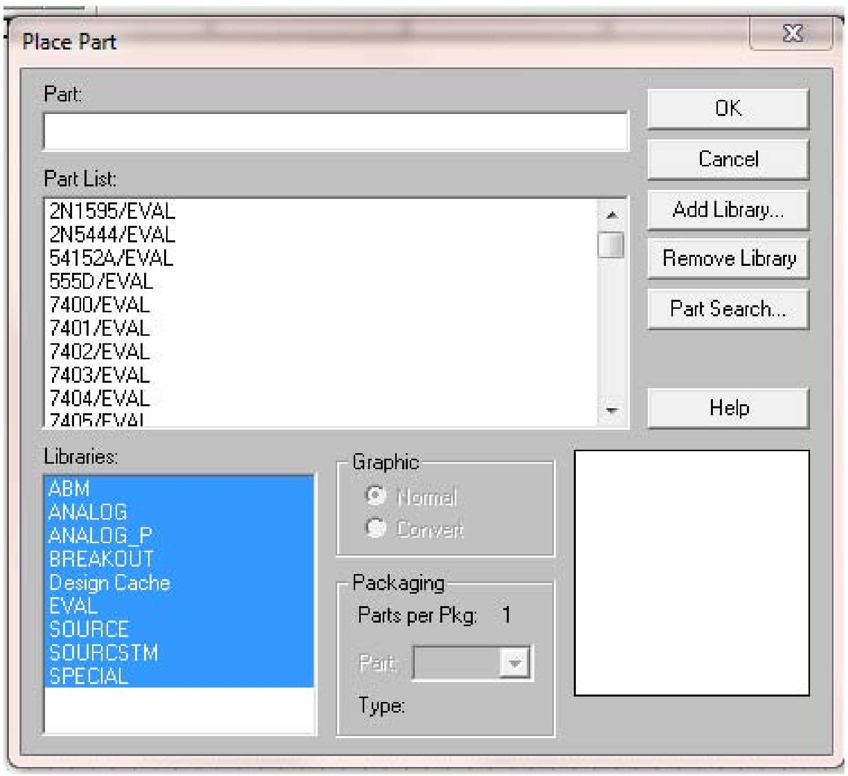
Every circuit must contain a ground and the name of the node is "0" that is the number zero.



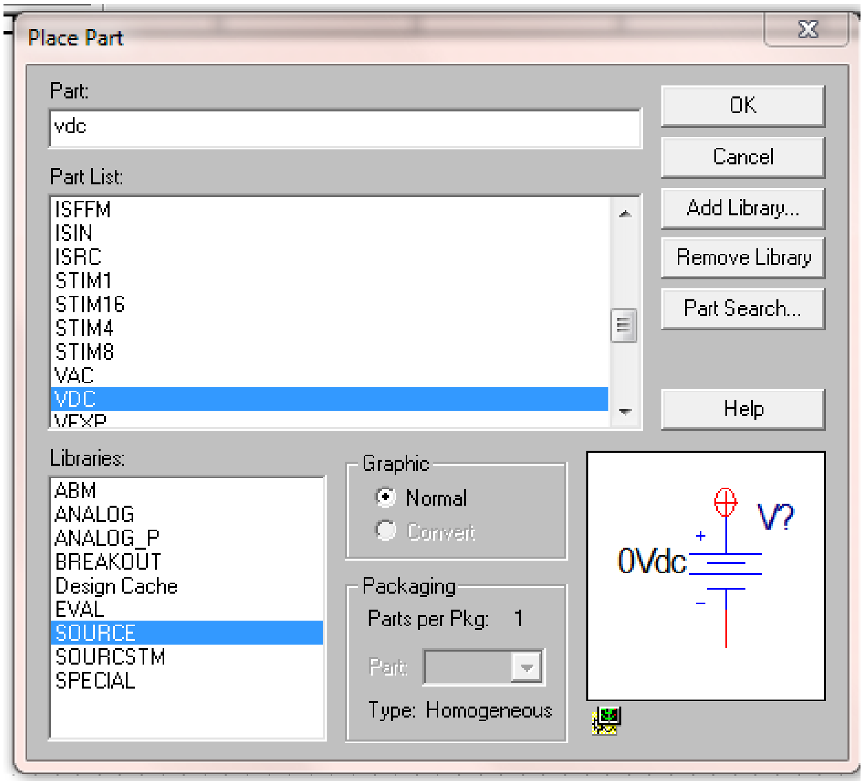
On the right there is a group of fast access keys, the most important is the place part (could also be accessed from the menu bar Place → part. The place ground button (ground is also a part you can access it from the place part button or menu. The place wire (also from the menu Place → Wire). Place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

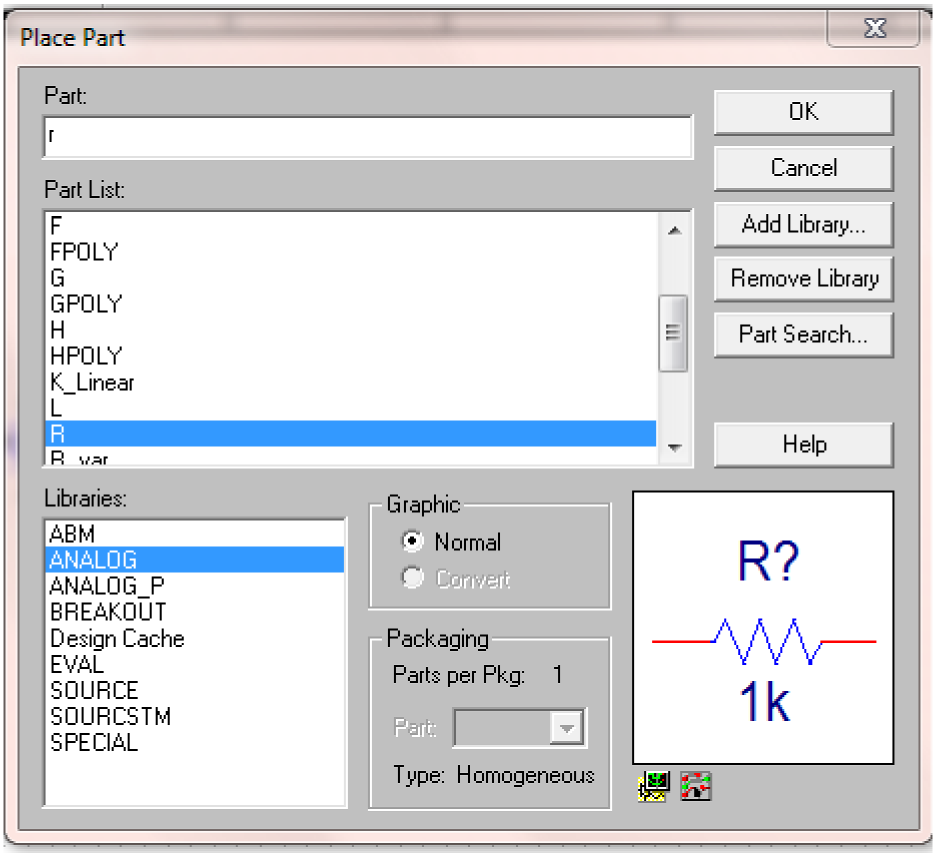
The window could be floating, or could be pinned to the right upper corner of the schematic page. Also it may contain different libraries depending on the initial configuration of the software.



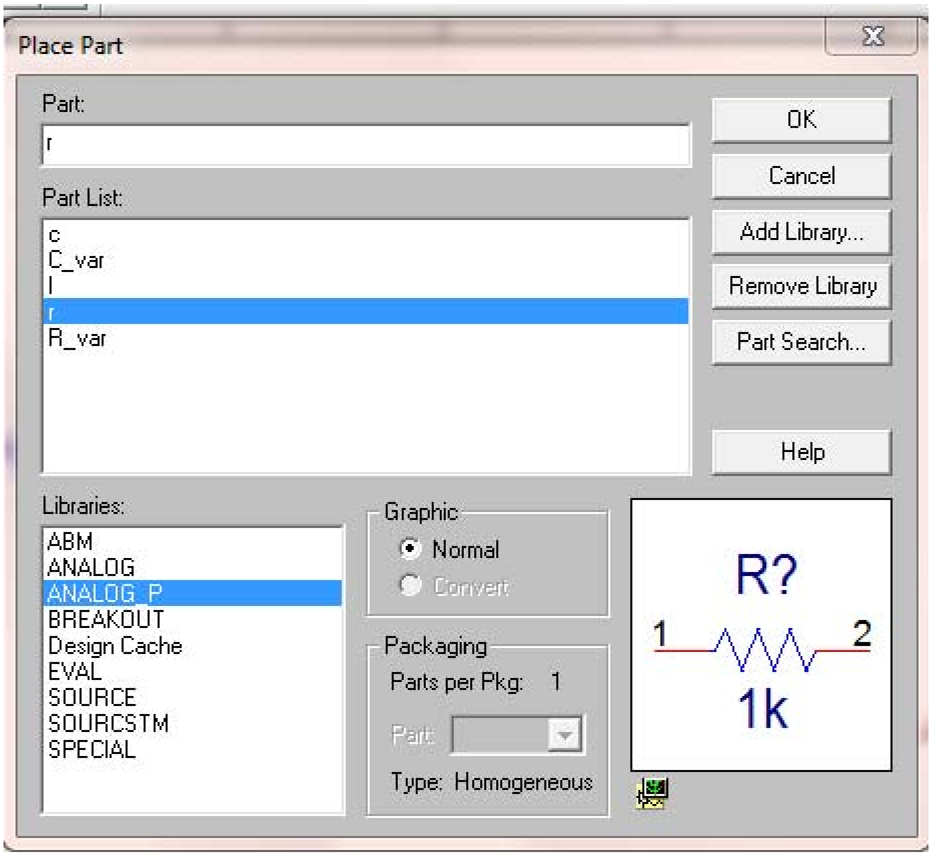
Highlight Source library and write Vdc in “part”. As soon as you right V, all parts that start with V appears, Choose Vdc (A DC power supply) as shown below.

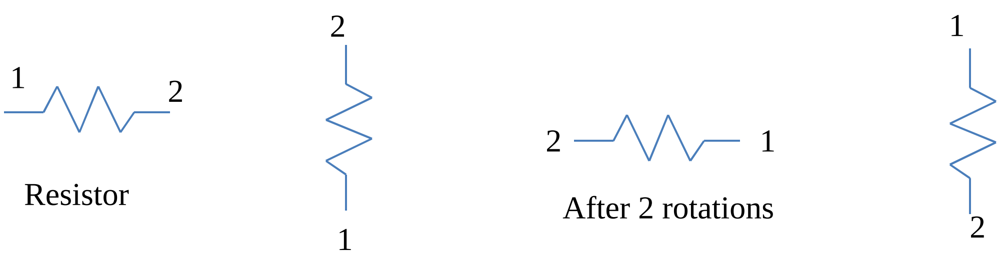


After you chose VDC click O.K (or double click on the part name). The window disappears and the symbol appears attached to the mouse cursor. Go anywhere in the schematic and left click, a copy will be inserted. After that the part still attached to the mouse cursor and you can insert more DC power supplies if you want. We just need one, so right click on the mouse and choose end mode (or hit the Esc key).
Highlight ANALOG library and type r in part, the following window appear.



Click 4 times in the schematic to place 4 resistors
Note
If you choose ANALOG_P library, then you type r in part a resistor will appear as shown below.
You notice that the shown resistor appears identical to the resistor in ANALOG except for the 2 numbers 1 and 2 at the 2 ends of the resistor.
Important
Note that in Spice every resistor (2 terminal devices in general) added has 2 terminals 1 and 2. When you rotate the part (by highlighting it and pressing “r”) the part is rotated counter clockwise. So when the shown R is rotated (even the one without the node numbers displayed) it will rotate such that terminal 2 up and 1 is down as shown in the figure blow..
When SPICE calculates the current in the resistor, it will take the positive direction of the current entering from 1 and leaving from 2. If you rotate it once such that 2 is up, if the current is entering from 2 it will be shown as negative current. You can rotate it twice after that to make terminal 1 up.





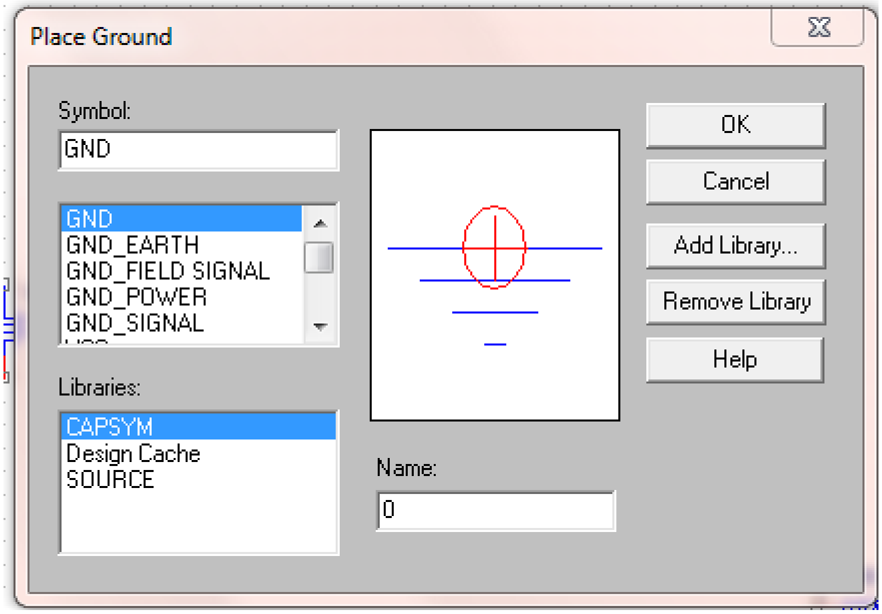
Resistor

After 1 rotation

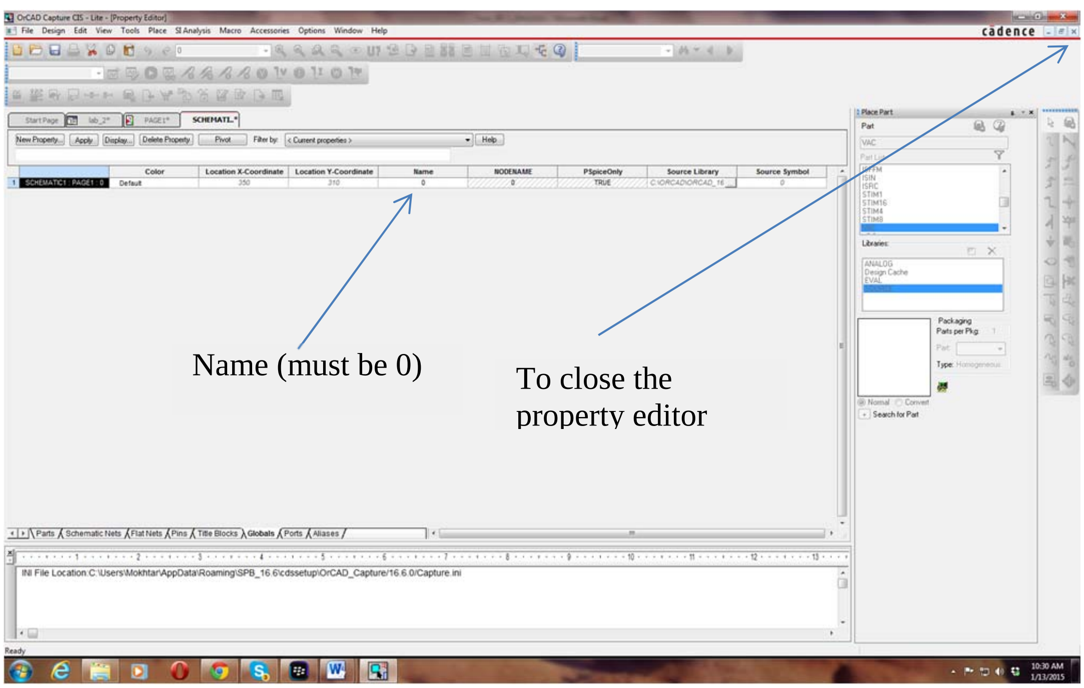
After 2 rotations

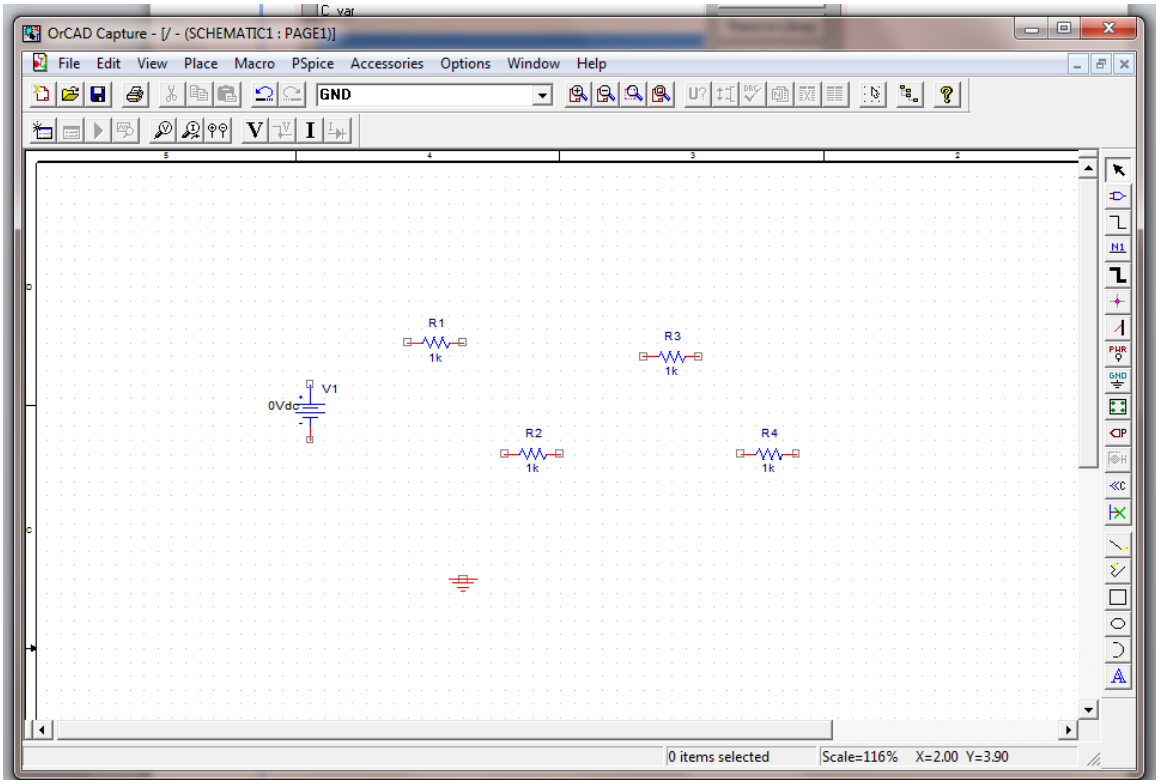
After 3 rotations

Then you have to add a ground. In SPICE THERE MUST BE A NODE NUMBERED 0 without it SPICE will not work.
Click on the button “GROUND” in the side menu as shown below, place the ground anywhere in the schematic. BE SURE TO PUT THE NUMBER 0 in Name. SPICE assumes that every circuit has a ground (node 0), if there is no node 0 we will not be able to simulate your circuit. If you did not do that, you can do it later as will be explained

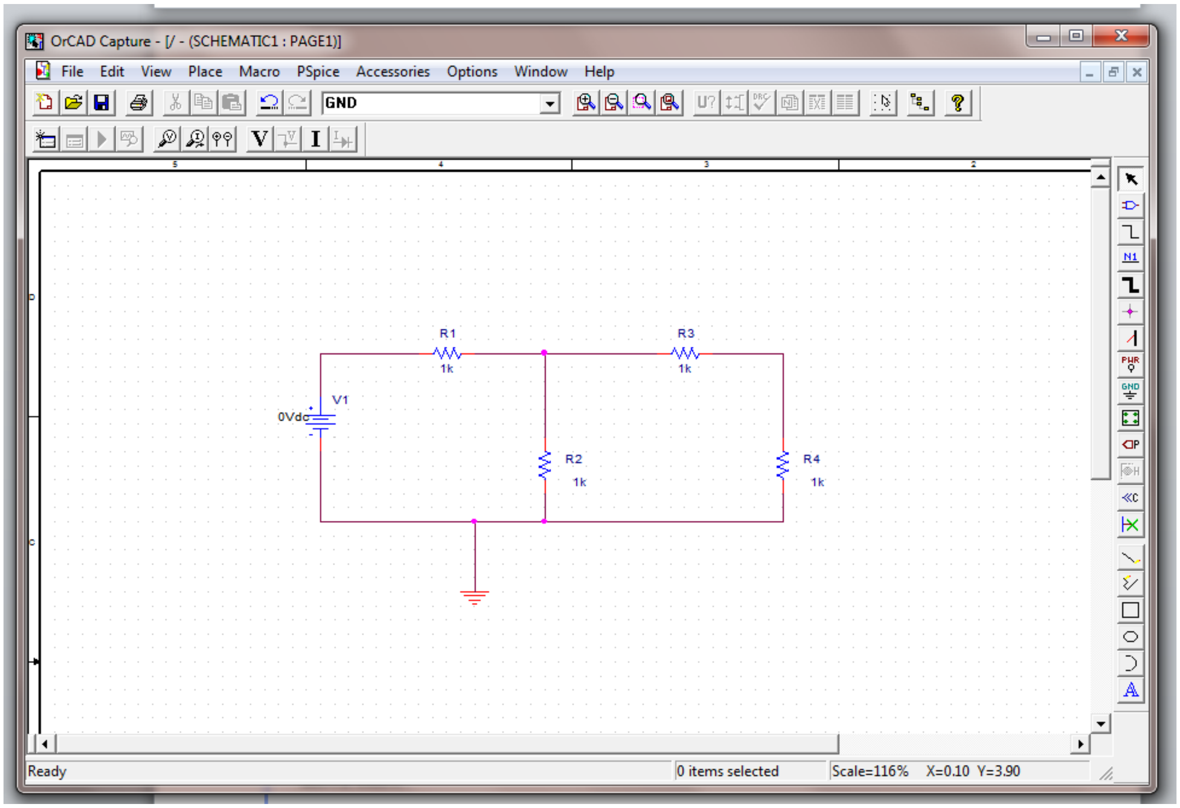


Now the schematic looks like the following.
Note also that you can place part by clicking on the part button in the side menu.
If you forgot to name the ground "0", there is another way to do it.
Double click on the ground symbol. The property editor appears, in the field called Name type 0.

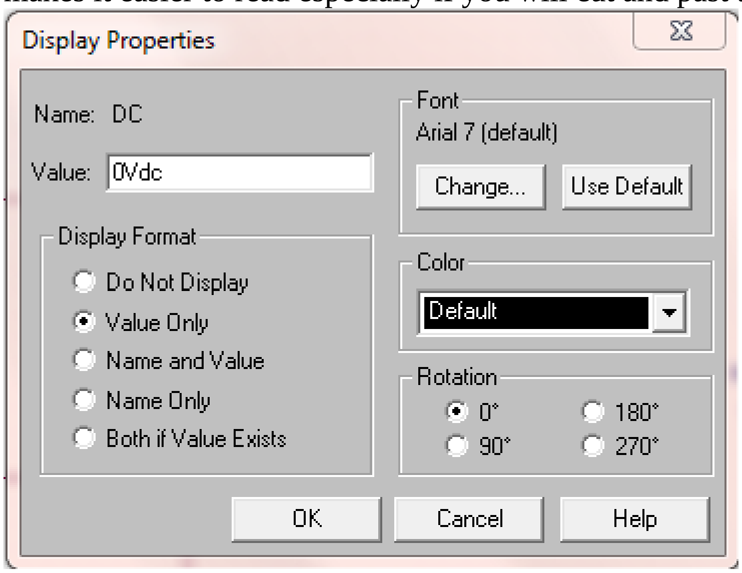




Click on R2, then CTL-R to rotate the resistor into the vertical position. Repeat the same for R4. You can also right click on the resistor and choose rotate.
Now we want to connect the parts using wire. go to the menu Place → wire, or click on wire in the side menu (third button). Go to V1 click on the upper terminal of V1 and move the cursor to R2 and clock on the left terminal of R2. Repeat to connect the parts as shown below.

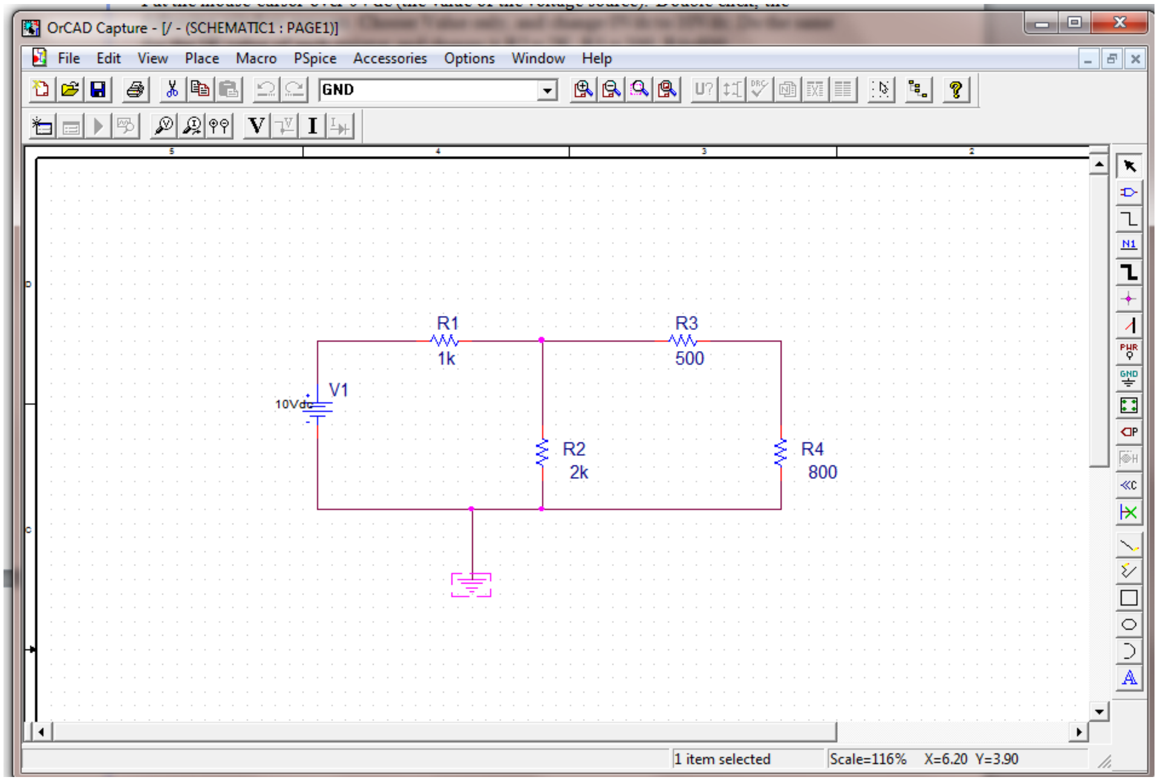


Note that the value of the voltage source is 0V, all the resistors have a value of 1 KΩ. The names of the resistors are automatically numbered R1, R2, …
Put the mouse cursor over 0Vdc (the value of the voltage source). Double click, the following window appears. Choose Value only, and change 0Vdc to 10Vdc. Do the same for the 1K value of each resistor and change it R2 = 2K, R3 = 500, R4=800. By clicking on the part name (R1) you can change the name of the part also. You can also click on Change (Font part) and you can change the font and size of the displayed value. That makes it easier to read especially if you will cut and past the circuit in your report.



The schematic now appears as follows. (I changed the font size to 10 to make it easier to read). Double click on the name (R1 for example), click on font change and set the size to any size you want.

Click on the wire connecting the top of R4. Click on N1 from the side menu, or Place → Net Alias. A square a window appears, put Vout in Alias, after OK move the cursor to touch the top wire of R4 and place it there. The right click and end node. Then save the file using File → Save.



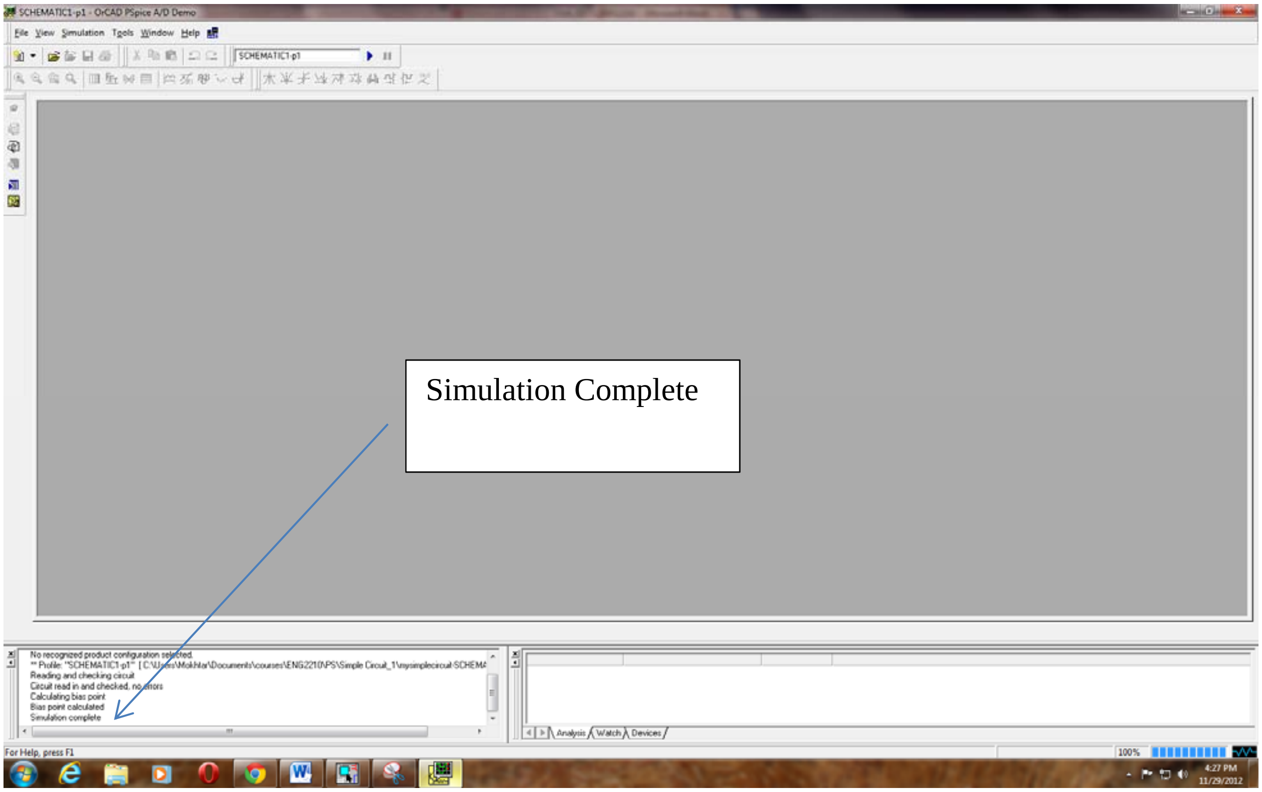
Now the file is saved and ready to be dealt with by SPICE.

To start the simulation, go to the top menu PSpice → New Simulation Profile. In the name field, enter P1 and none in the "inherit from" field, click create. A simulation setting windows appears

In the Analysis tab, set the "Analysis type" to "Bias Point". then OK.

Now we are ready to run the simulation. Go to PSpice → Run and the simulation starts running. The following window appears. The simulation complete line in the bottom left window indicates the simulation has finished running.

You can use the menu View → Output File and the output file appears.

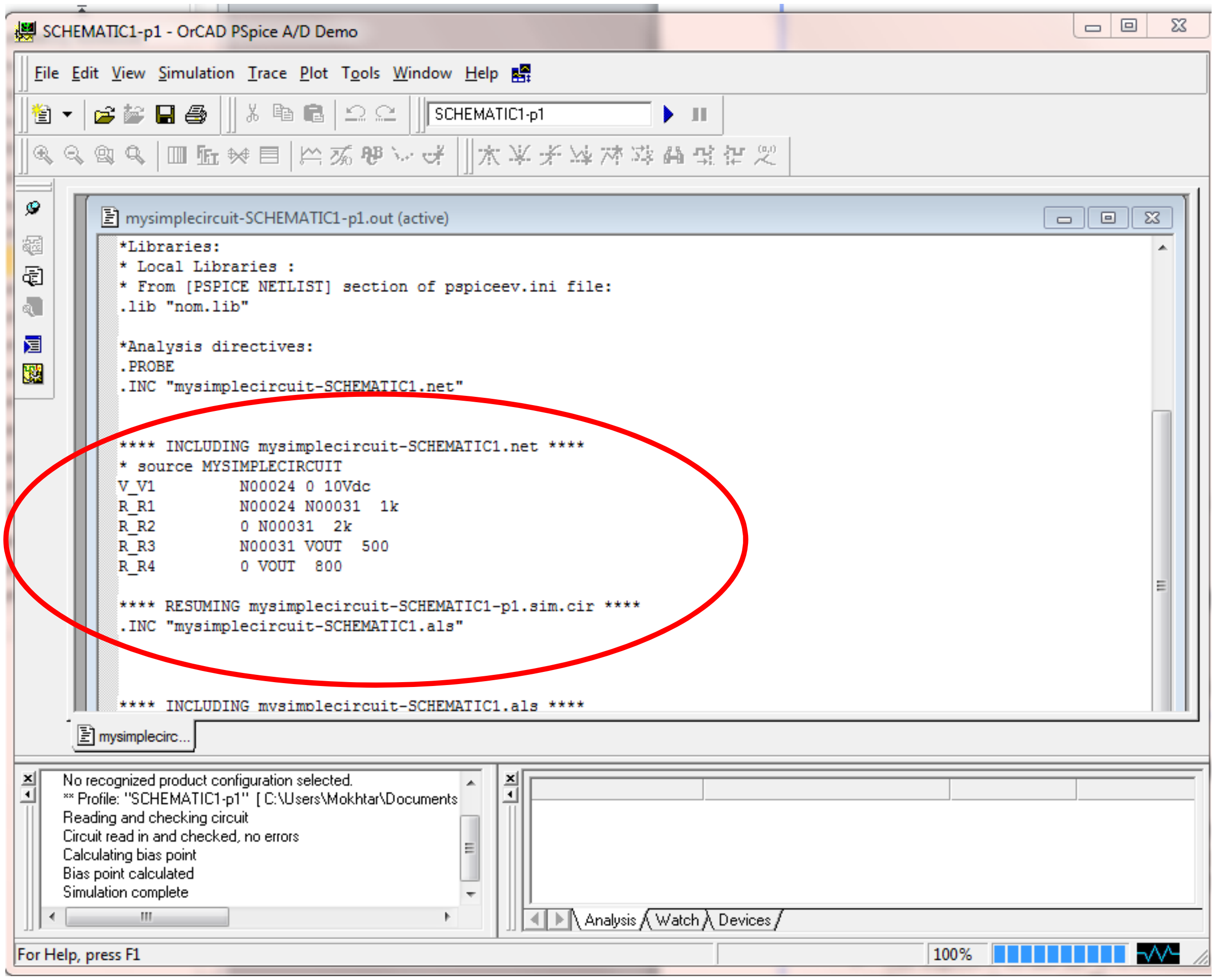


The output file appears in the top window. The output file contains a lot of information about the circuit and the output; here we concentrate on 2 parts of the file.

The first part is the SPICE netlist (shown in the red oval).
It describes the circuit we constructed. For example the first line V_V1 N00024 0 10V. Since the element starts with a V, it is an independent voltage source between nodes N00024 (the positive terminal) to node 0 (ground) and the value is 10 V. Spice picks numbers or names for the net that we did not specify. Fir the alias we used Vout, it appears as the name of one of the node.
If you scroll down, the last line is shown below. It indicates the voltage values of the different nodes (for example, Vout is 2.7119 V and so on).
XXXX show enable V and I

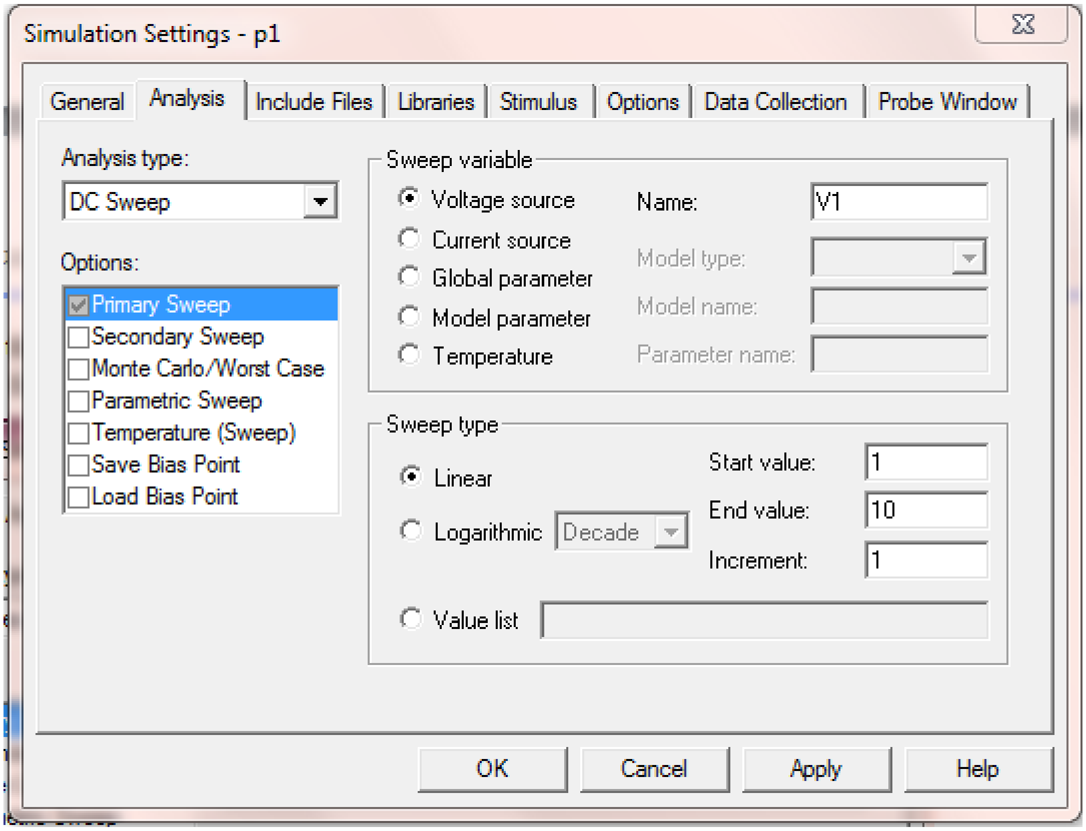
NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE

(VOUT) 2.7119 (N00024) 10.0000 (N00031) 4.4068

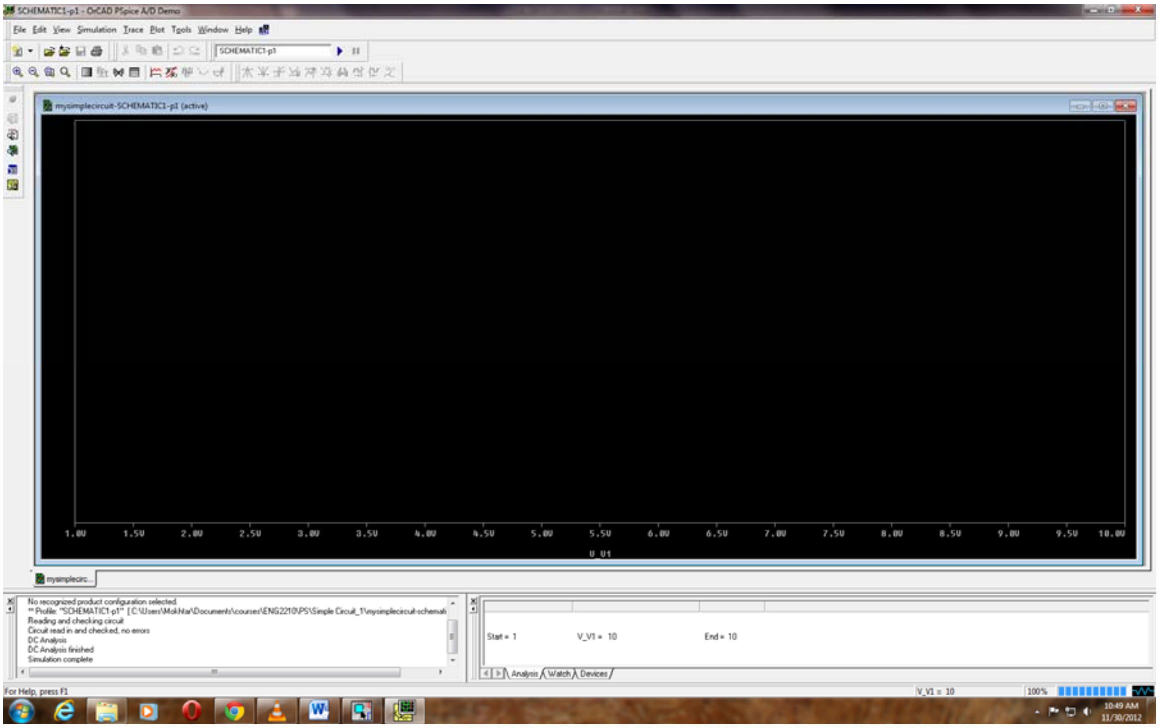


DC Sweep

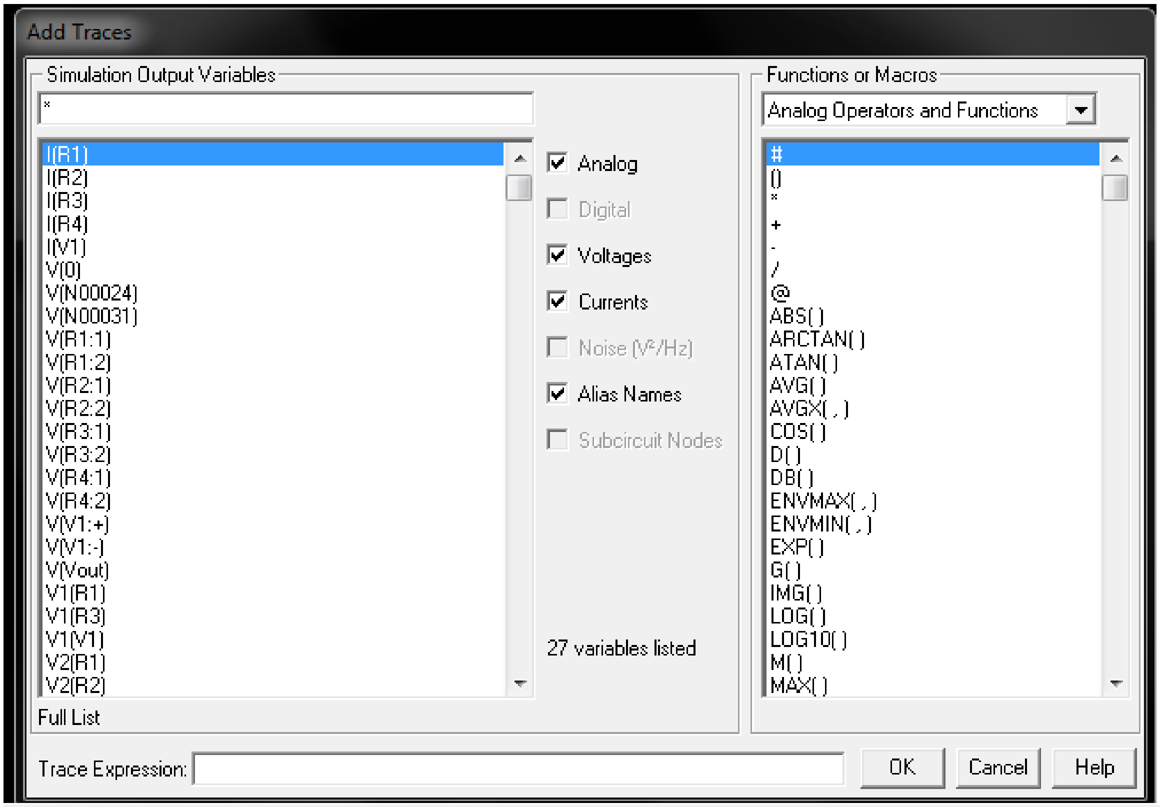
Now, we will perform a DC sweep. We change one (or more) of the DC sources and check the voltage across, or current, in an element.
Go back to the schematic diagram, top menu. Choose PSpice → edit simulation profile, you get the following window

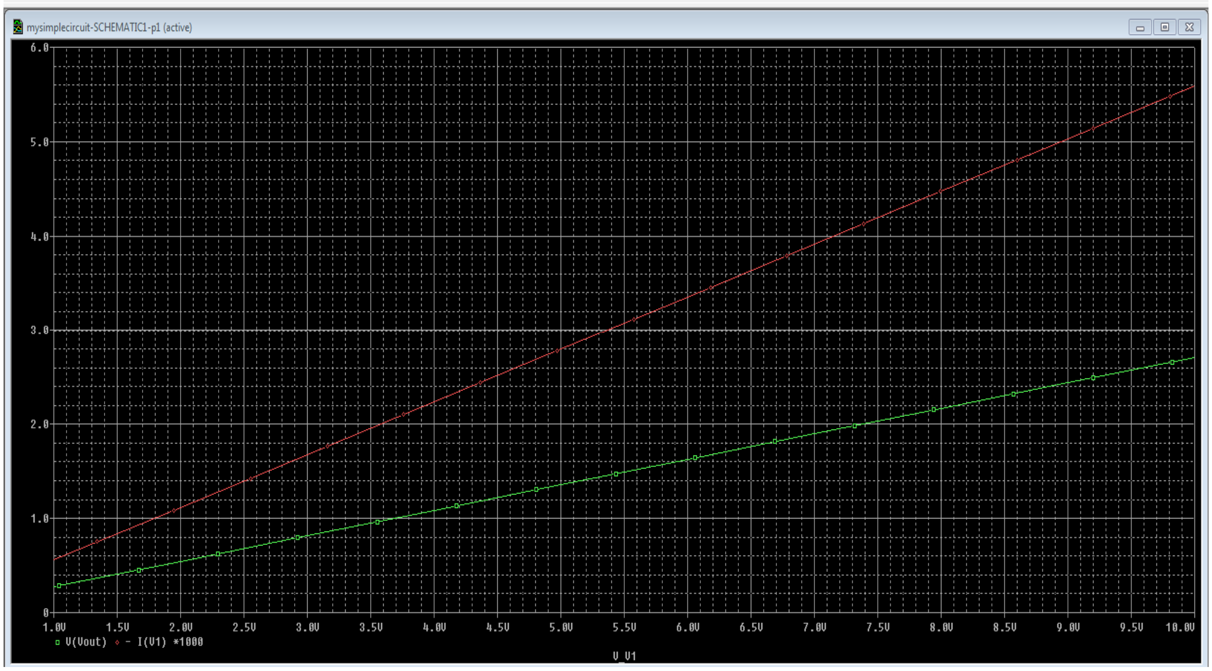


In the Analysis tab set the "Analysis type" to be DC Sweep. Choose primary sweep from "Options".
In Sweep variable, choose Voltage source
Set name to V1 (The name of the voltage source in your schematic).
In Sweep type, choose linear, start value = 1, End value = 10, increment = 1.
PSpice will change the voltage source V1 from 1 to 10 in linear increment of 1.
Now Apply and O.K..
In Schematic choose PSpice → run
A windows 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. 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[Vout], note that is the net label we choose for our output voltage
V[Vout] appears in the trace expression field, click O.K.
Now the diagram showing the relation between Vout and V1 appears.
Again, go to top menu Trace → add trace
The previous window appears.
Click on “-“ from functions or Macros
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



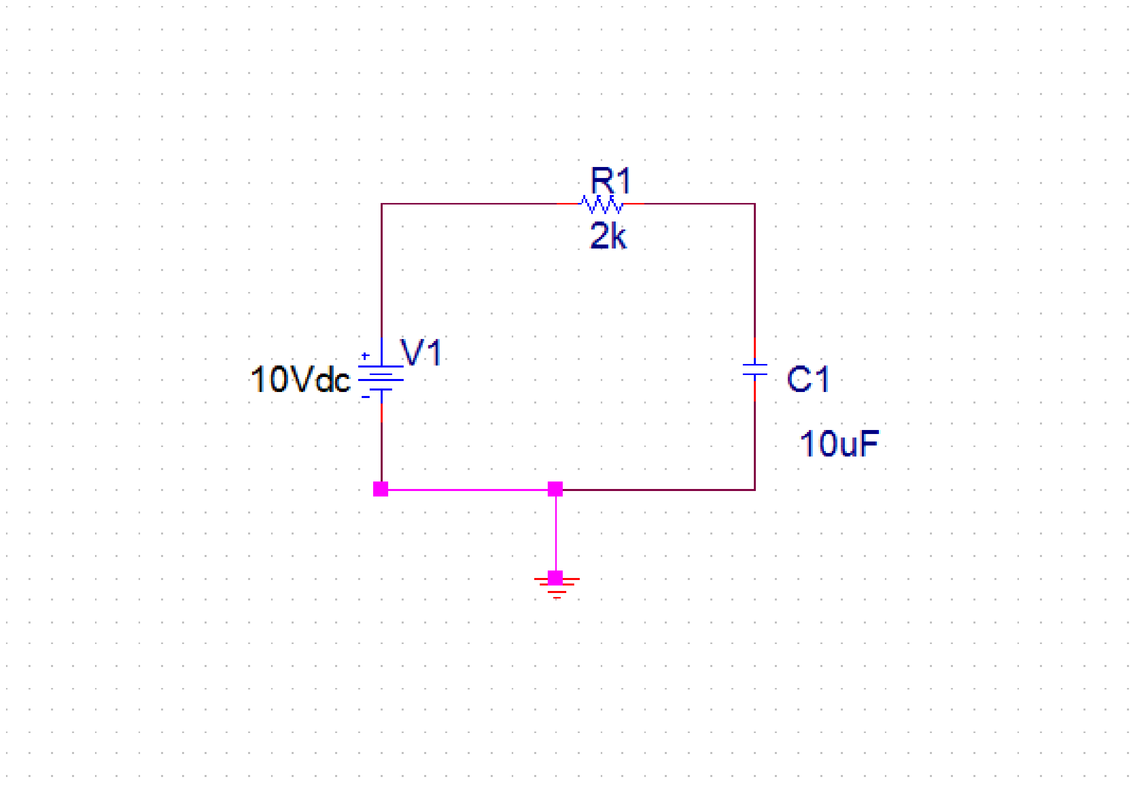


The plot shows two graphs, the green one is Vout and the red one is I[V1].
Now, why did we do –I[V1]*1000?
I[V1] is the current flowing in the v1 source, by definition the current in a voltage source is flowing from the positive terminal to the negative terminal. That is a negative quantity (since the actual current is leaving the positive terminal of V1).

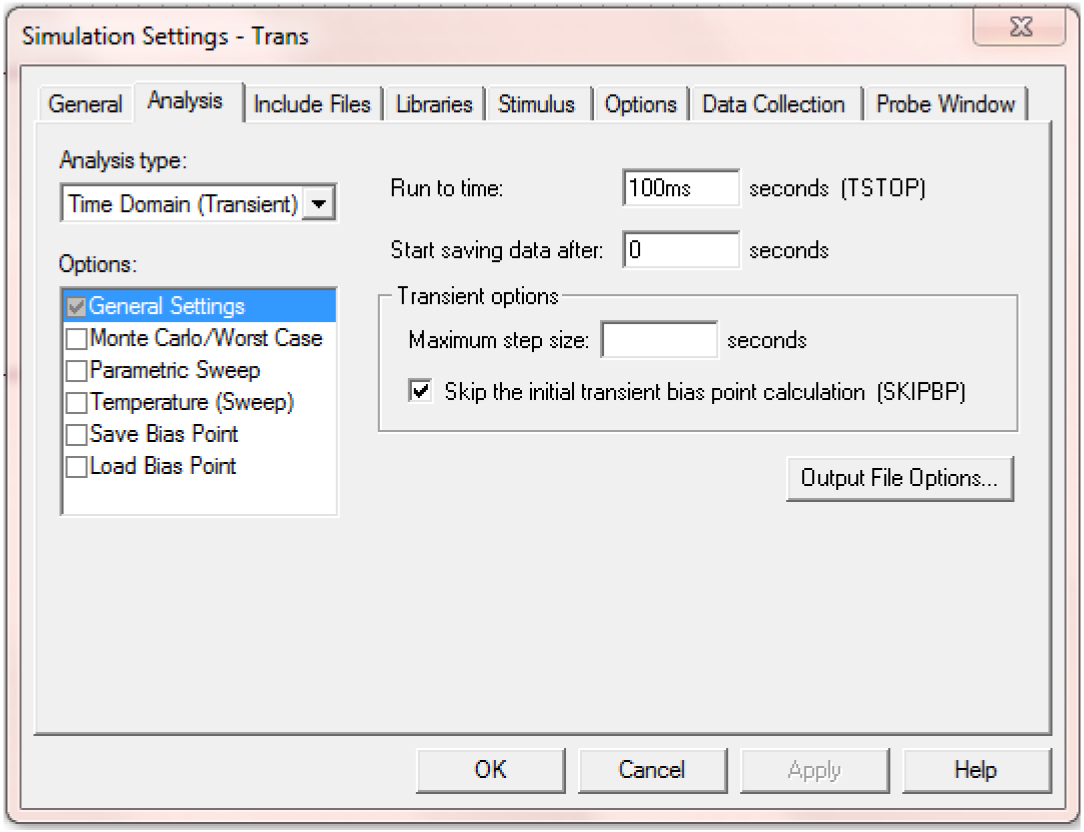
That should be O.K., however the value of the current will be negative, and the graph will be resized to show both positive and negative values. We prefer to see the current in the positive direction as we imagine the current will be flowing gin the circuit.
Also, the value of the current will be in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

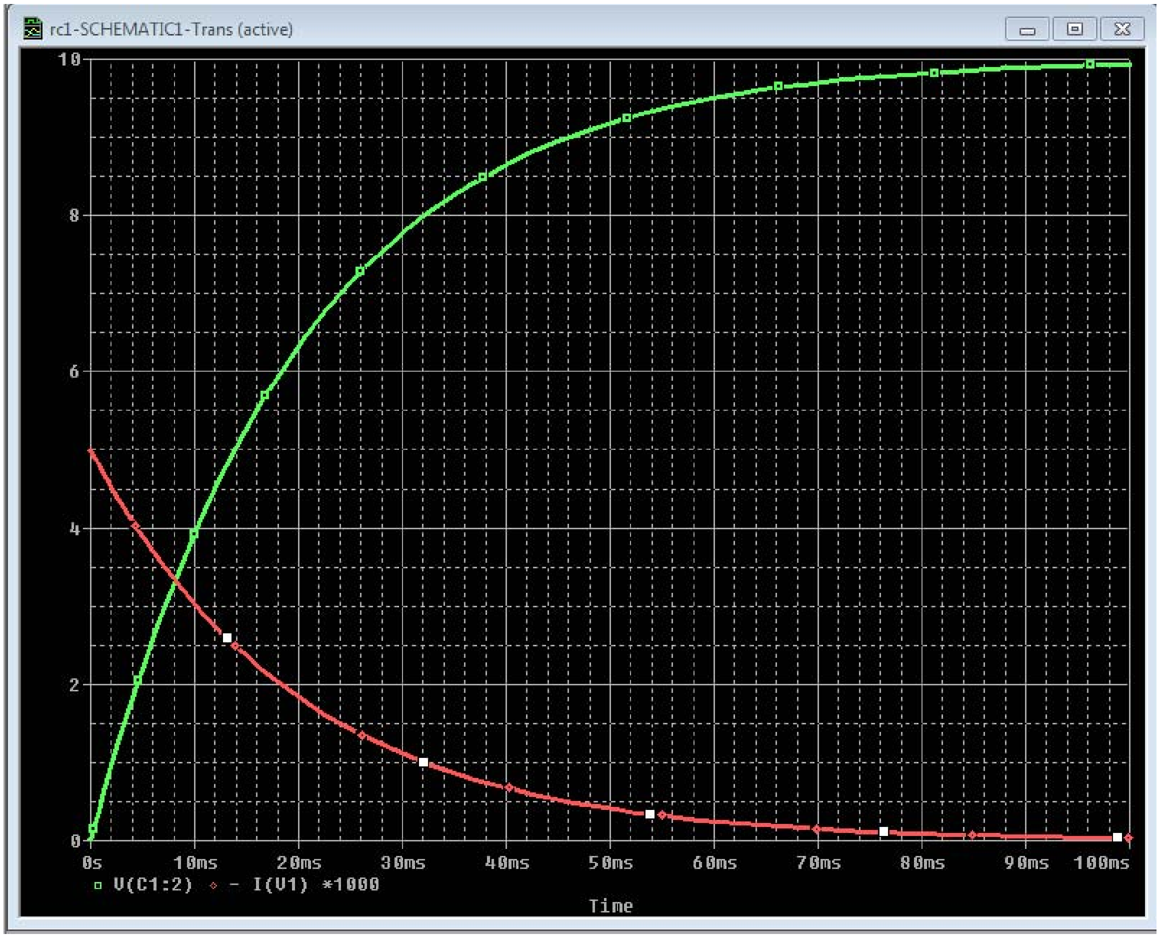
Now start a new project called RC1 to simulate the step response of a simple RC Cicuit. Draw the following circuit. Do not forget to cgane the name og the ground from GND to 0 or SPICE will not work. Save the file



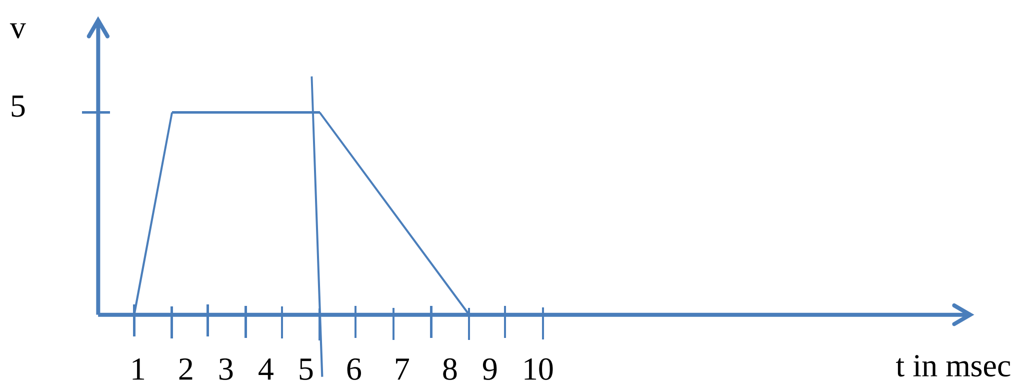
Now, we have to set the simulation profile.
Top menu PSpice → new simulation profile.
Call it tran, no inheritance from other profiles, then O.K.



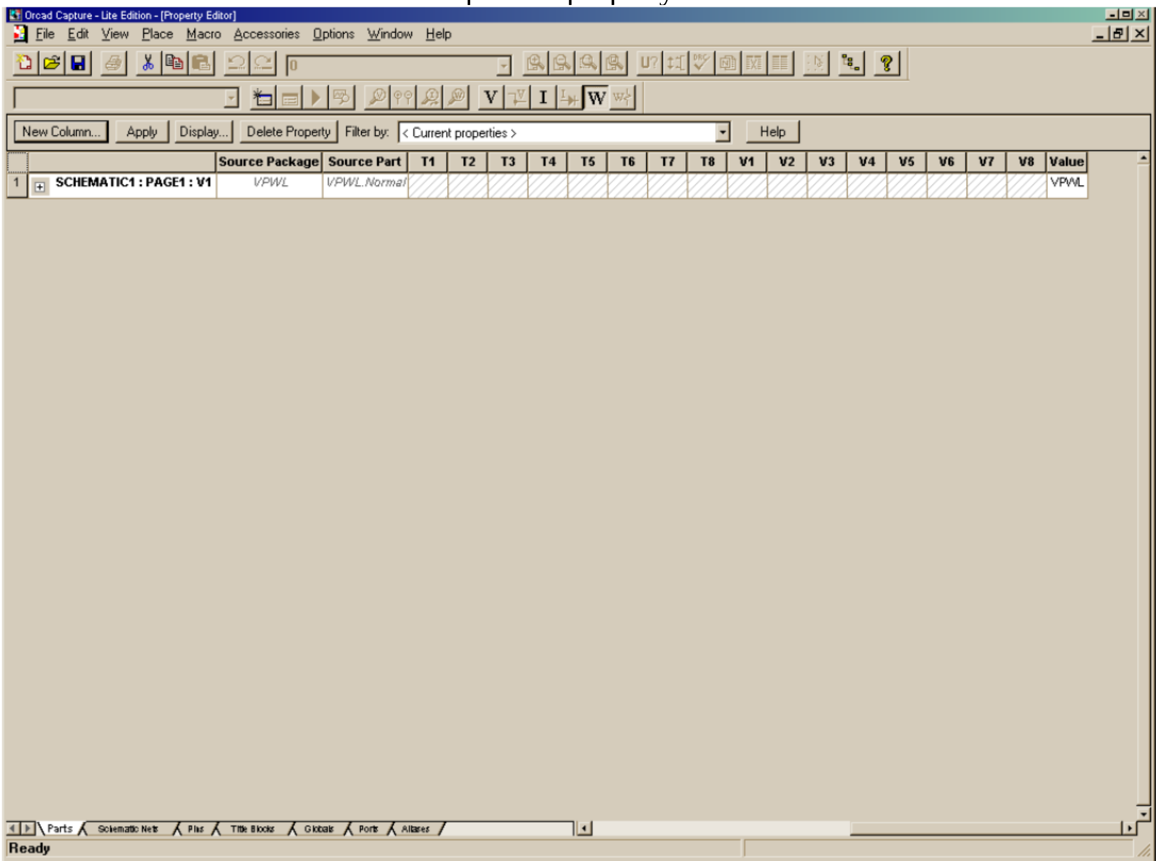
In the Analysis tab choose analysis type to be Time domain (transient), general setting Choose the run toi time 100 msec. (the time constant in the circuit is 20 ms, we will run it five times that). Also start saving data after set it to 0.
Click on “Skip the initial transient bias point calculation”. 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 isthe 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.



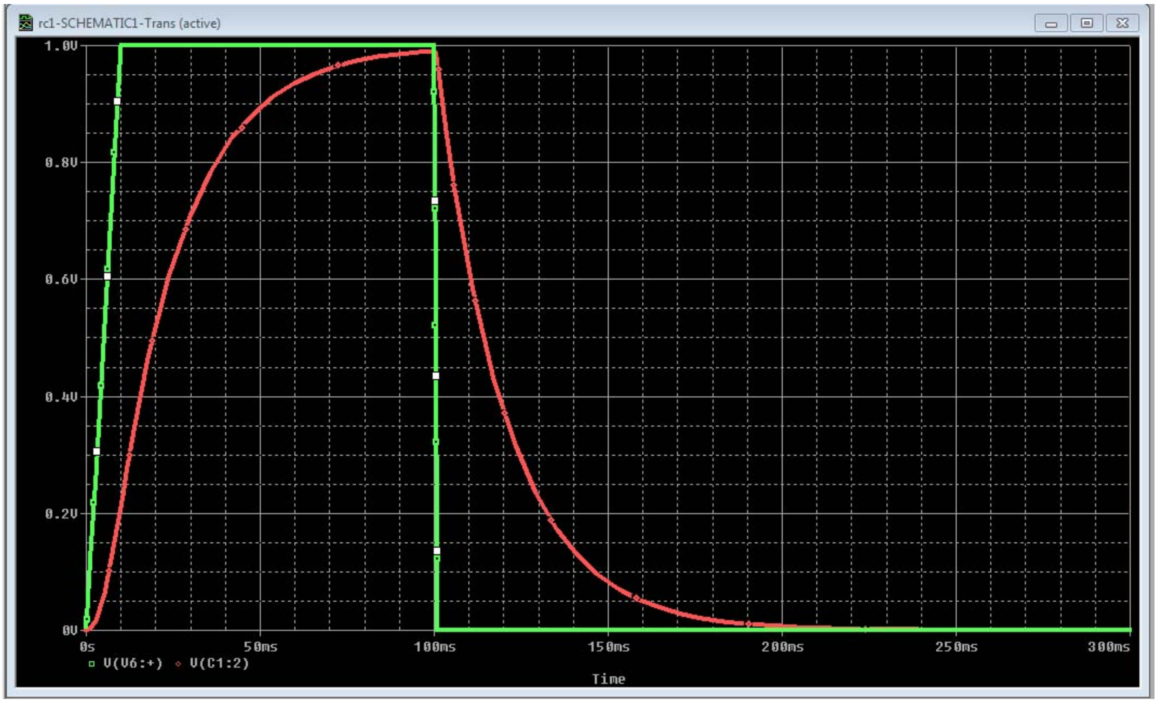
The above waveform is piece linear waveform. It stays at 0 V till 1 msec. Then it, goes linearly from 0 to 5 V for another millisecond. Stays at 5 V for 4 msec. then it goes linearly to 0 in 4 msec.
Replace the DC source in your circuit with a PWL source.
Highlight V1 in the schematic and delete it
Insert 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



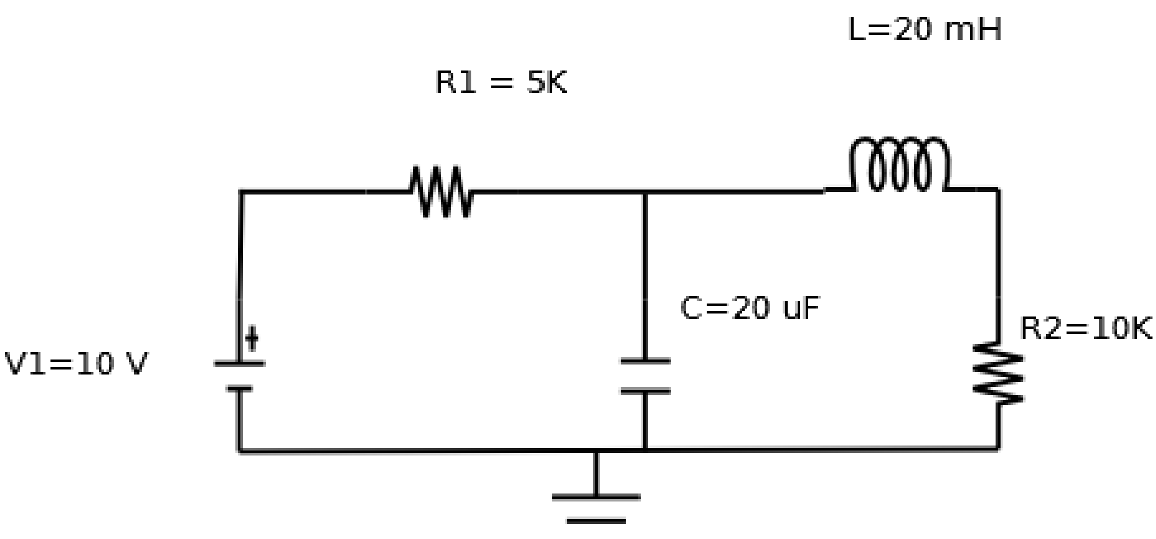
In the Parts tab (bottom part of the window) start inserting columns. Insert columns with the following names and values

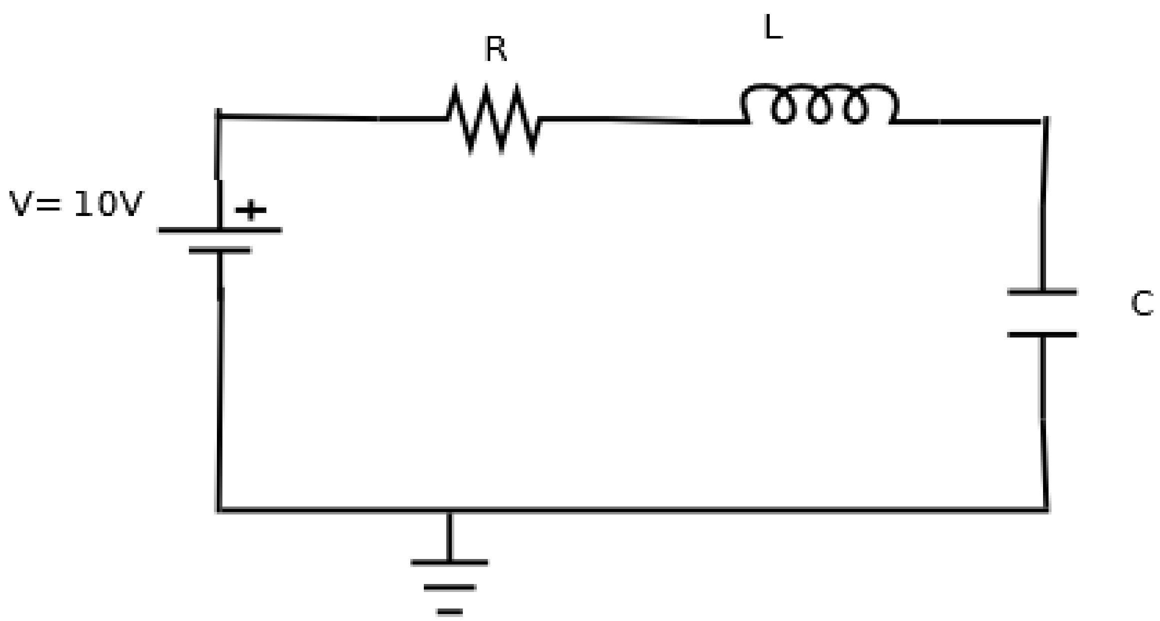
Name	T1	T2	T3	T4	T5	T6	T7	T8	T9	T10
Value	0	10	100	101	200	210	220	230	240	250
Name	V1	V2	V3	V4	V5	V6	V7	V8	V9	V10
Value	0	1	1	0	0	0	0	0	0	0

Apply and exit.
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[V6:+]. Note that V6 is the name of the PWL source in my simulation. The result should look like that



Exercise
Simulate the following circuit





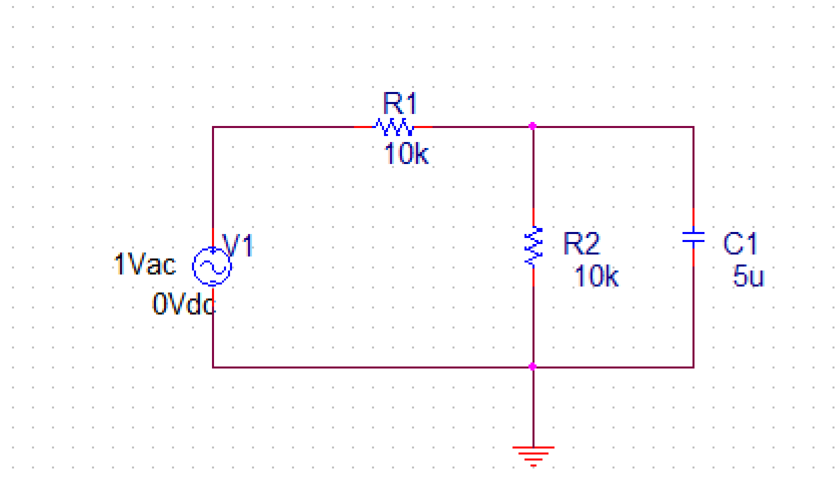
Simulate three cases as follows

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**.

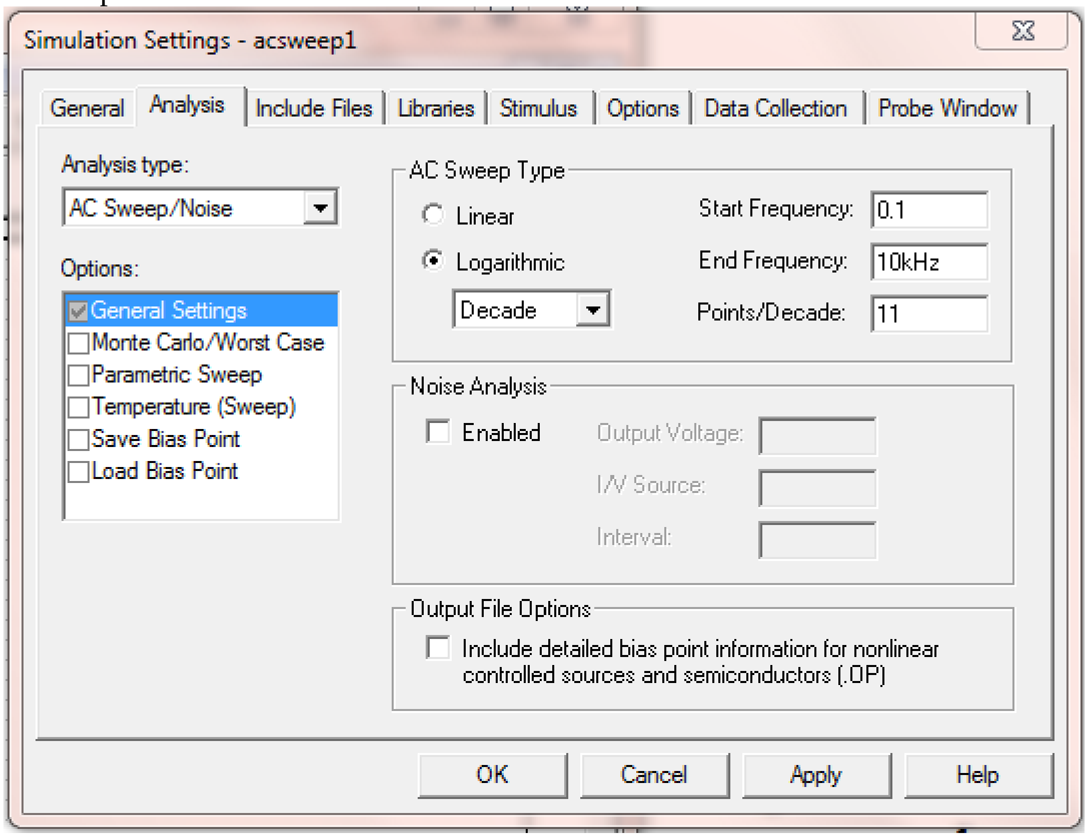
AC Analysis

In this section, we xxxxxx
Start a new project and construct the following circuit

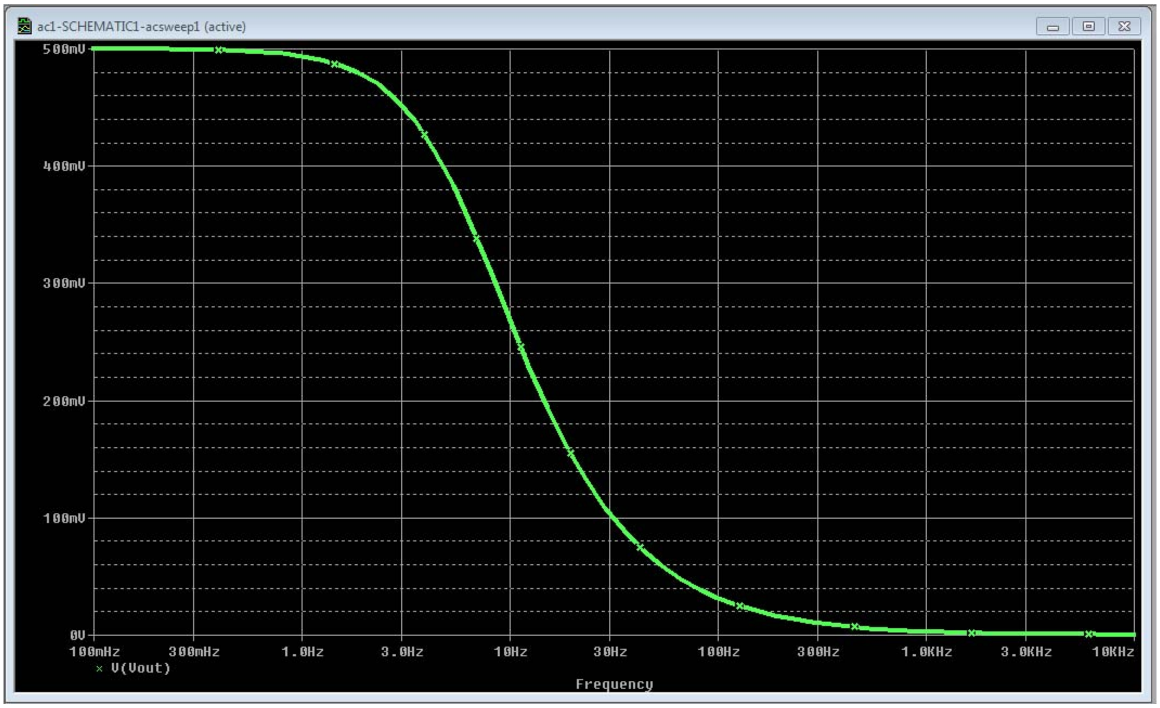


Choose the parameters as shown.

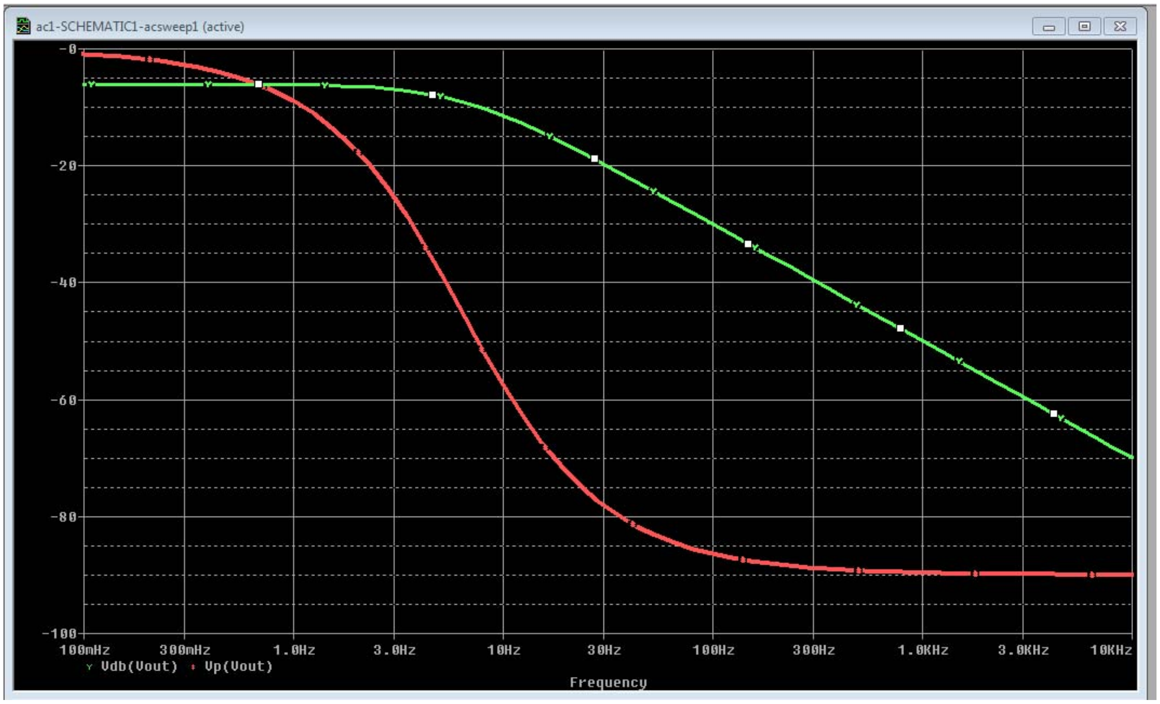
in the Vac source, double click on 1Vac and choose a value of 1 (there is no change here, just to know how to set it).
Save the file,
In capture, top menu PSice → New simulation profile, choose a suitable name, no inheritance.
Set the parameters as shown. Click OK

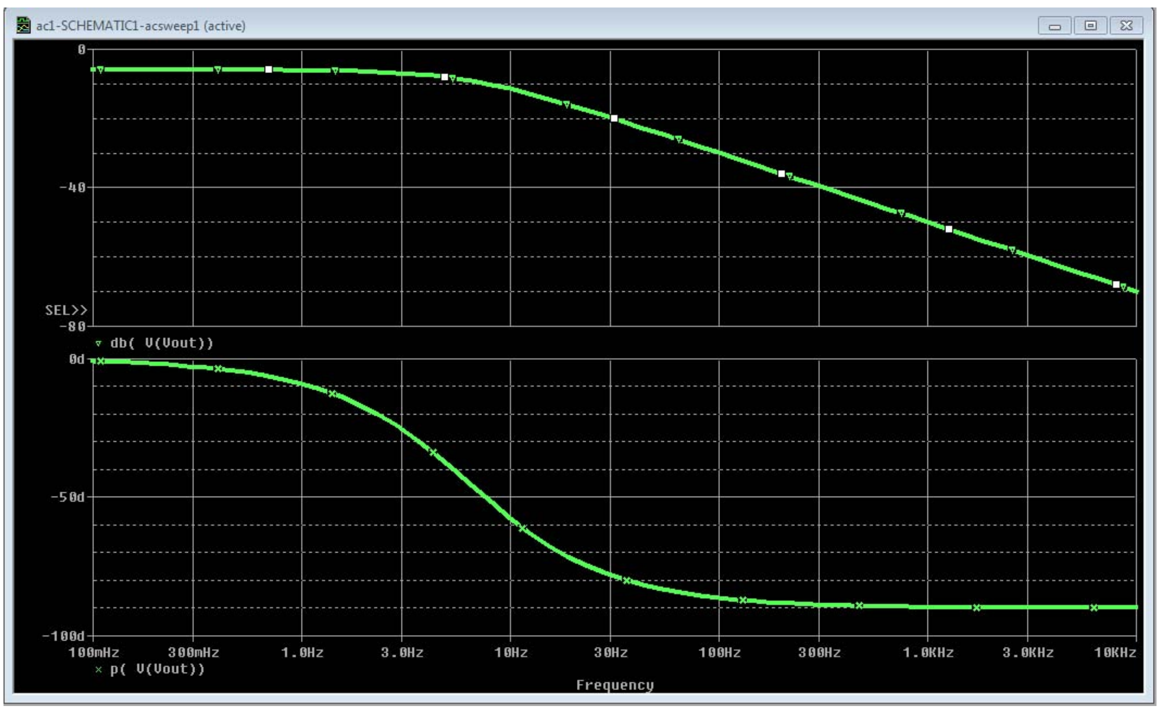


PSpice → Run, the probe widow appears.
Now, top menu Trace → add trace add V(Vout)



Now we can see Vout vs frequency.
For Bode plot, we would like to plot Vout in dB. In that case Trace → remove all traces. Then add a new trace. From the output variable choose Vdb(Vout) (choose V(out) then insert db after V).
Add another trace Vp(out) (again insert p after V in V(Vout). This is to show the phase of Vout.
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.
Also you can use Vdb(Vout) or db(V(Vout)) to plot Vout in dB. Also for the phase of Vout you can use Vp(Vout) or p(V(Vout)).





Active Circuits

Now you should be familiar with Orcad capture and PSpice. The next 2 examples illustrate how to simulate circuits that contains active elements (operational amplifiers and transistors).

Operational Amplifiers

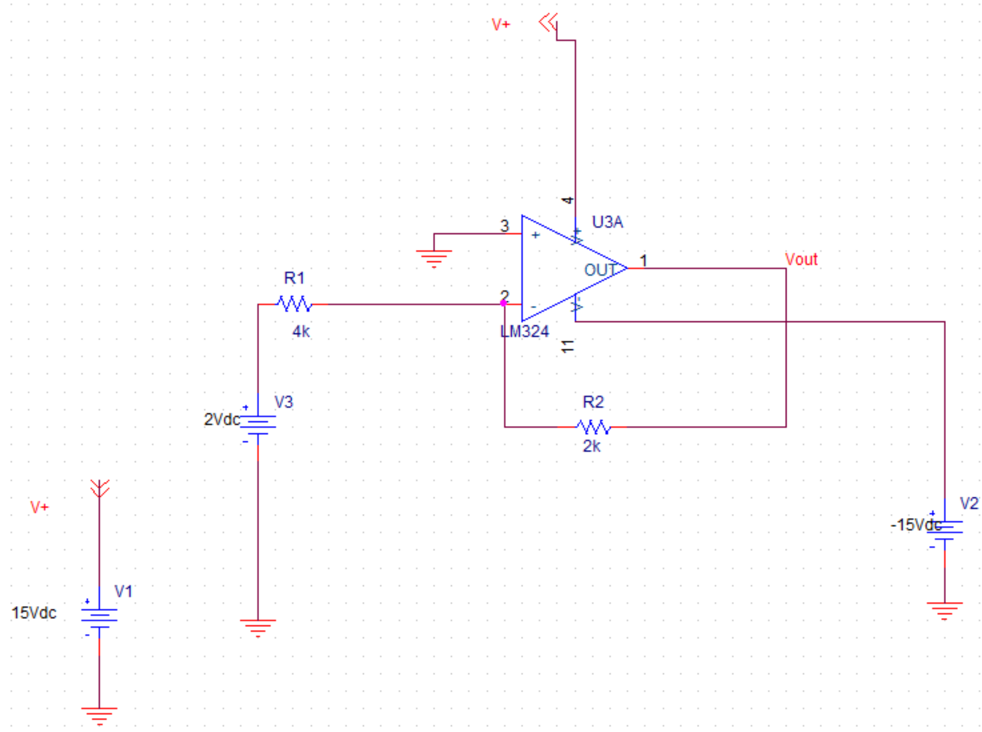
Construct the circuit shown below. Few new things in SPICE will be explained here. First, the operational amplifier LM324 is in the library EVAL. Insert it like any other circucit elements.

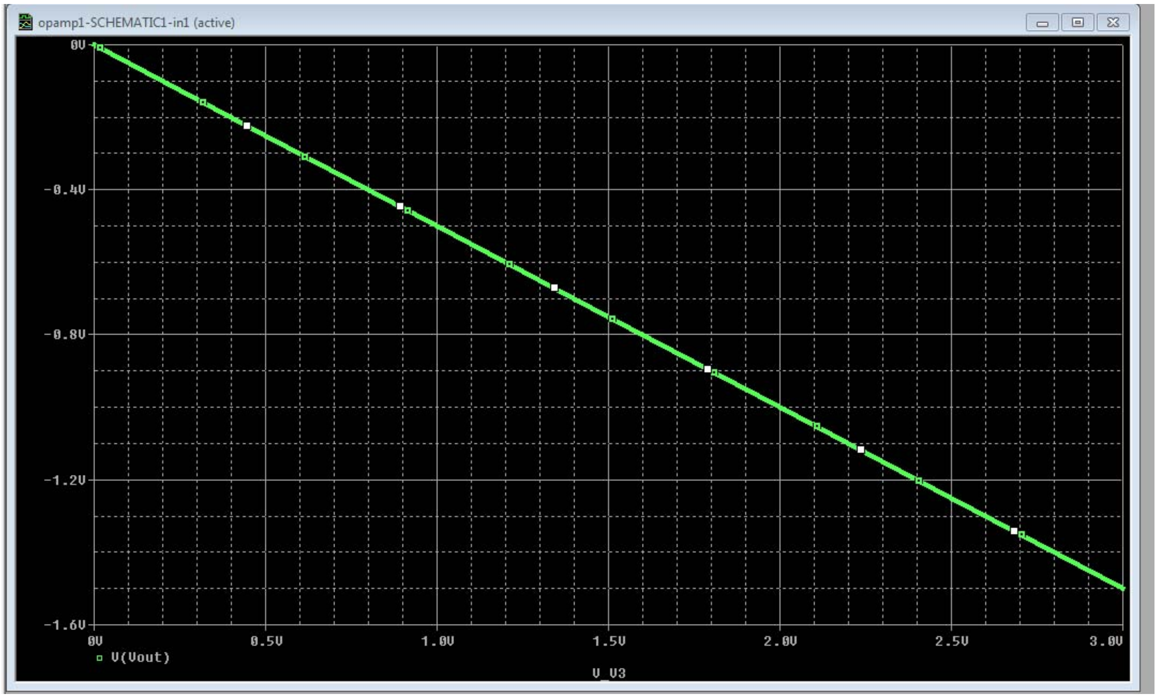
The LM324 has 5 terminals. Terminal 1 is the output, 2 and 3 are the negative and positive inputs. 4 and 11 are power inputs. 4 must be connected to the positive voltage supply, 11 should be connected to the negative power supply (+15, -15).

In the schematic, node 11 is connected to a Vdc of -15 V. However node 4 is connected in a slightly different way. This is usually done to decrease the clutter in the schematic due to wires crossing each other. Voltage source of 15V (V1) is constructed and connected to ground. Then we place a **net alias** use the top menu Place → Net alias (or use the side menu the button labeled N1, third button from top). We place a net alias connected to the positive terminal of the voltage source V1. Then we add another net alias (with the same name) and connect it to terminal 4 of the opamp. Note that any two net aliases with the same name are considered to be connected.

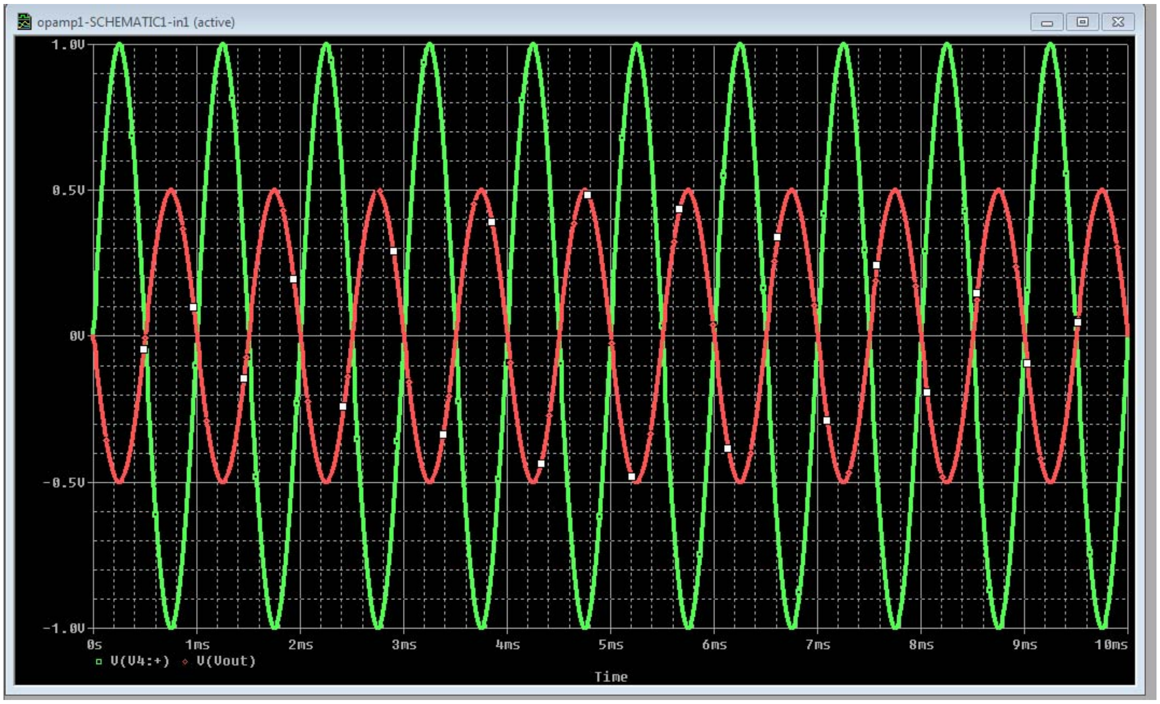
Make a simulation profile to simulate the bias point. What is the value of Vout?

Modify the simulation profile to make it DC sweep. Change the value of V3 from 0V to 3V in 0.5V increments and simulate your design, add trace for Vout, you will see the following graph

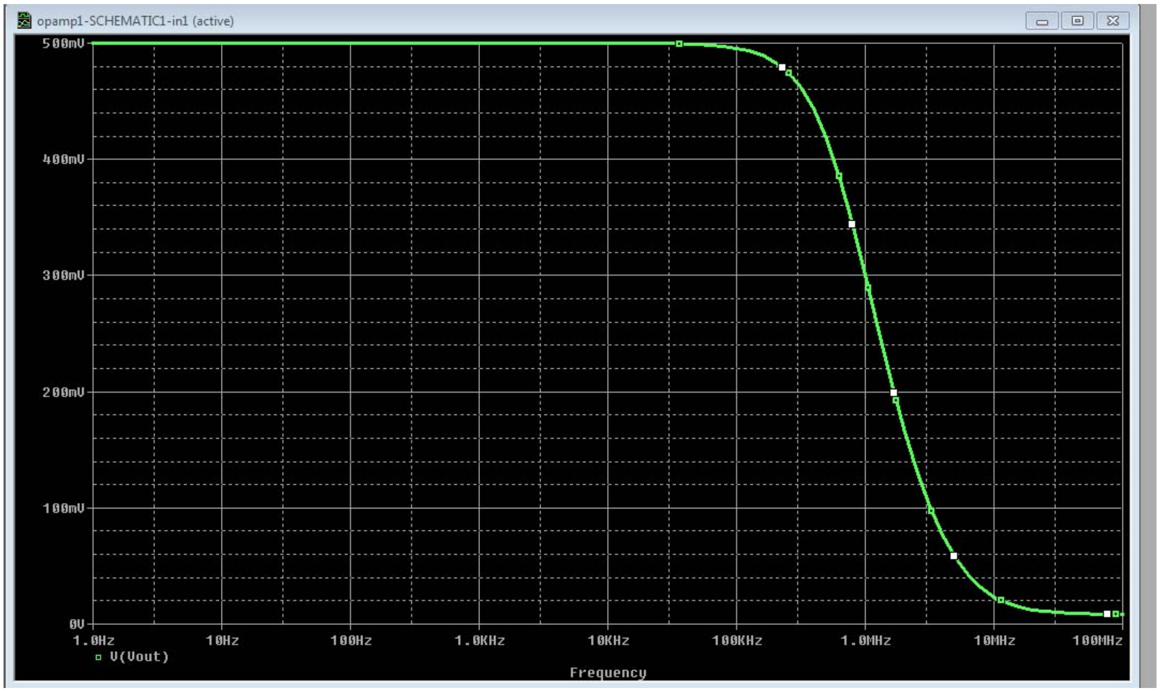




You can see clearly that Vout = -0.5Vin as expected for an inverting amplifier configuration.
Replace the V3 source with a Vsin (from the source library). Set the offset voltage to 0V, amplitude to 1V and frequency to 1 KHz.
Change the simulation profile to Transient. Do not check the box Skip the initial transient point calculation SKIPBP. Run and display the traces. You will find that the sine wave does not look nice, go back to edit simulation profile and set max step size to 0.01 ms (PSICE chooses it automatically). Now the output looks like the following. Again Vout = -0.5 Vin

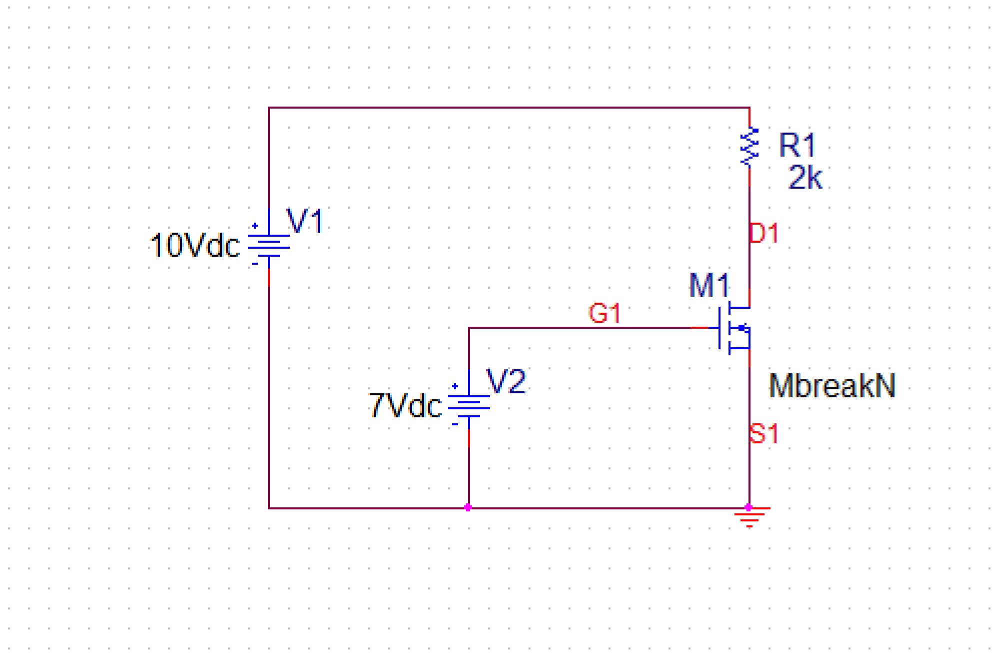


Now, change the Vsin source to Vac.
Edit the simulation profile to simulate AC sweep. Change the voltage from 1Hz to 100MHz run the simulation, view the result and add trace for Vout, the result is

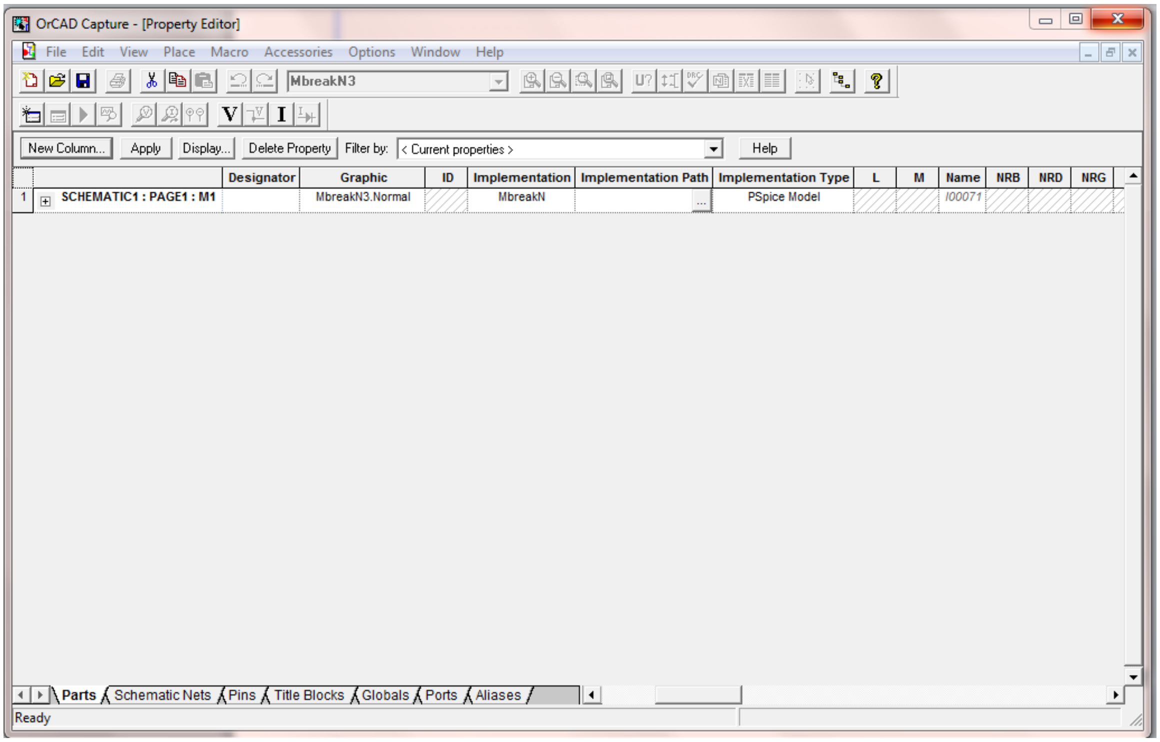


MOSFET

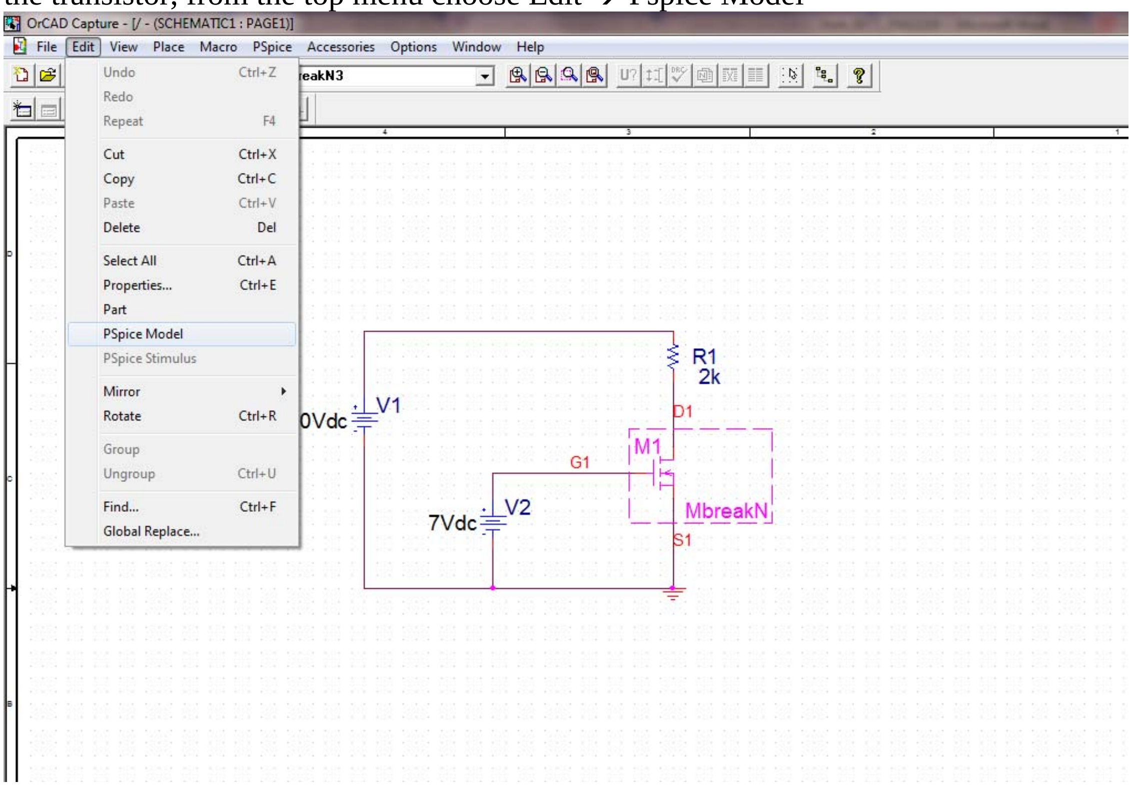
In this part, we will simulate a MOSFET transistor. Start a new project and call it simplenmos. Construct the circuit shown below.



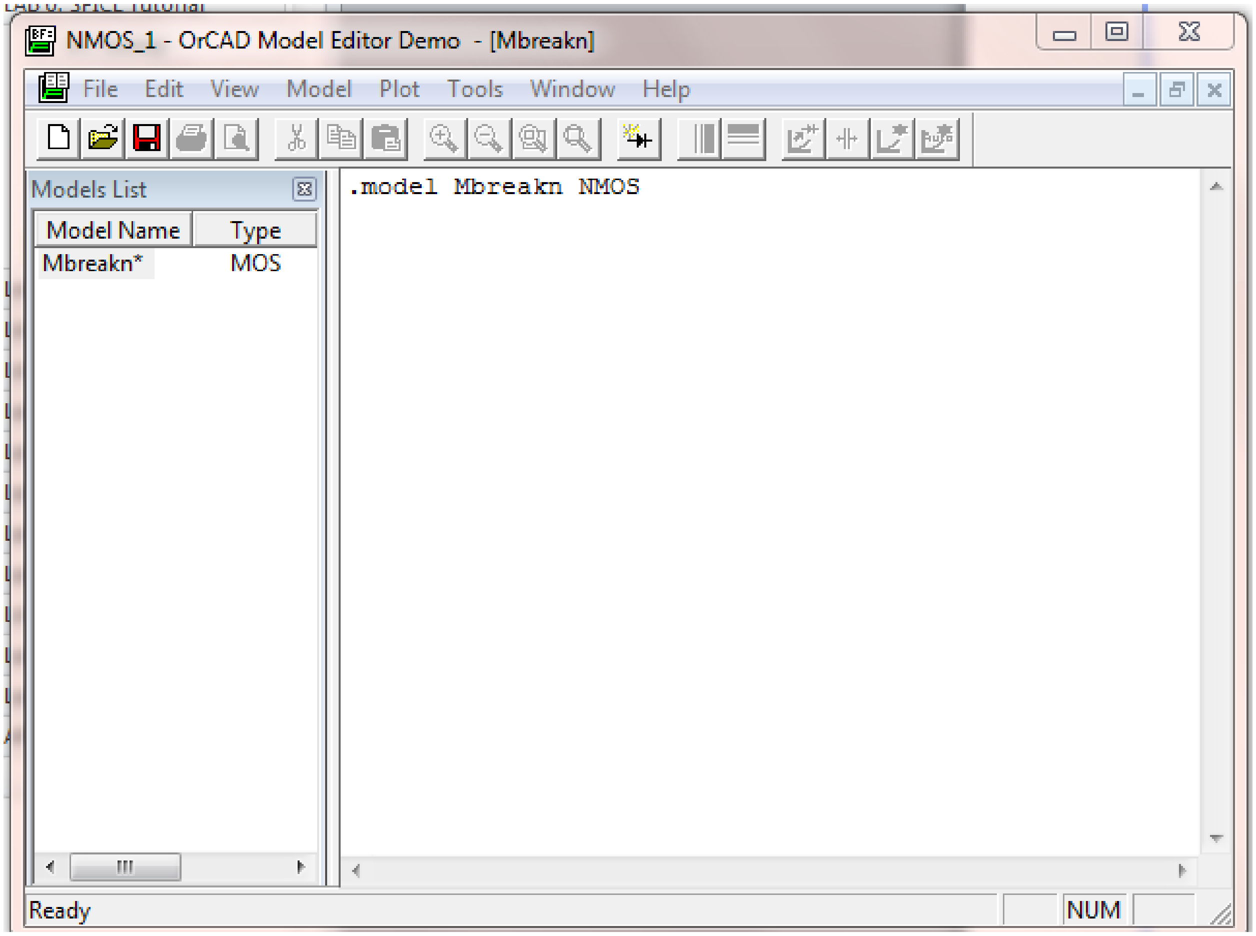
The only new thing here is the MbreakN. In Spice a device starts with M is a MOSFET transistor. From the Breakout library choose "MbreakN 3". Use net alias to name the three nodes of the MOSFET to G1 (gate) S1 (source) and D1 (drain).
Now, we will learn how to set the parameters of the NMOS transistor. Highlight the transistor, right click and choose Edit properties. The property editor appears.
Click on the scroll;ing bar and drag it to the right until you see the property L and W
Click in L and type 1e-6 (I µm length for length) and in W type 10e-6 (10 µm for width).
Click apply and close the property editor.
.



Now, we will edit the SPICE model of our transistor. Highlight the transistor. Highligh the transistor, from the top menu choose Edit → Pspice Model

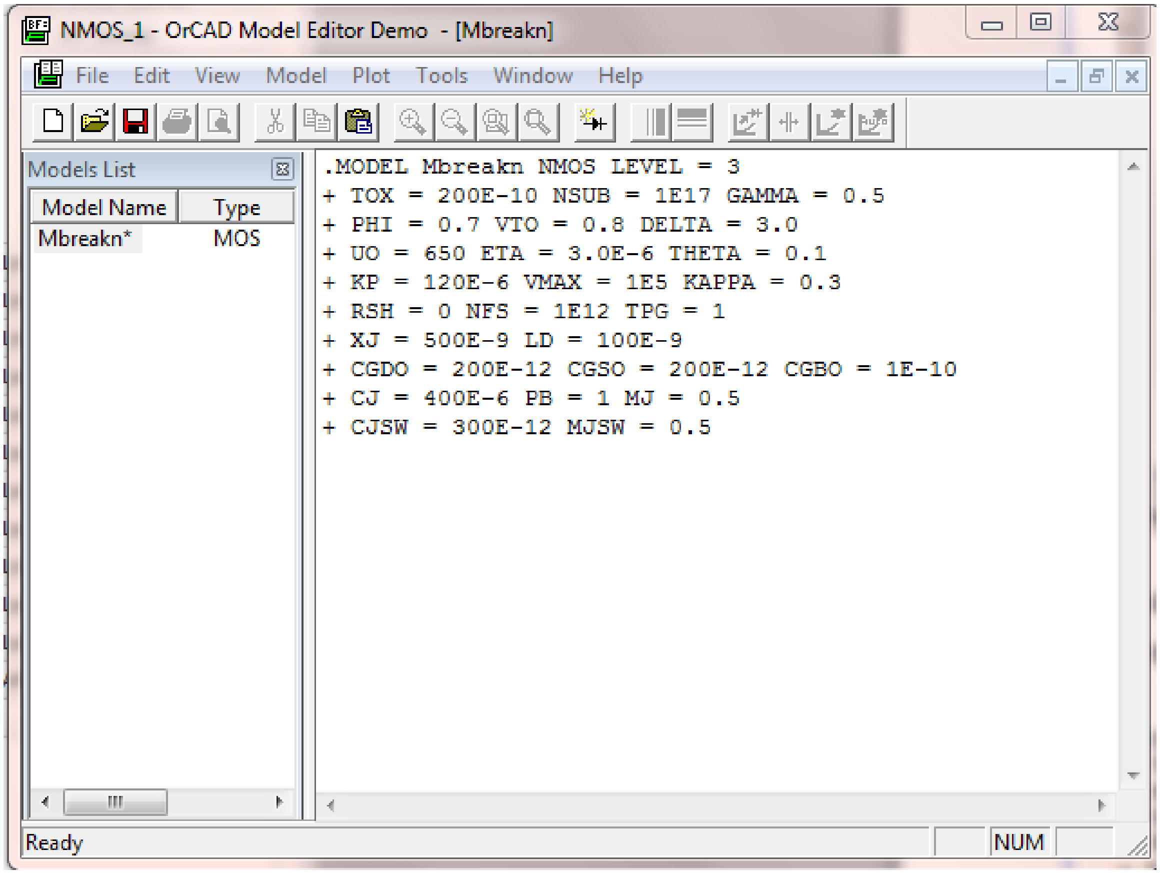


The model editor appears.

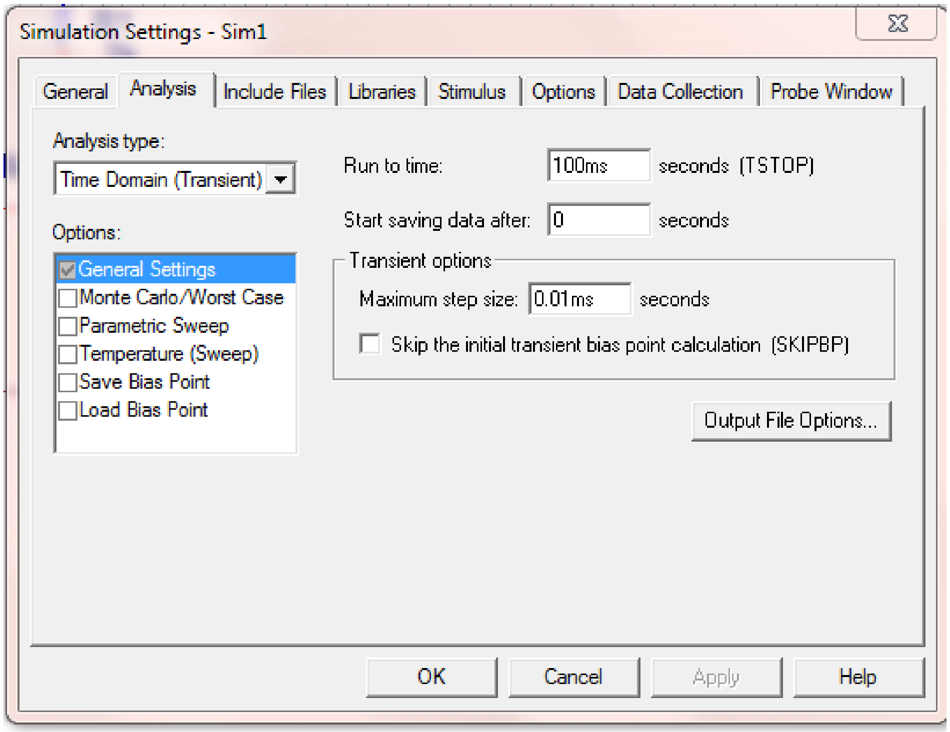


Delete the line .model Mbreakn NMOS
Copy and paste the following model.

.MODEL Mbreakn NMOS LEVEL = 3
+ TOX = 200E-10 NSUB = 1E17 GAMMA = 0.5
+ PHI = 0.7 VTO = 0.8 DELTA = 3.0
+ UO = 650 ETA = 3.0E-6 THETA = 0.1
+ KP = 120E-6 VMAX = 1E5 KAPPA = 0.3
+ RSH = 0 NFS = 1E12 TPG = 1
+ XJ = 500E-9 LD = 100E-9
+ CGDO = 200E-12 CGSO = 200E-12 CGBO = 1E-10
+ CJ = 400E-6 PB = 1 MJ = 0.5
+ CJSW = 300E-12 MJSW = 0.5
It should look like



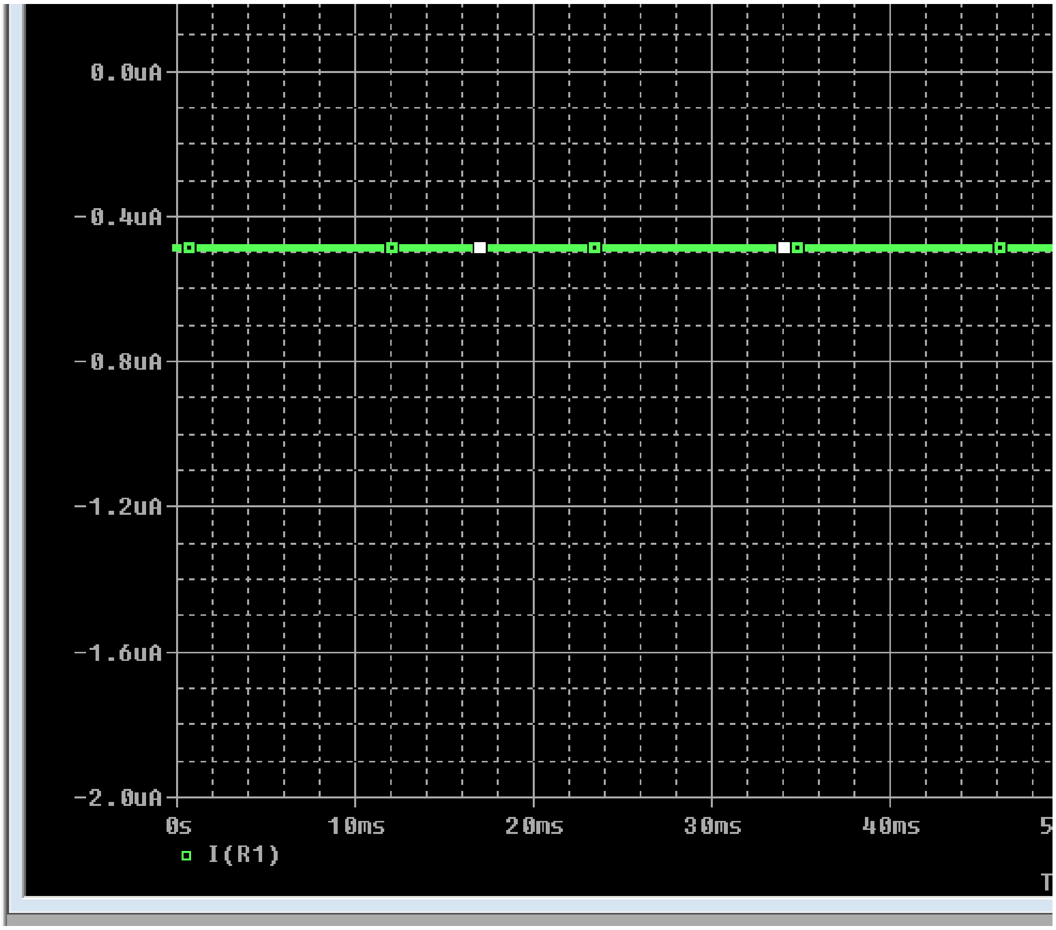
Top menu File →save, then close the model editor dialogue box.
Now the circuit is ready to be simulated.
In Schematic, top menu PSpice → create new simulation profile
Choose a suitable name and inherit from non
Set the Run to time to 100ms and the maximum step size to 0.01ms.



Apply and close the box.
From the top menu in schematic chose PSice → Run
The simulation result window appears.
You can go to view → output File to view the output file. Examine the file. You will find a complete description of your circuit is spice as follows

**** INCLUDING nmos_1-SCHEMATIC1.net ****
* source NMOS_1
R_R1 D1 N00024 2k
V_V1 N00024 0 10Vdc
V_V2 G1 0 7Vdc
M_M1 D1 G1 0 0 Mbreakn
+ L=1e-6
+ M=10e-6

Close the output file window; go to View → Simulation results to get the simulation result window.
Add trace and choose I(R1), a plot of the current appears.

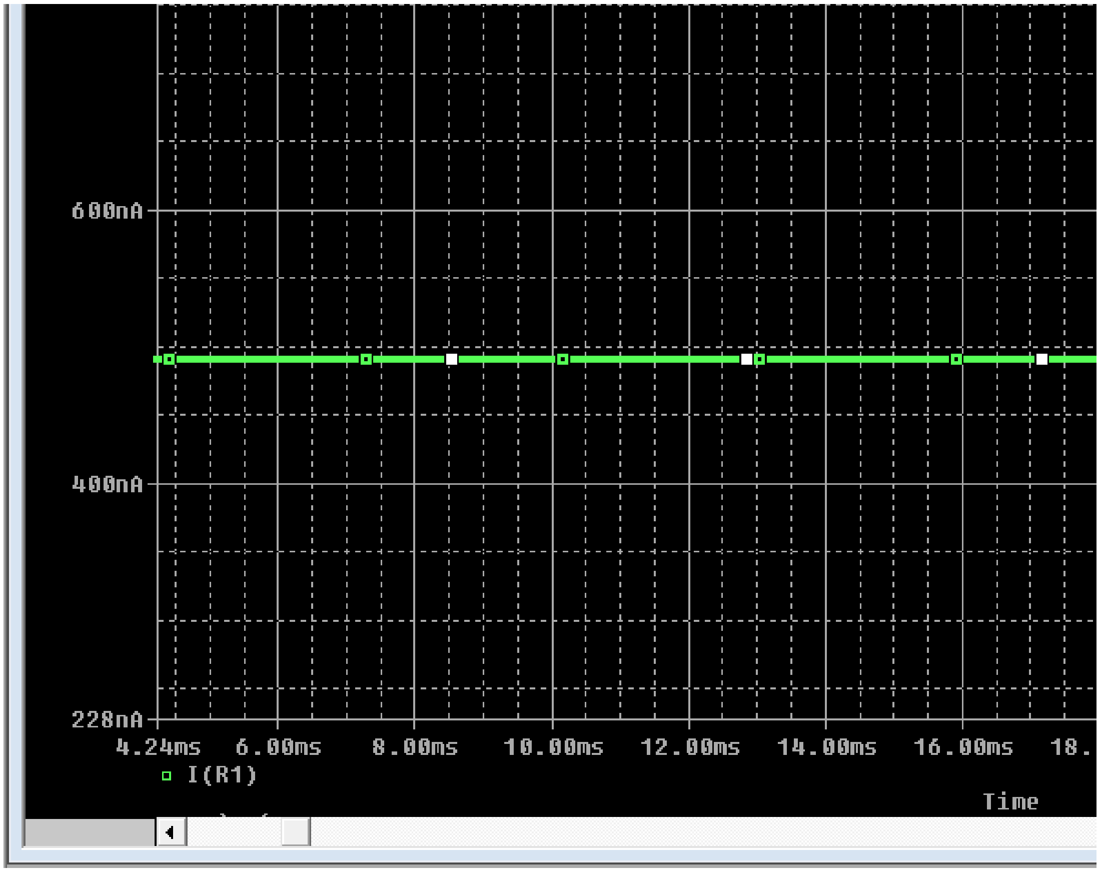


You will notice that the current is negative (-0.5 µA). The current in the Rsistor must be positive. Why the current is shown as negative?

Keep in mind that the current in the resistor R1 I(R1) is the current entering the first terminal. When you inserted the resistor it was horizontal, to place it in a vertical position you highlighted it and hit “r” or right click rotate. That rotated it counterclockwise and terminal 1 is the bottom terminal of the resistor. Thus the current flowing into the transistor is going from terminal 2 to terminal 1 of the resistor, which makes it negative.

To correct this, highlight the transistor, hit “r” twice and terminal 1 is the upper terminal of R1.

Run the simulator, add trace I(R1) and it looks like the following (current is positive).



Adding a library

The student Capture comes with few libraries added already. if you want to add another library, first you have to locate it. In the example here, I will describe how to add ON_bjt library (a library for BJT transistors).
Go to http://www.cadence.com/products/orcad/pages/downloads.aspx you will find a huge number of libraries from different manufacturers. Searc for ON Semiconductor, and then for on_bjt. You will find two files on_bjt.lib (the model library) and on_bjt.olb (the symbol library).
Download these two files, store them with the rest of libraries of Capture (on my machine C:\PSP\Capture\Library\Pspice).
Then go to capture, add a part. When you get the choose part windoe, choose "Add Library". Navigate to the directory where the library is and open the library on_bjt.olb.
You still need to do this in your schematic design.
In capture, go to Pspice→ Edit Simulation Profile (or New Simulation Profile if you do not have any).
In the Simulation Setting Window, click on the Libraries tab. browse to the new library location and choose on_bjt.lib (you can add it as global, or local to this profile only).

LAB 0_B: PSPICE and Capture

OBJECTIVE

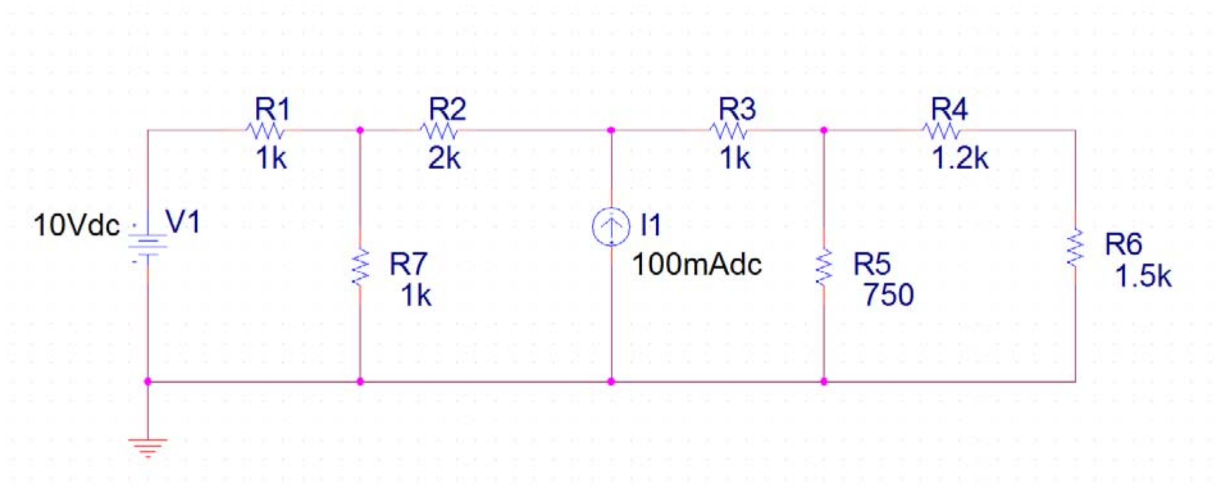
The objective of this lab is to get familiar with Orcad Capture and PSPICE.

In this lab, you will be asked to simulate a number of circuits. For each simulation you will submit the following

- Schematic diagram from Capture with the circuit
- SPICE code
- Required graphs
- Data tables if applicable

Part 1

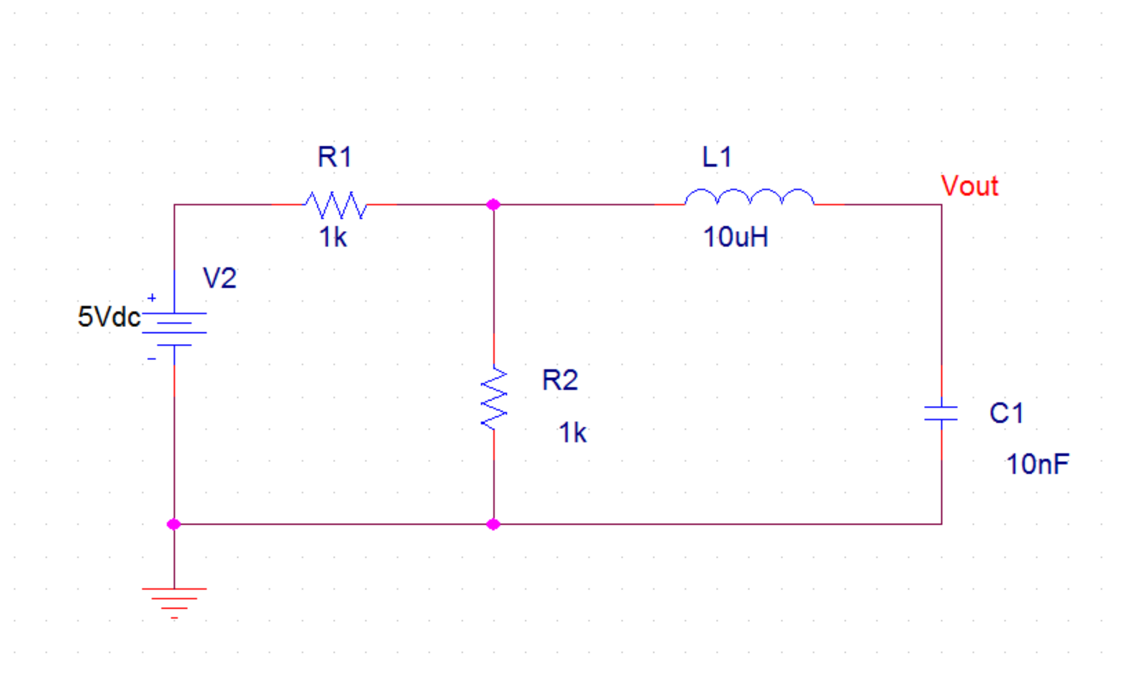
For the following circuit



Using Capture and PSpice construct and simulate the circuit shown above.
Get the currents in every branch and voltages of every node
Draw a plot showing the power dissipated in the R1 resistor as V1 changes from 2 to 20 V
Construct a Thevenin equivalent circuit for the above circuit as seen by R6. Note that R6 should not be part of the Thevenin equivalent circuit. basically what you should do is construct (modify) the above circuit to calculate Thevenin voltage source and Thevenin resistance.
Hint get the open circuit voltage and short circuit current, or the open circuit voltage and the resistance seen by R6 after removing sources, you can do this by replacing R6 by a current source with 1A and calculating the voltage across it. Also note that SPICE cannot deal well with hanging nodes, so instead of open circuit, add a resistance that is much bigger than any resistor in the circuit.

Part 2

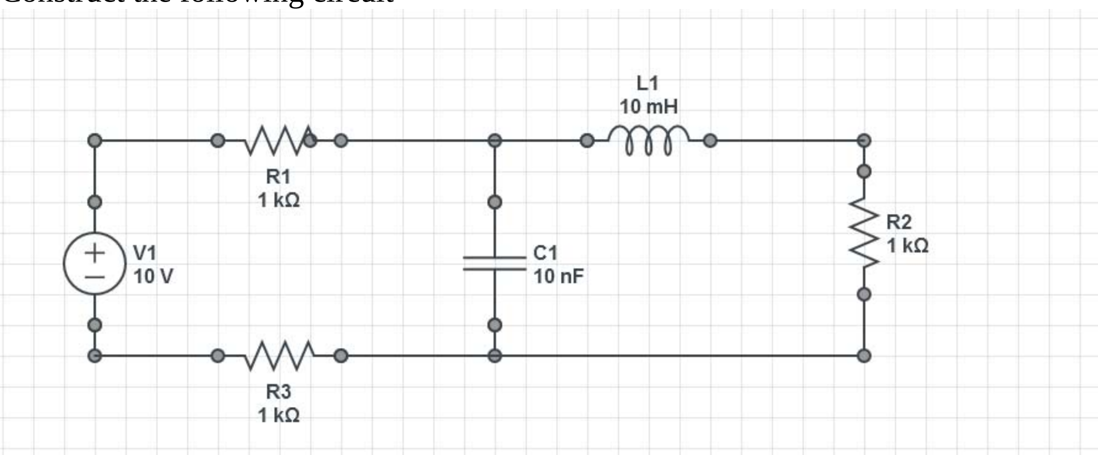
For this part, simulate the following circuit using Capture and PSpice



Plot Vout vs. time Choose the time scale to show the transient response of the circuit. For the values shown, you should not get overshoots. (not underdamped).
Change the value of the capacitor to get a reasonable underdamping. What is the value you choose? Plot Vout vs. time to show the step response.
How long did it take Vout to reach 90% of its steady state value?

Part 3

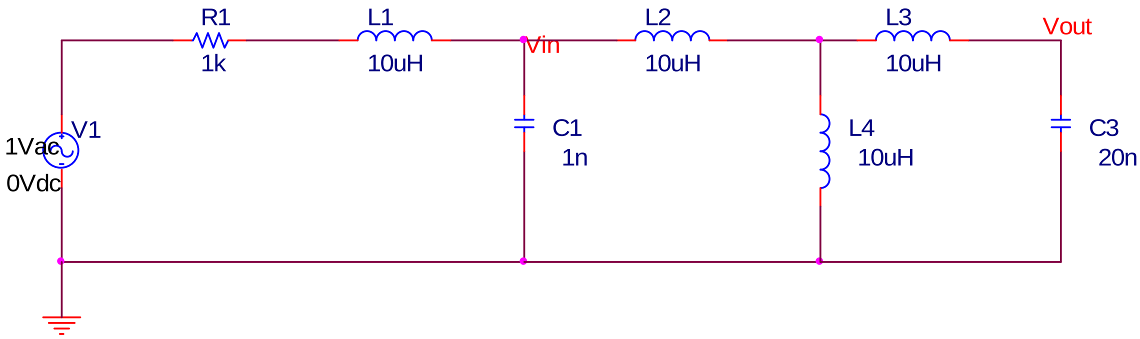
Construct the following circuit



Simulate the circuit displaying the current in the resistor R2 as a function of time. Choose the time to show the complete behaviour of the current in R2.

Part 4

Construct the following circuit



Simulate this circuit showing V_{out} and V_{in} as a function of frequency (consider the range from 10Hz to 50MHz).
Repeat the above step showing V_{out}/V_{in} in dB instead of the absolute value

LAB 1: Operational Amplifiers

Get the datasheet for LM741CNNC/NOPB-ND (the operational amplifier you will use in the lab). You can get it from www.tigerdirect.ca

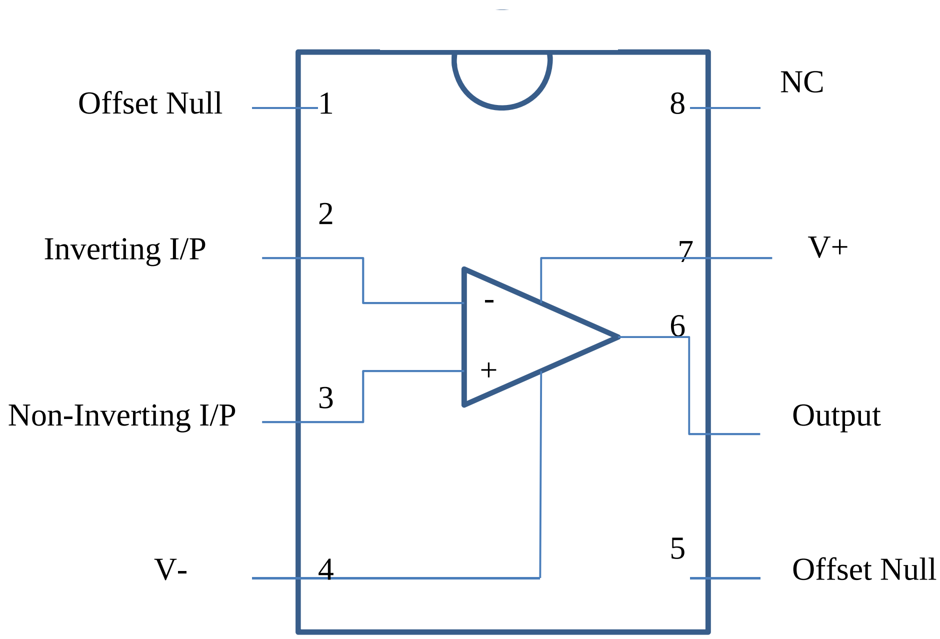
Objective:

- The objective of this lab is to study the operational amplifiers and its use in basic configuration (inverting amplifier, non-inverting amplifier …)
- To be able to simulate opamp circuits using SPICE.
- Understanding how the operational amplifier can be used as an integrator and differentiator and constructing an integrating circuit and testing it.
- Understanding the frequency response of the op-amp and its effect on gain
- Constructing circuit to illustrate the slew rate of the amplifier and measuring it.

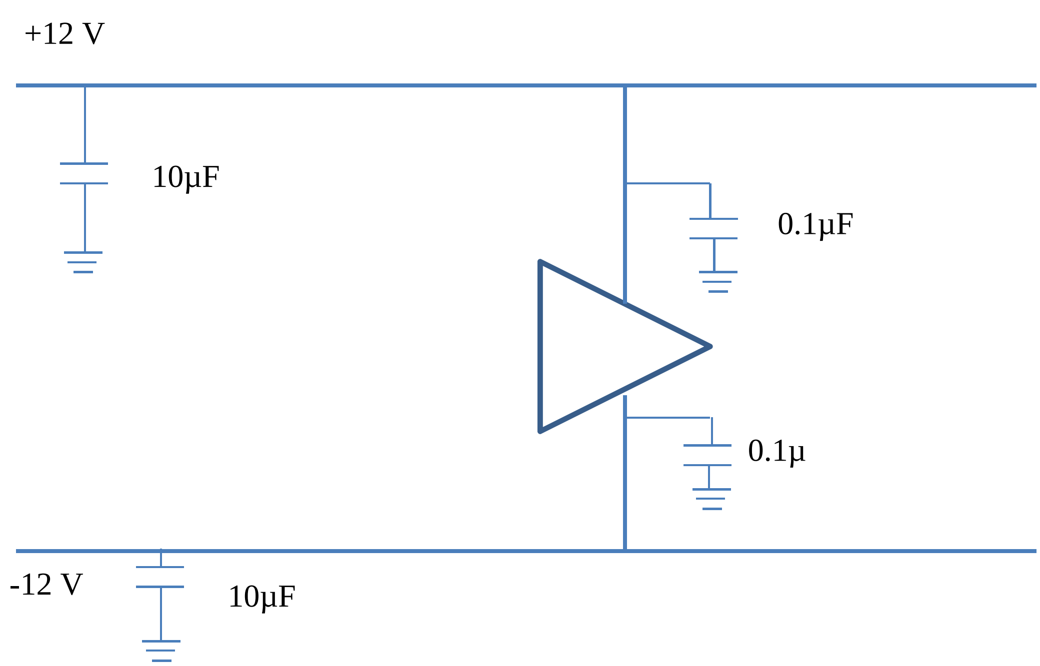
Introduction:

Operational amplifiers are very useful building blocks in many electronic circuits. The operational amplifier is an amplifier with a very high (ideally infinite) open loop gain, very high input impedance (ideally infinite) and very low output impedance (ideally zero). The operational amplifier is available as an IC as shown below,

The Chip pins are as shown below



Since this is your first opamp circuit, some hints are in order here to produce a good design and avoid common mistakes.
Grounding: Keep all grounding wires short, the longer the wire the longer the resistance and the more deviation from the ground.
It is a good idea to use bypassing capacitors with power supplies. For opamps we usually add a large bypass capacitor at the point of entry of the DC power to the board (10µF). If you have more than one opamp you can use a small capacitor (0.1 µF) near the power supply pins as shown in the Figure. The capacitor will form a low impedance path to the ground for high frequency noise on the power line.



As you can see from the above Figure, pin 8 is not connected, pins 7 and 4 are for power, 2 and 3 for input and 6 is for output. Pins 1 and 5 are for offset nulling, we will use this later. For now consider them to be not connected.

Prelab

1. Simulate the circuit shown in Fig 2_1 using PSpice. Find the output voltage and the current in each resistor.

Compare the simulation results with the solution using a simple opamp model.
Submit the schematic, code and results

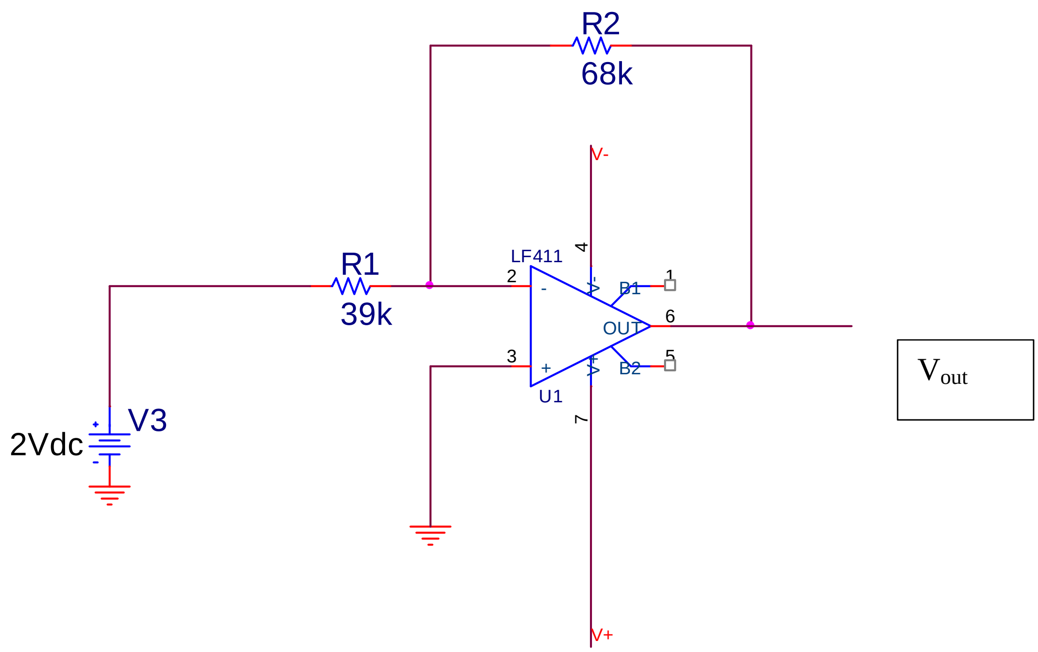


Figure 2_1

2. In this part, you will simulate an *integrator*. The integrator circuit is shown in Figure 2_2 below. As we mentioned in the class, the integrator does not need R2, we have to add it for practical consideration.

For the input, consider a Vpulse from the source library.the source parameters are set as follows:

TD	Time delay till the first pulse = 0
TR	Rise time, 1 µsec
TF	Fall time 1 µsec
V1	first level = 0V
V2	Second level =2V
PW	Pulse width = 1msec
PER	period = 5msec.

a. Simulate the circuit Fig. 2_2 using Capture and PSPICE after removing R2.
b. Repeat the simulation with R2 = 1MΩ.
c. Repeat the simulation with R2=10KΩ and R1 = 5KΩ

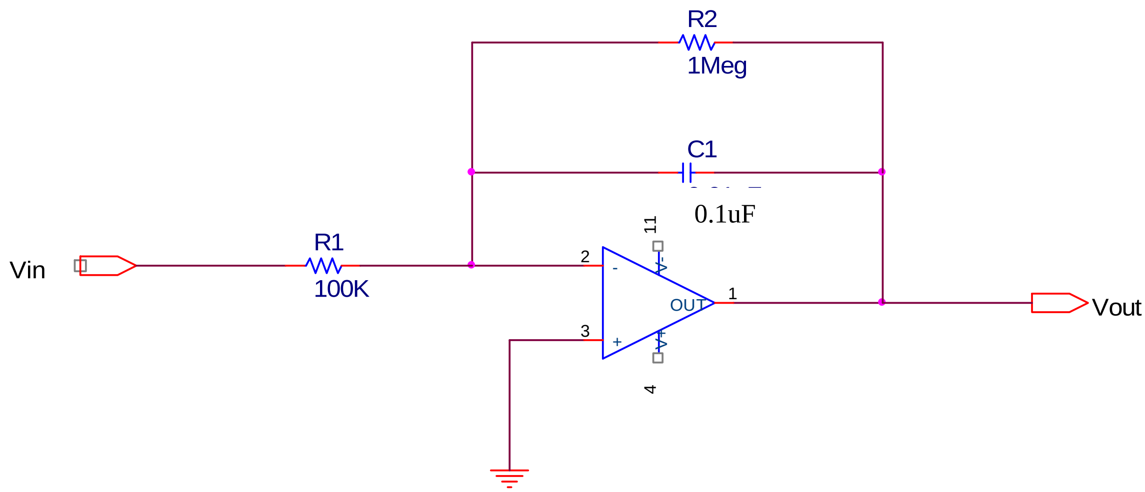


Figure: 2_2

In your report, show Vin and Vout for the 3 cases above.
Discuss in your report how is that circuit works as integrator and what are the differences between the three cases above?

LAB

PART I: Inverting opamp

Assemble the following circuit (Fig 2_3) using the LM741 on a breadboard. Note that the power connections are not shown. Make V+ to be +15V and V- to be -15V.

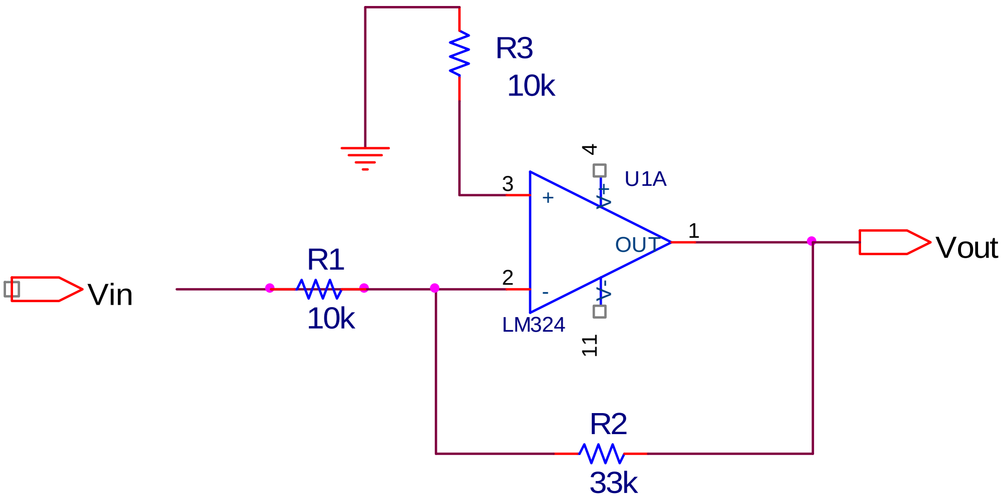


Figure 2_3

After finishing the layout, get the TA to check the connection.
Apply a sinusoidal input signal of 0.5 V (p-p) with a frequency of 1 KHz.
Show both the input and output signals on the oscilloscope.
Measure the amplitude of the input channel and output channel. How close the gain is to the calculated results assuming an ideal opamp?
Draw the 2 waveforms in your report.
Insert DMM to measure the input current into the circuit (out of the function generator).
Measure the Vin (either by using the scope or the DMM)
Calculate the input resistance of the amplifier circuit. How close the measured results are to the theoretical result?

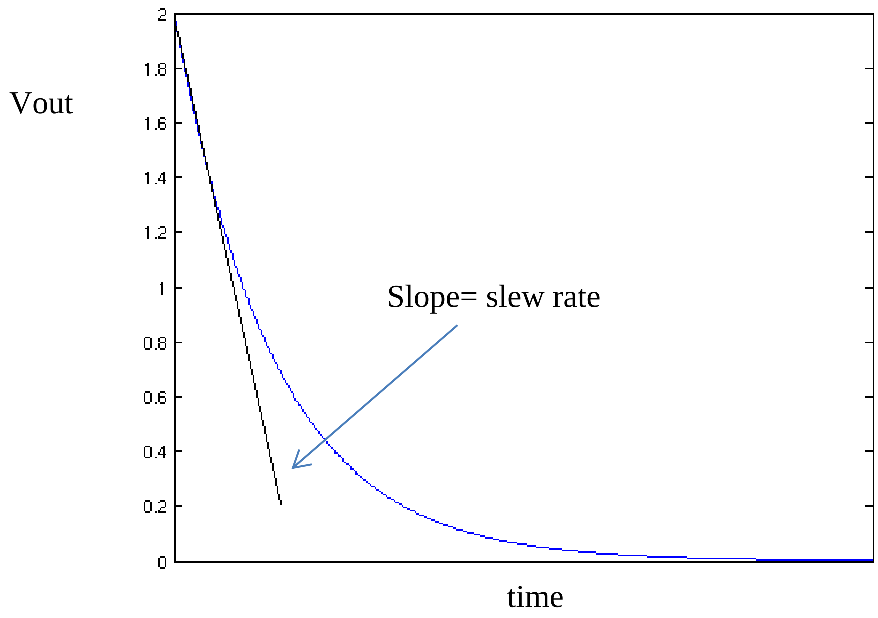
PART II: Integrator

In this part, we will use the operational amplifier to perform integration.
Construct the circuit you used in prelab (Fig 2_2).

Connect Vin to the signal generator output
Adjust the signal generator output to be a 500 Hz sine wave with 1V p-p
Display both input and output on the scope
Sketch (or print) the input and output (you can sketch it in your notebook and print it for the report).
Change the input to a square wave with the same frequency
Display the input and output on the scope and sketch them
Sketch (or print) the input and output
Is it what you expected? Explain
Change the input to a triangular waveform
Sketch (or print) the input and output
Is it what you expected?

PART III: Slew rate

Remember from the class notes and the text book that the output of an operational amplifier cannot change with a rate higher than the slew rate. In this experiment we will illustrate the concept of slew rate and measure it.
The slew rate, as mentioned above, is the maximum rate the amplifier can change its output. It is measured in V/µsec.
Our goal is to design a circuit and test the limit of its output change. One way is to construct an amplifier (inverting or non-inverting), or a voltage follower. In this part we will use an inverting amplifier. Design an inverting amplifier with a gain of about 2. The resistors available to you are (100K, 33K, 12K, 6.8K).
Start with a square wave (1 p-p V) with frequency of 5 KHz.
Display the input and output signal on the scope.
Through a combination of increasing frequency and scaling the oscilloscope output till the output appears as a decaying exponential as shown in the figure below, calculate the slew rate and compare it with the nominal value in the data sheet



PART IV: Gain-Bandwidth product

Review the concept of gain-bandwidth product.
Construct an inverting amplifier with a gain of about 2. We will measure the output voltage and compare it with the input for different frequencies.
Keep in mind that we do not want the slew rate effect. The output voltage must be small to avoid that (the slew rate is measured in V/μsec. The greater the V, the greater the slew rate effect).
You have an idea about the approximate GBP from the data sheet. Take measurements and construct a table showing Gain, B.W. and GBP.
How close it is to the nominal value?

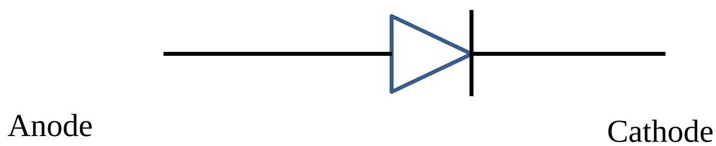
Lab 2: Diode

OBJECTIVE:

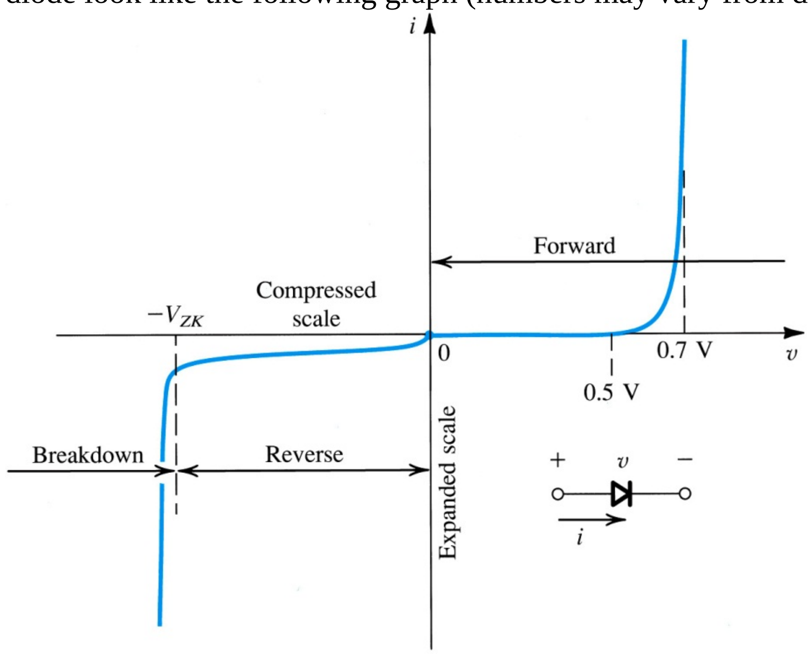
- Understand the basic operation and characteristics of the diodes.
- Using the scope to display the i-v characteristics of the diodes
- Understanding basic circuits that contain diodes
- Understand the half and full wave rectifier
- Designing, implementing, and testing simple circuits with diodes

INTRODUCTION

Diodes are 2-terminal devices that act differently depending on the polarity of the voltage applied to it



The diode allows current to flow in only one direction (usually, since in the breakdown region the current can flow in the opposite direction). Typically the *i*-*v* characteristics of a diode look like the following graph (numbers may vary from diode to diode).



We would like to measure the i-v characteristics of the diode; we will do this in two different ways. First, we construct a simple circuit that allows us to measure the current for different values of the voltage across the diode and plot them. Second, we will use the scope to display the i-v characteristics of the diode.

PRELAB

1. Simulate the following circuit (Fig 3_1) using Orcad Capture and PSpice.

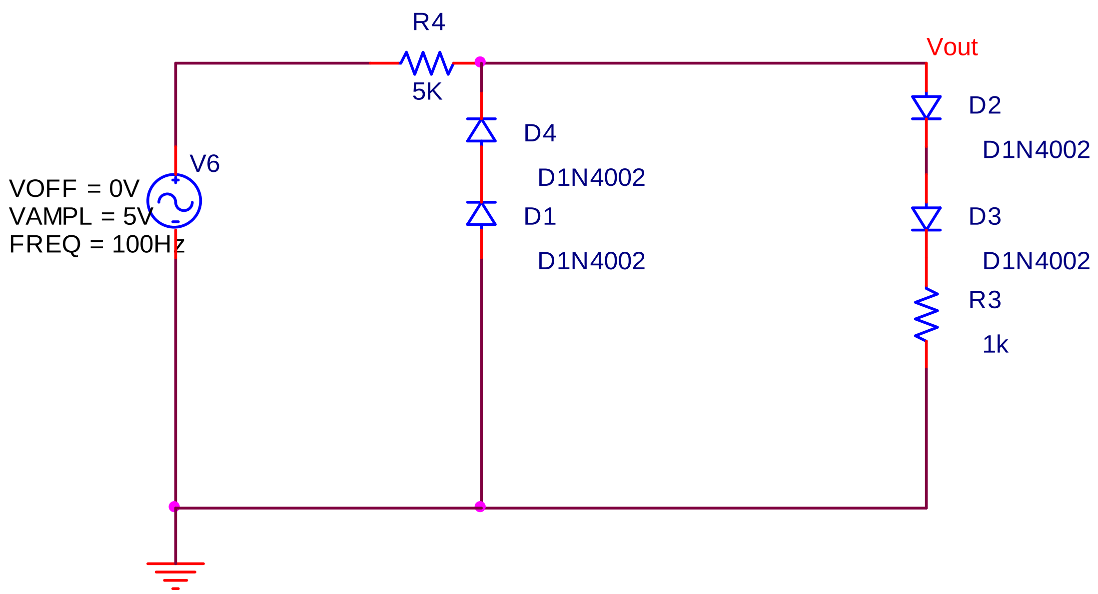


Figure: 3_1

   Plot the input voltage and the voltage across the two diodes D4 and D1 (you may want to use net alias to make this step easy).
   Plot the input voltage and the voltage across the two diodes D2 and D3.
   Plot the input voltage and the current in R3 and R4 (you have to be careful with the units to produce a reasonable graph).

2. Construct the circuit shown below in Fig 3_2 using Capture and simulate it for 80 msec. Show the voltage across the 100K resistor.
   This is a half wave rectifier. As you see this is not really DC.
   Now, connect a 1µF capacitor across the 100K resistor, simulate the circuit and show the voltage across the 100K resistor.
   If I want the variation in the voltage level to be limited to 10%, what is the minimum capacitor value to achieve that? Show the output voltage produced by SPICE.

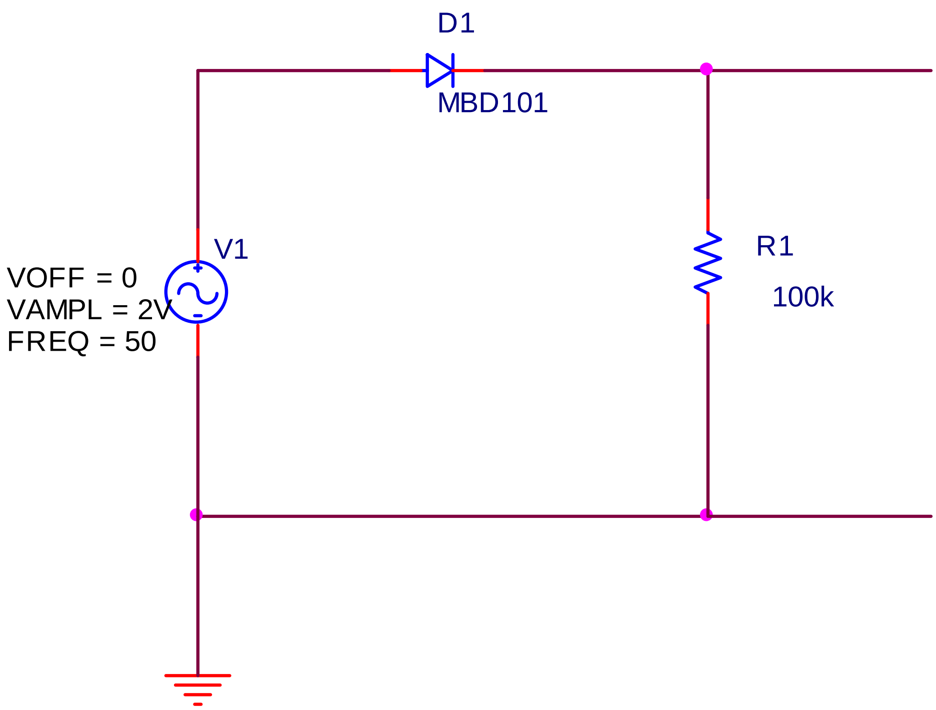


Figure: 3-2

LAB

Part I

Construct the following circuit (Fig 3_3). Get the TA to check it before proceeding.

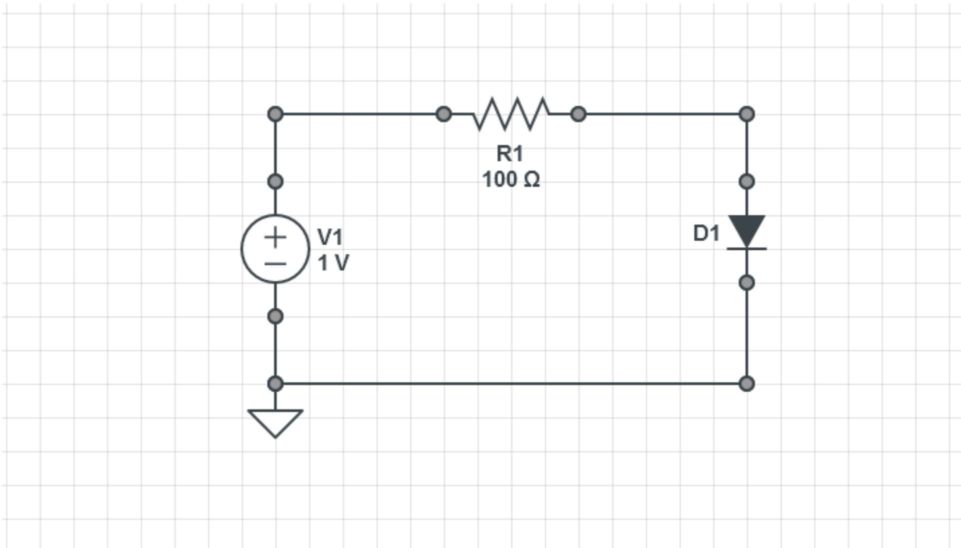


Figure 3_3

The diode available in the lab is 1N4933. Get the datasheet of the diode (www.digikey.ca) and check the maximum reverse voltage and the average forward current. You have to be sure not to exceed these values in your experiment (the average forward current is quite high, but you still have to check the datasheet).
Vary the input voltage, measure the current in the diode and the voltage across the diode. Plot the i-v characteristics for this diode in the forward region.

Part II

In this part you will use the previous circuit to estimate the diode parameters (parameters extraction).
An accurate equation for the diode current vs voltage is given as

$$I_D = I_s\left(e^{\frac{V_D}{nV_T}} - 1\right)$$

For Vd > 0.3 V you can neglect the -1 and the equation becomes

$$I_D = I_s\left(e^{\frac{V_D}{nV_T}}\right)$$

The values of *n* and *Is* can be different from a diode to another diode. You will measure them for the diode you are given in the lab.
Construct a circuit similar to the one in Fig 3_3, be sure to measure the value oif the resistor.
Take 2 measurements for 2 different V_D and I_D,
Now, you have 2 equations in 2 unknowns, with simple algebraic manipulations, you can get n as

$$n = \frac{V_{D1} - V_{D2}}{V_T \ln\left(\frac{I_{D1}}{I_{D2}}\right)}$$

You can find *n*,
Then find I_S
Compare the values you got with the values from the data sheet.

Part III

In this part, you will plot the “i-v” characteristics of the diode using the scope.
Set the scope in the x-y mode
The main idea is that if we connect the x (horizontal) to measure the voltage, and the y (vertical) to measure the current. By sweeping the voltage the current will be displayed on the y axis. The voltage across the diode is easy to measure. But, how to measure the

current? By measuring the voltage across a sensing resistance R_{sens}, the current is proportional to the voltage.

Construct the following circuit (Fig 3_4)

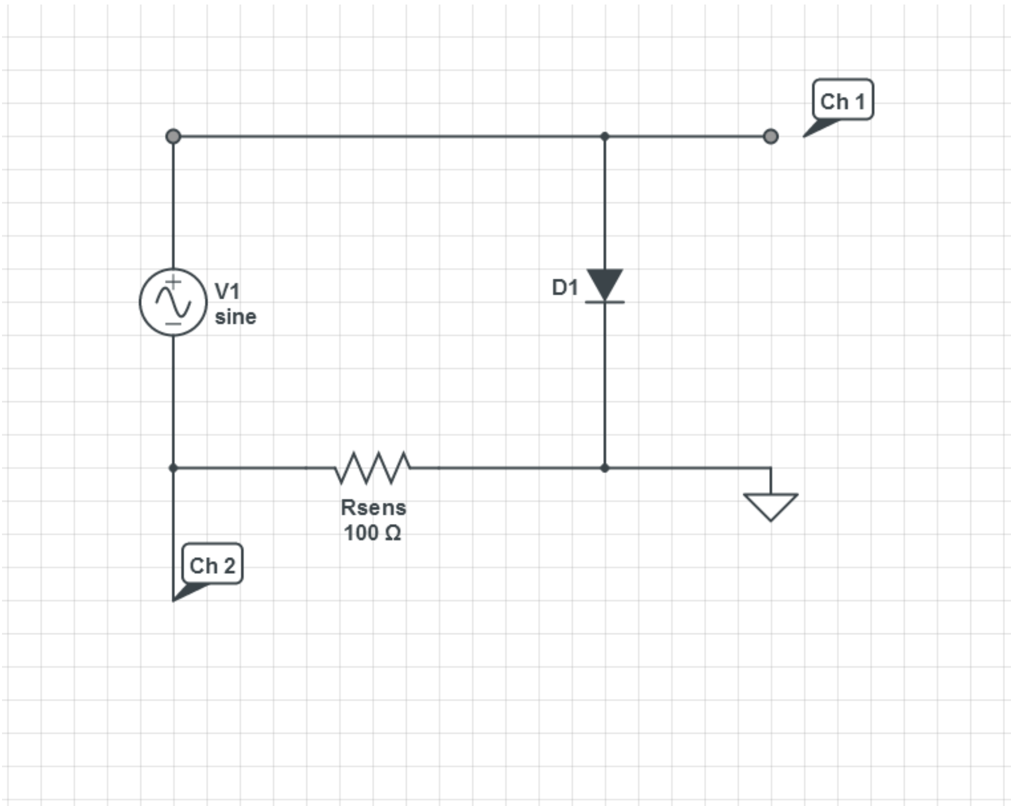


Figure: 3_4

Note that the function generator is floating, do not connect the ground to Rsens or you short it and a huge amount of current will flow through the diode destroying it.
Also, Ch 2 should be inverted to give the correct polarity.
Start with the function generator set to 2 V p-to-p. Adjust the scope setting to see a standard looking i-v curve for the diode.
Increase the input voltage to 4 V p-to-p. What is the effect on the i-v curve?
What is the limit on the input voltage in such a circuit?

Part IV

In this part, we will construct a full-wave rectifier.

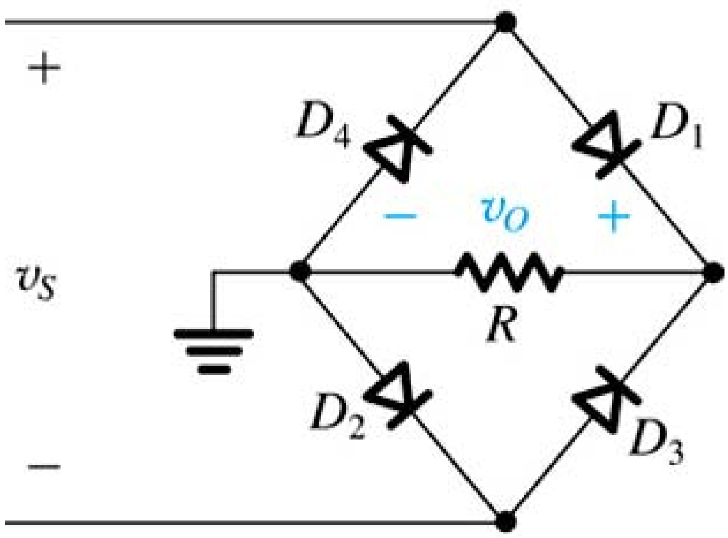


Figure: 3_5

Construct the above circuit. Set R = 1 KΩ.
Set the input to be 50 Hz sine wave with a 5 V p-to-p.
Display and sketch the output on the scope.
Now, shunt the load with a 10μF.
Display and sketch the waveform

PART V

In this part, you are required to measure the dynamic resistance of the diode. To recall

$$r_d = \left[\frac{\partial i_D}{\partial v_d}\right]_{i_D = I_D}$$

Construct a circuit to measure the dynamic resistance of the diode. Given the data sheet of the diode, justify your choice of I_D.
Show the schematic to the TA before you start
Compare your results with the datasheet.