EEM 201
ELECTRICAL CIRCUIT LABORATORY-I

EXPERIMENT BOOK

Prepared by:

T. Özge ONUR, Rıfat HACIOĞLU

BÜLENT ECEVİT UNIVERSITY

FACULTY OF ENGINEERING

DEPARTMENT OF ELECTRICAL AND ELECTRONICS ENGINEERING

ZONGULDAK, 2023

CONTENTS

Page Number

INTRODUCTION

SPICE is a powerful general purpose analog and mixed-mode circuit simulator that is used to verify circuit designs and to predict the circuit behavior. This is of particular importance for integrated circuits. It was for this reason that SPICE was originally developed at the Electronics Research Laboratory of the University of California, Berkeley (1975), as its name implies:

Simulation **P**rogram for **I**ntegrated **C**ircuits **E**mphasis.

PSpice is a PC version of SPICE (which is currently available from OrCAD Corp. of Cadence Design Systems, Inc.). A student version (with limited capabilities) comes with various textbooks. The OrCAD student edition is called PSpice AD Lite. Information about Pspice AD is available from the OrCAD website: http://www.orcad.com/pspicead.aspx

The PSpice Light version has the following limitations: circuits have a maximum of 64 nodes, 10 transistors and 2 operational amplifiers.

SPICE can do several *types of circuit analyses*. Here are the most important ones:

- Non-linear DC analysis: calculates the DC transfer curve.
- Non-linear transient and Fourier analysis: calculates the voltage and current as a function of time when a large signal is applied; Fourier analysis gives the frequency spectrum.
- Linear AC Analysis: calculates the output as a function of frequency. A bode plot is generated.
- Noise analysis
- Parametric analysis
- Monte Carlo Analysis

In addition, PSpice has analog and digital libraries of standard components (such as NAND, NOR, flip-flops, MUXes, FPGA, PLDs and many more digital components,). This makes it a useful tool for a wide range of analog and digital applications.

All analyses can be done at different temperatures. The default temperature is 300K. The circuit can contain the *following components*:

- Independent and dependent voltage and current sources

• Resistors

• Capacitors

• Inductors

• Mutual inductors

• Transmission lines

• Operational amplifiers

• Switches

• Diodes

• Bipolar transistors

• MOS transistors

• JFET

• MESFET

• Digital gates

• and other components (see users manual).

PSpice with OrCAD Capture (release 9.2 Lite edition)

Before one can simulate a circuit one needs to specify the circuit configuration. This can be done in a variety of ways. One way is to enter the circuit description as a text file in terms of the elements, connections, the models of the elements and the type of analysis. This file is called the SPICE input file or source file and has been described somewhere else (see http://www.seas.upenn.edu/%7Ejan/spice/spice.overview.html).

An alternative way is to use a schematic entry program such as OrCAD CAPTURE. OrCAD Capture is bundled with PSpice Lite AD on the same CD that is supplied with the textbook. Capture is a user-friendly program that allows you to capture the schematic of the circuits and to specify the type of simulation. Capture is non only intended to generate the input for PSpice but also for PCD layout design programs.

The following figure summarizes the different steps involved in simulating a circuit with Capture and PSpice. We'll describe each of these briefly through a couple of examples.

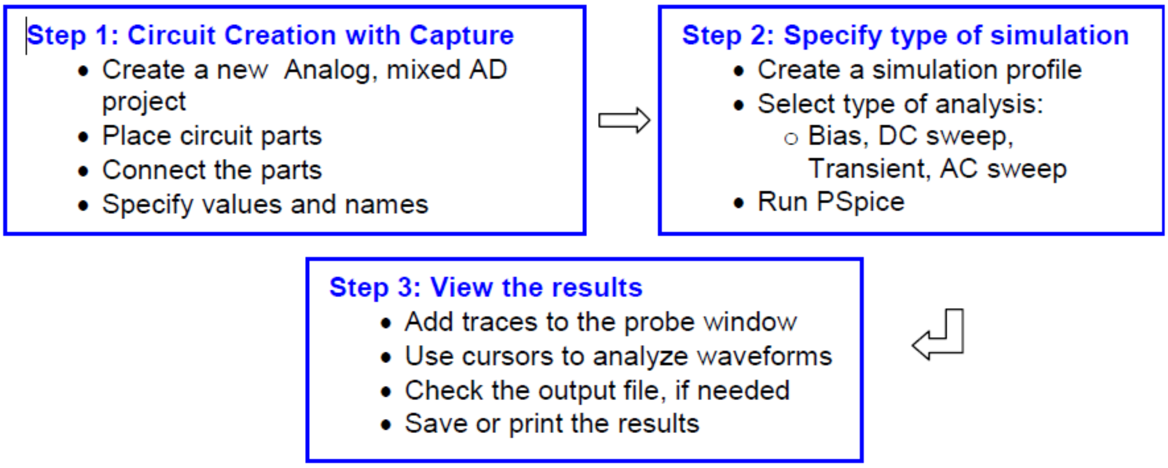


Figure 1. Steps involved in simulating a circuit with PSpice.

The values of elements can be specified using scaling factors (upper or lower case):

T or Tera (= 1E12);	U or Micro (= E-6);
G or Giga (= E9);	N or Nano (= E-9);
MEG or Mega (= E6);	P or Pico (= E-12)
K or Kilo (= E3);	F of Femto (= E-15)
M or Milli (= E-3);	

Both upper and lower case letters are allowed in PSpice and HSpice. As an example, one can specify a capacitor of 225 picofarad in the following ways:

225P, 225p, 225pF; 225pFarad; 225E-12; 0.225N

Notice that Mega is written as MEG, e.g. a 15 megaOhm resistor can be specified as 15MEG, 15MEGohm, 15meg, or 15E6. Be careful not to use M for Mega! When you write 15 Mohm or 15M, Spice will read this as 15 milliOhm! We'll illustrate the different types of simulations for the following circuit:

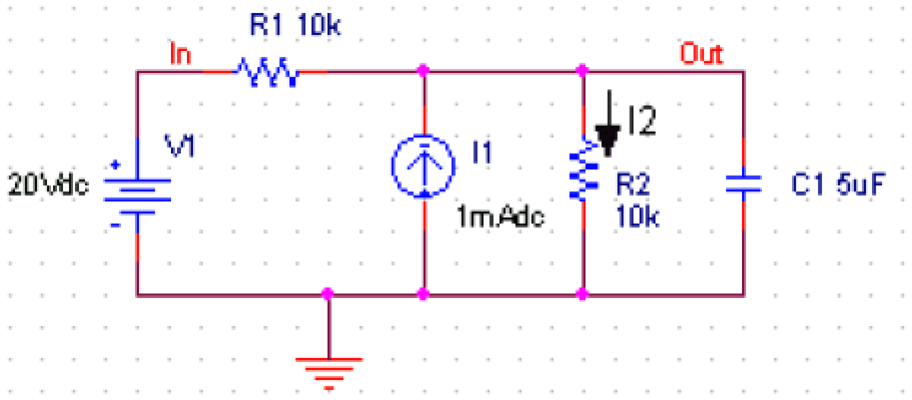


Figure 2. Circuit to be simulated (screen shot from OrCAD Capture).

Step 1: Creating the circuit in Capture

Create new project:

1. Open OrCAD Capture
2. Create a new Project: FILE MENU/NEW_PROJECT
3. Enter the name of the project
4. Select Analog or Mixed-AD
5. When the Create PSpice Project box opens, select "Create Blank Project".

A new page will open in the Project Design Manager as shown below.

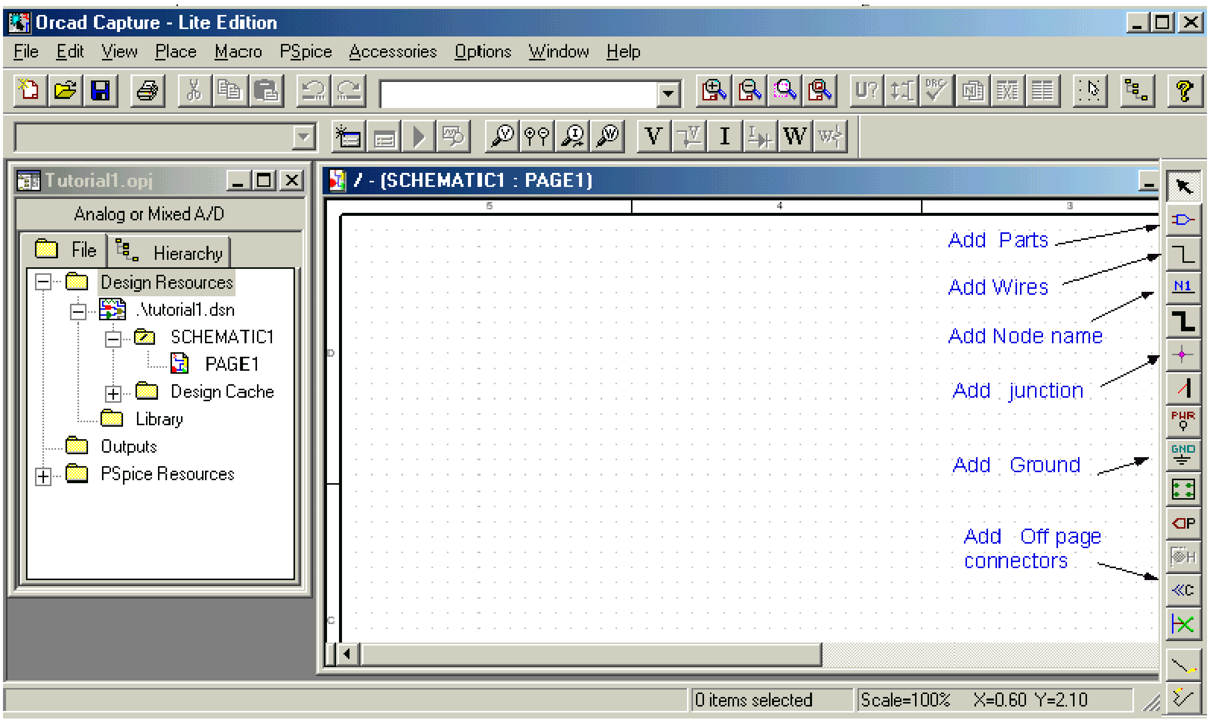


Figure 3. Design manager with schematic window and toolbars (OrCAD screen capture)

Place the components and connect the parts

1. Click on the Schematic window in Capture.
2. To Place a part go to PLACE/PART menu or click on the Place Part Icon. This will open a dialog box shown below.

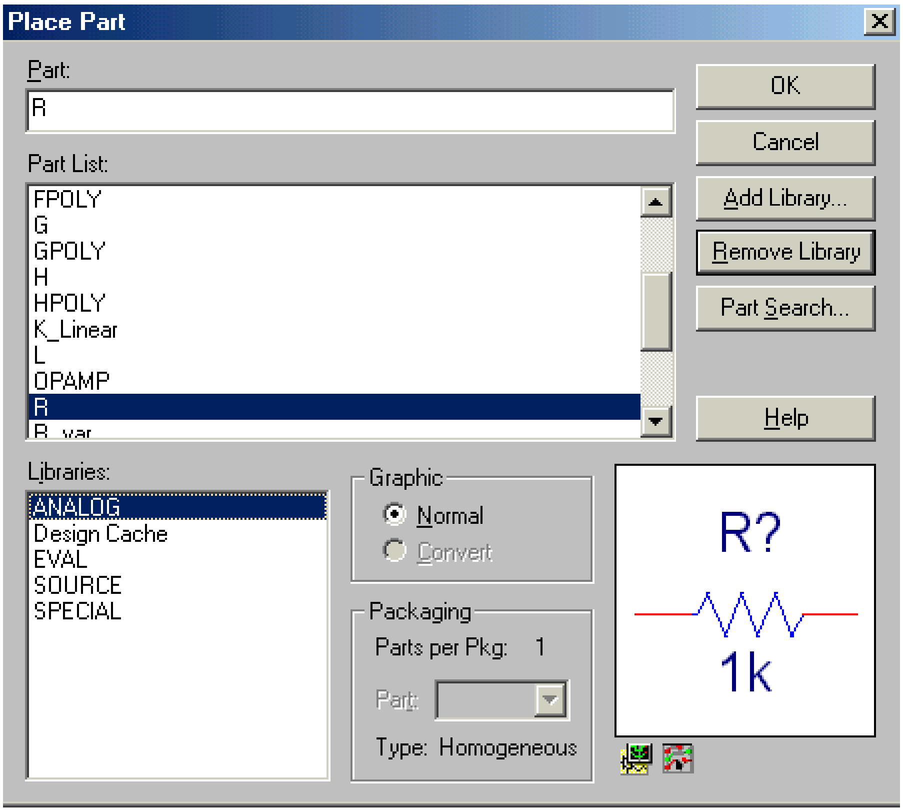


Figure 4. Place Part window

3. Select the library that contains the required components. Type the beginning of the name in the Part box. The part list will scroll to the components whose name contains the same letters. If the library is not available, you need to add the library, by clicking on the Add Library button. This will bring up the Add Library window. Select the desired library. For Spice you should select the libraries from the Capture/Library/PSpice folder.

   Analog: contains the passive components (R,L,C), mutual inductane, transmission line, and voltage and current dependent sources (voltage dependent voltage source E, current-dependent current source F, voltage-dependent current source G and current-dependent voltage source H).

Source: give the different type of independent voltage and current sources, such as Vdc, Idc, Vac, Iac, Vsin, Vexp, pulse, piecewise linear, etc. Browse the library to see what is available.

Eval: provides diodes (D…), bipolar transistors (Q…), MOS transistors, JFETs (J…), real opamp such as the u741, switches (SW_tClose, SW_tOpen), various digital gates and components.

Abm: contains a selection of interesting mathematical operators that can be applied to signals, such as multiplication (MULT), summation (SUM), Square Root (SWRT), Laplace (LAPLACE), arctan (ARCTAN), and many more.

Special: contains a variety of other components, such as PARAM, NODESET, etc.

4. Place the resistors, capacitor (from the Analog library), and the DC voltage and current source. You can place the part by the left mouse click. You can rotate the components by clicking on the R key. To place another instance of the same part, click the left mouse button again. Hit the ESC key when done with a particular element. You can add initial conditions to the capacitor. Double-click on the part; this will open the Property window that looks like a spreadsheet. Under the column, labeled IC, enter the value of the initial condition, e.g. 2V. For our example we assume that IC was 0V (this is the default value).
5. After placing all part, you need to place the Ground terminal by clicking on the GND icon (on the right side toolbar – see Figure 3). When the Place Ground window opens, select GND/CAPSYM and **give it the name 0 (i.e. zero).** Do not forget to change the name to 0, otherwise PSpice will give an error or "Floating Node". The reason is that SPICE needs a ground terminal as the reference node that has the node number or name 0 (zero).

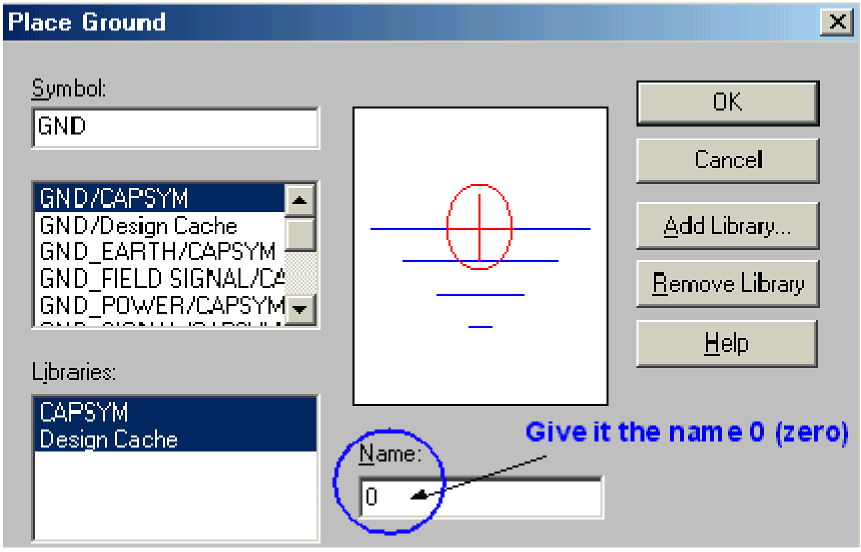


Figure 5. Place the ground terminal box; the ground terminal should have the name 0

6. Now connect the elements using the Place Wire command from the menu (PLACE/WIRE) or by clicking on the Place Wire icon.
7. You can assign names to nets or nodes using the Place Net Alias command (PLACE/NET ALIAS menu). We will do this for the output node and input node. Name these Out and In, as shown in Figure 2.

Assign Values and Names to the parts

1. Change the values of the resistors by double-clicking on the number next to the resistor. You can also change the name of the resistor. Do the same for the capacitor and voltage and current source.
2. If you haven't done so yet, you can assign names to nodes (e.g. Out and In nodes).
3. Save the project

Netlist

The netlist gives the list of all elements using the simple format:

R_name node1 node2 value

C_name nodex nodey value, etc.

1. You can generate the netlist by going to the PSPICE/CREATE NETLIST menu.
2. Look at the netlist by double clicking on the Output/name.net file in the Project Manager Window (in the left side File window).

Note on Current Directions in elements:

The positive current direction in an element such as a resistor is from node 1 to node 2. Node 1 is either the left pin or the top pin for an horizontal or vertical positioned element (.e.g a resistor). By rotating the element 180 degrees one can switch the pin numbers. To verify the node numbers you can look at the netlist:

e.g. R_R2 node1 node2 10k

e.g. R_R2 0 OUT 10k

Since we are interested in the current direction from the OUT node to the ground, we need to rotate the resistor R2 twice so that the node numbers are interchanged:

R_R2 OUT 0 10k

Step 2: Specifying the type of analysis and simulation

As mentioned in the introduction, Spice allows you do to a DC bias, DC Sweep, Transient with Fourier analysis, AC analysis, Montecarlo/worst case sweep, Parameter sweep and Temperature sweep. We will first explain how to do the Bias and DC Sweep on the circuit of Figure 2.

BIAS or DC analysis

1. With the schematic open, go to the PSPICE menu and choose NEW SIMULATION PROFILE.
2. In the Name text box, type a descriptive name, e.g. Bias
3. From the Inherit From List: select none and click Create.
4. When the Simulation Setting window opens, for the Analyis Type, choose Bias Point and click OK.
5. Now you are ready to run the simulation: PSPICE/RUN
6. A window will open, letting you know if the simulation was successful. If there are errors, consult the Simulation Output file.
7. To see the result of the DC bias point simulation, you can open the Simulation Output file or go back to the schematic and click on the V icon (Enable Bias Voltage Display) and I icon (current display) to show the voltage and currents (see Figure 6).

The check the direction of the current, you need to look at the netlist: the current is positive flowing from node1 to node1 (see note on Current Direction above).

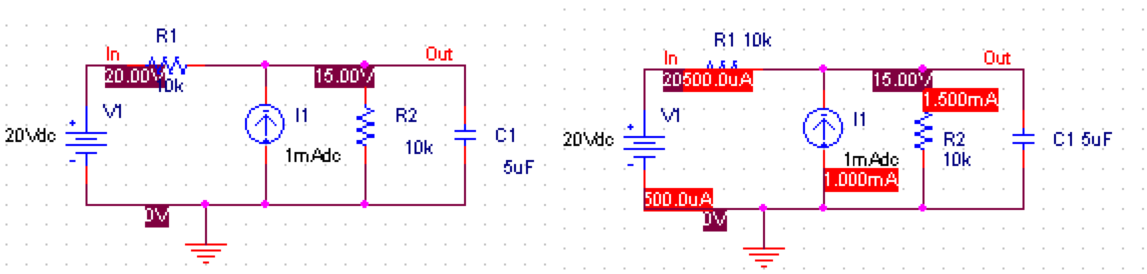


Figure 6. Results of the Bias simulation displayed on the schematic.

DC Sweep simulation

We will be using the same circuit but will evaluate the effect of sweeping the voltage source between 0 and 20V. We'll keep the current source constant at 1mA.

1. Create a new New Simulation Profile (from the PSpice Menu); We'll call it DC Sweep
2. For analysis select DC Sweep; enter the name of the voltage source to be swept: V1. The start and end values and the step need to be specified: 0, 20 and 0.1V, respectively (see figure below).

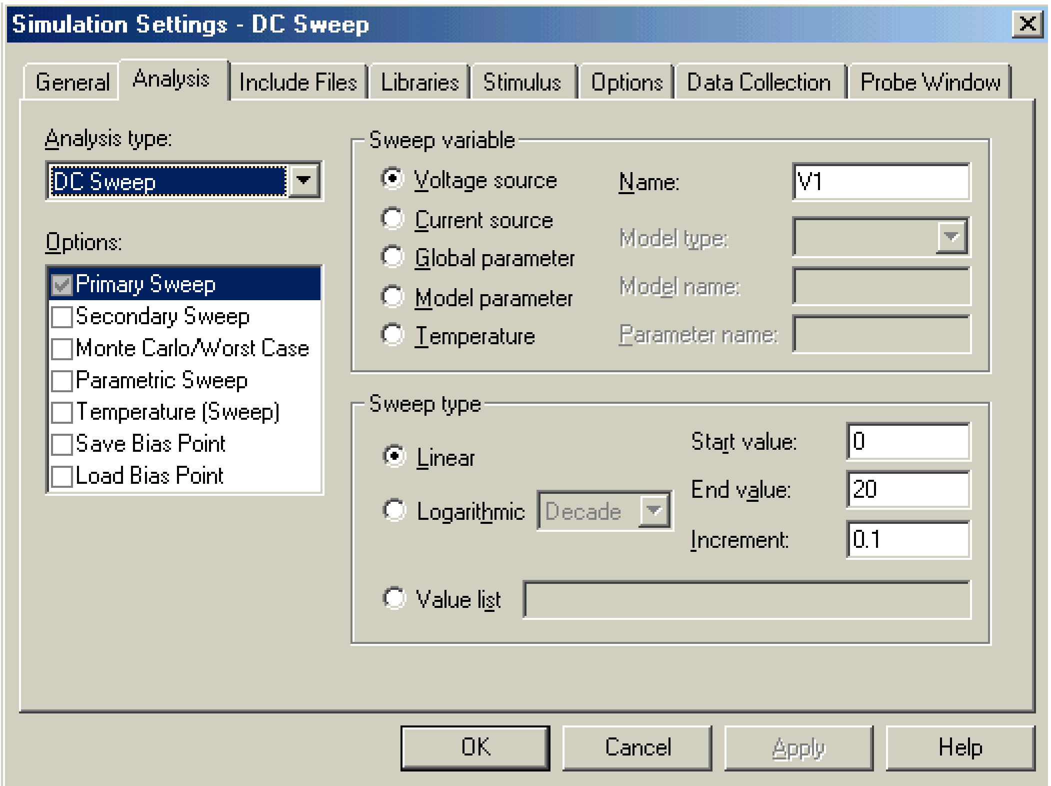


Figure 7. Setting for the DC Sweep simulation.

1. Run the simulation. PSpice will generate an output file that contains the values of all voltages and currents in the circuit.

Step 3: Displaying the simulation Results

PSpice has a user-friendly interface to show the results of the simulations. Once the simulation is finished a Probe window will open.

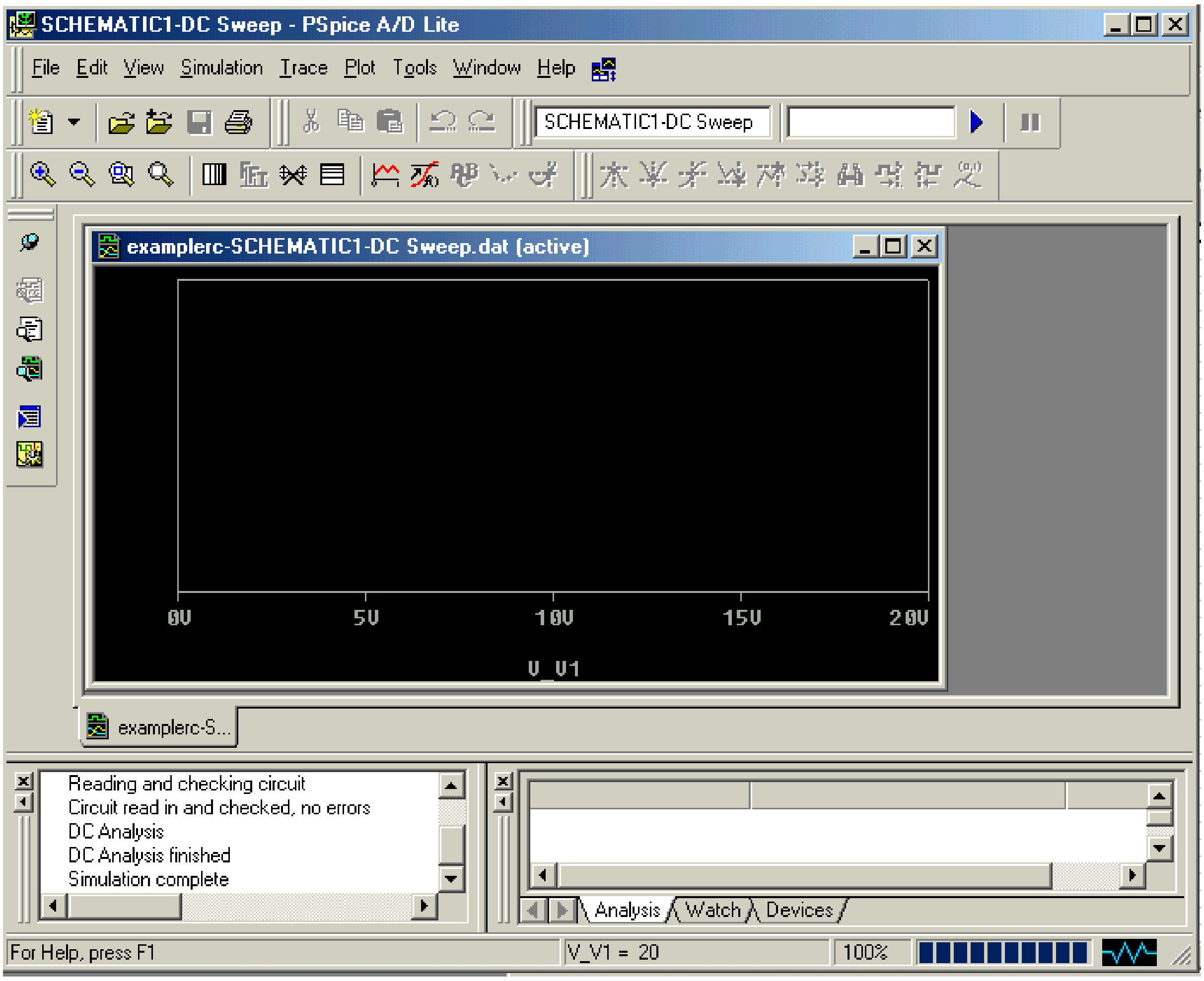


Figure 8. Probe window

1. From the TRACE menu select ADD TRACE and select the voltages and current you like to display. In our case we'll add V(out) and V(in). Click OK.

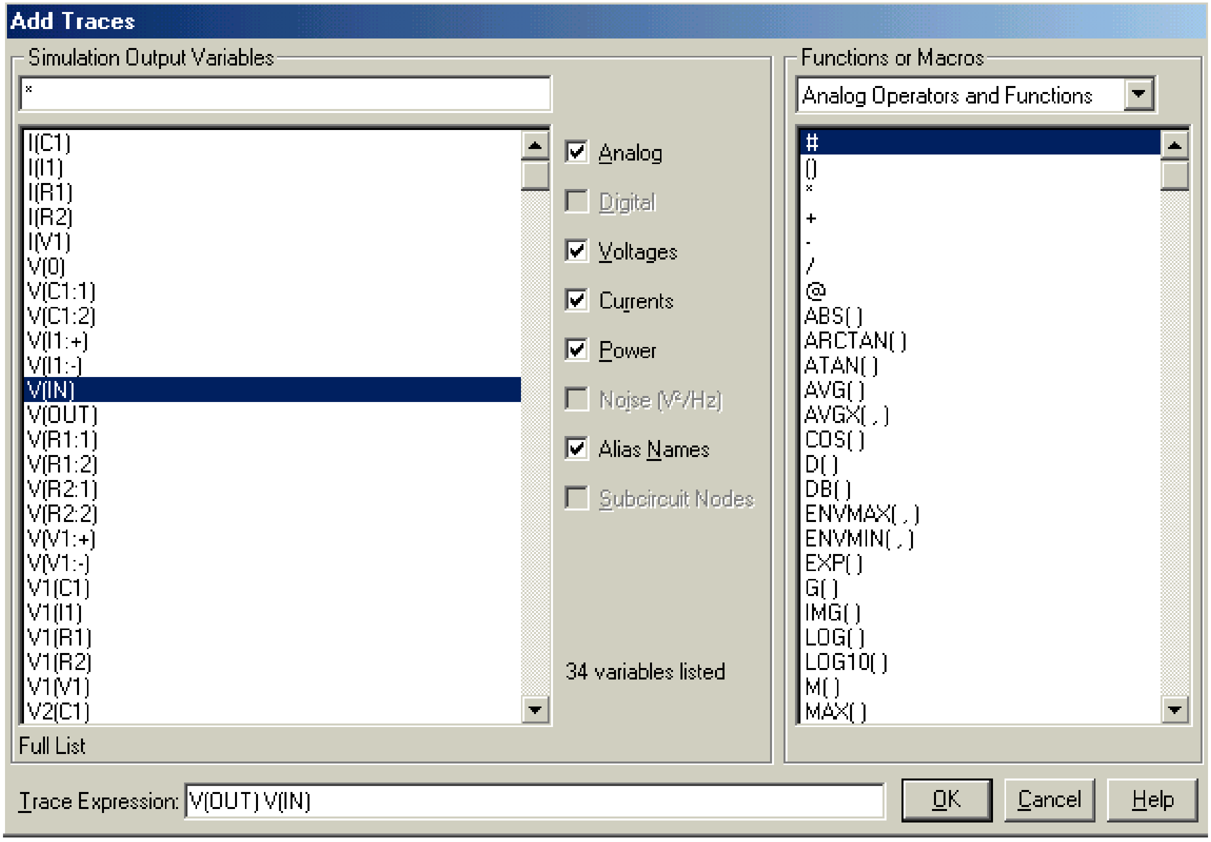


Figure 9. Add Traces window

2. You can also add traces using the "Voltage Markers" in the schematic. From the PSPICE menu select MARKERS/VOLTAGE LEVELS. Place the makers on the Out and In node. When done, right click and select End Mode.

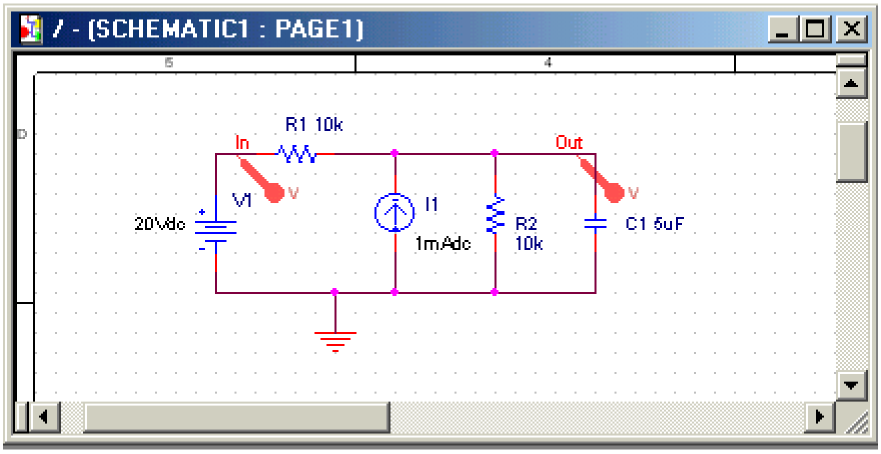


Figure 10. Using Voltage Markers to show the simulation result of V(out) and V(in)

3. Go to back to PSpice. You will notice that the waveforms will appear.

4. You can add a second Y Axis and use this to display e.g. the current in Resistor R2, as shown below. Go to PLOT/Add Y Axis. Next, add the trace for I(R2).

2. You can also use the cursors on the graphs for Vout and Vin to display the actual values at certain points. Go to TRACE/CURSORS/DISPLAY

3. The cursors will be associated with the first trace, as indicated by the small small rectangle around the legend for V(out) at the bottom of the window. Left click on the first trace. The value of the x and y axes are displayed in the Probe window. When you right click on V(out) the value of the second cursor will be given together with the difference between the first and second cursor.

4. To place the second cursor on the second trace (for V(in)), right click the legend for V(in). You'll notice the outline around V(in) at the bottom of the window. When you right click the second trace the cursor will snap to it. The values of the first and second cursor will be shown in Probe window.

5. You can chance the X and Y axes by double clicking on them.

6. When adding traces you can perform mathematical calculations on the traces, as indicated in the Add Trace Window to the right of Figure 9.

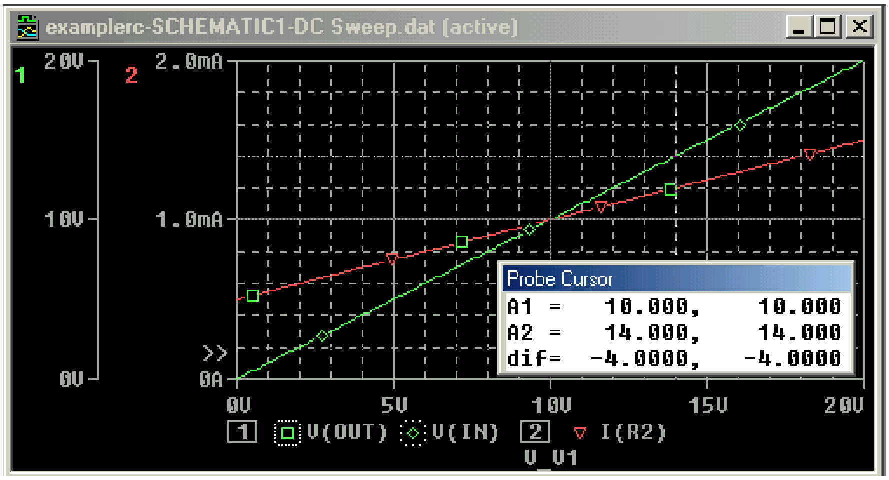


Figure 11. Result of the DC sweep, showing Vout, Vin and the current through resistor R2. Cursors were used for V(out) and V(in).

Other types of Analysis

Transient Analysis

We'll be using the same circuit as for the DC sweep, except that we'll apply the voltage and current sources by closing a switch, as shown in Figure 12.

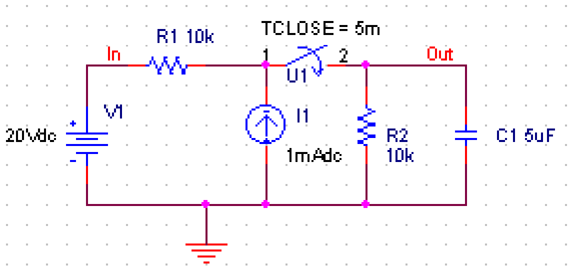


Figure 12. Circuit used for the transient simulation.

1. Insert the SW_TCLOSE switch from the EVAL Library as shown above. Double click on the switch TCLOSE value and enter the value when the switch closes. Lets make TCLOSE = 5 ms.
2. Set up the Transient Analysis: go to the PSPICE/NEW SIMULATION PROFILE.
3. Give it a name (e.g. Transient). When the Simulation Settings window opens, select "Time Domain (Transient)" Analysis. Enter also the Run Time. Lets make it 50 ms. For the Max Step size, you can leave it blank or enter 10us.
4. Run PSpice.
5. A Probe window in PSpice will open. You can now add the traces to display the results.

In the figure below we plotted the current through the capacitor in the top window and the voltage over the capacitor on the bottom one. We use the cursor to find the time constant of the exponential waveform (by finding the 0.632 x V(out)max = 9.48. The cursor gave a corresponding time of 30ms which gives a time constant of 30-5=25ms (5 ms is subtracted because the switch closed at 5ms).

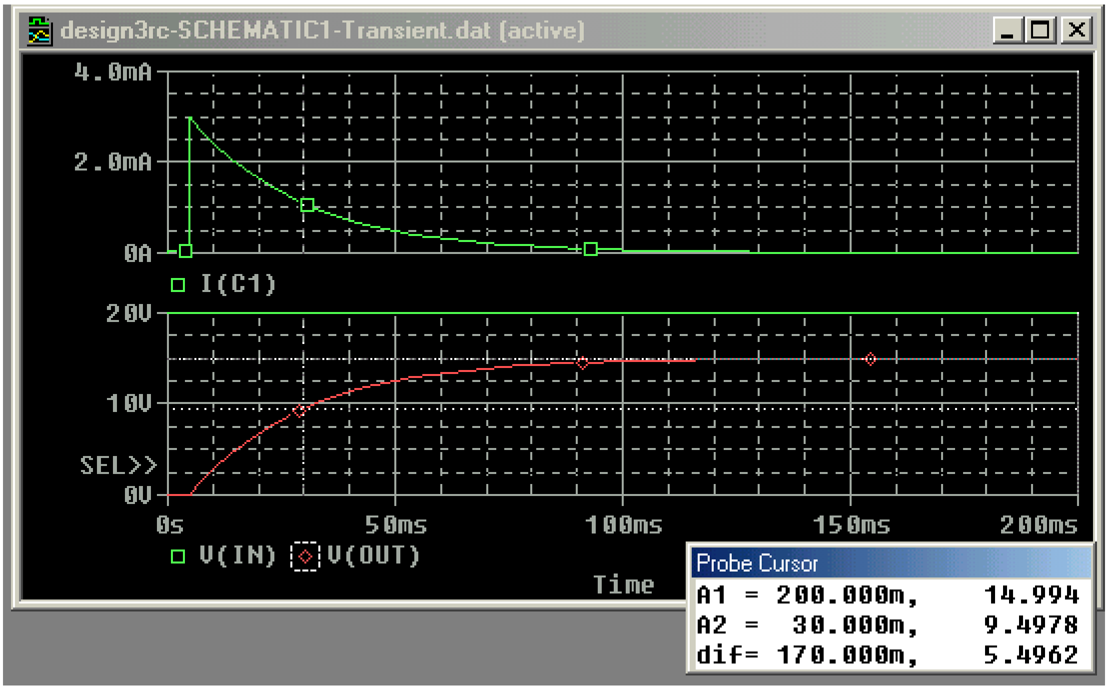


Figure 13. Results of the transient simulation of Figure 12.

6. Instead of using a switch we can also use a voltage source that changes over time. This was done in Figure 14 where we used the VPULSE and IPULSE sources from the SOURCE Library. We entered the voltage levels (V1 and V2), the delay (TD), Rise and Fall Times, Pulse Width (PW) and the Period (PER). The values are indicated in the figure below. For details on these parameters click here. A description of other Spice elements can be found in the User's guide or in the Spice Tutorial. (http://www.seas.upenn.edu/~jan/spice/)

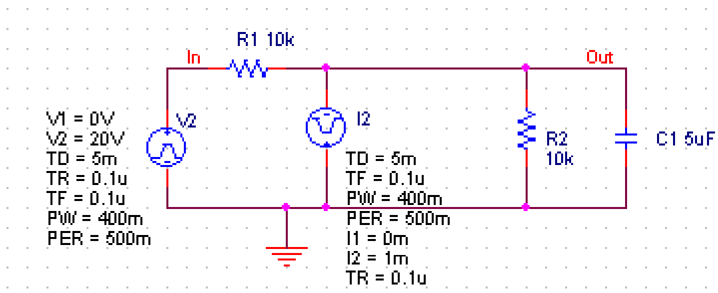


Figure 14. Circuit with a PULSE voltage and current source.

7. After doing the transient simulation results can be displayed as was done before
8. The last example of a transient analysis is with a sinusoidal signal VSIN. The circuit is shown below. We made the amplitude 10V and frequency 10 Hz.

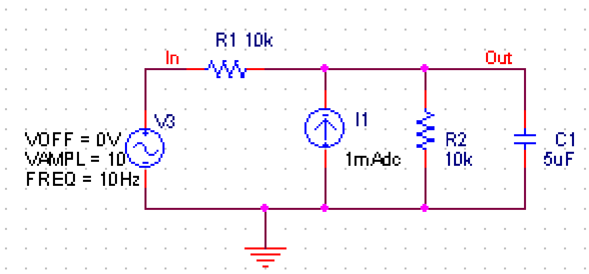


Figure 15. Circuit with a sinusoidal input.

9. Create a Simulation Profiler for the transient analysis and run PSpice.
10. The result of the simulation for Vout and Vin are given in the figure below.

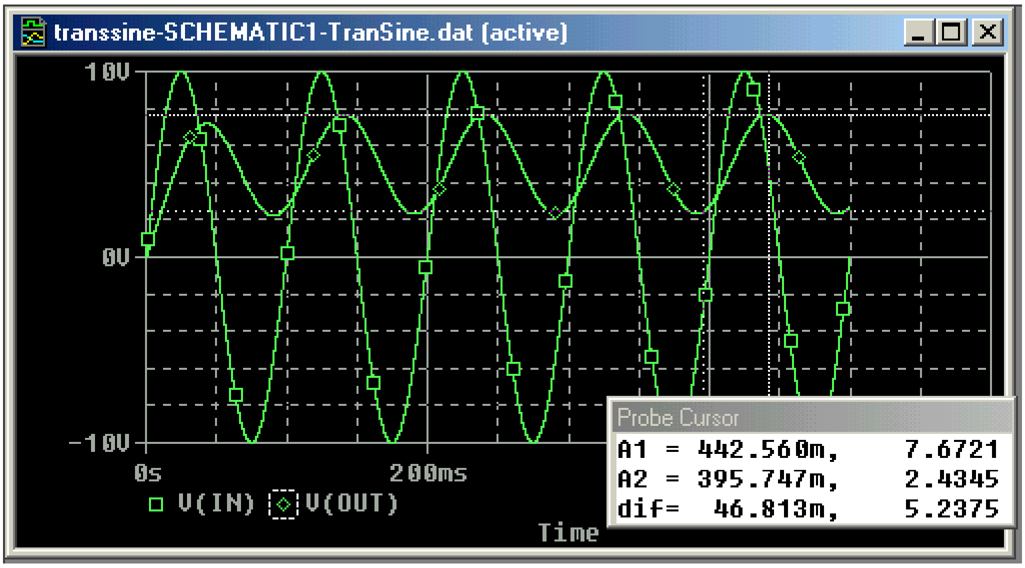


Figure 16. Transient simulation with a sinusoidal input.

AC Sweep Analysis

The AC analysis will apply a sinusoidal voltage whose frequency is swept over a specified range. The simulation calculates the corresponding voltage and current amplitude and phases for each frequency. When the input amplitude is set to 1V, then the output voltage is basically the transfer function. In contrast to a sinusoidal transient analysis, the AC analysis is not a time domain simulation but rather a simulation of the sinusoidal steady state of the circuit. When the circuit contains non-linear element such as diodes and transistors, the elements will be replaced their small-signal models with the parameter values calculated according to the corresponding biasing point.

In the first example, we'll show a simple RC filter corresponding to the circuit of Figure 17.

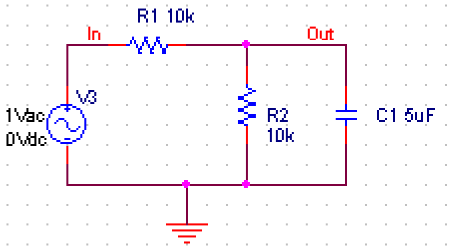
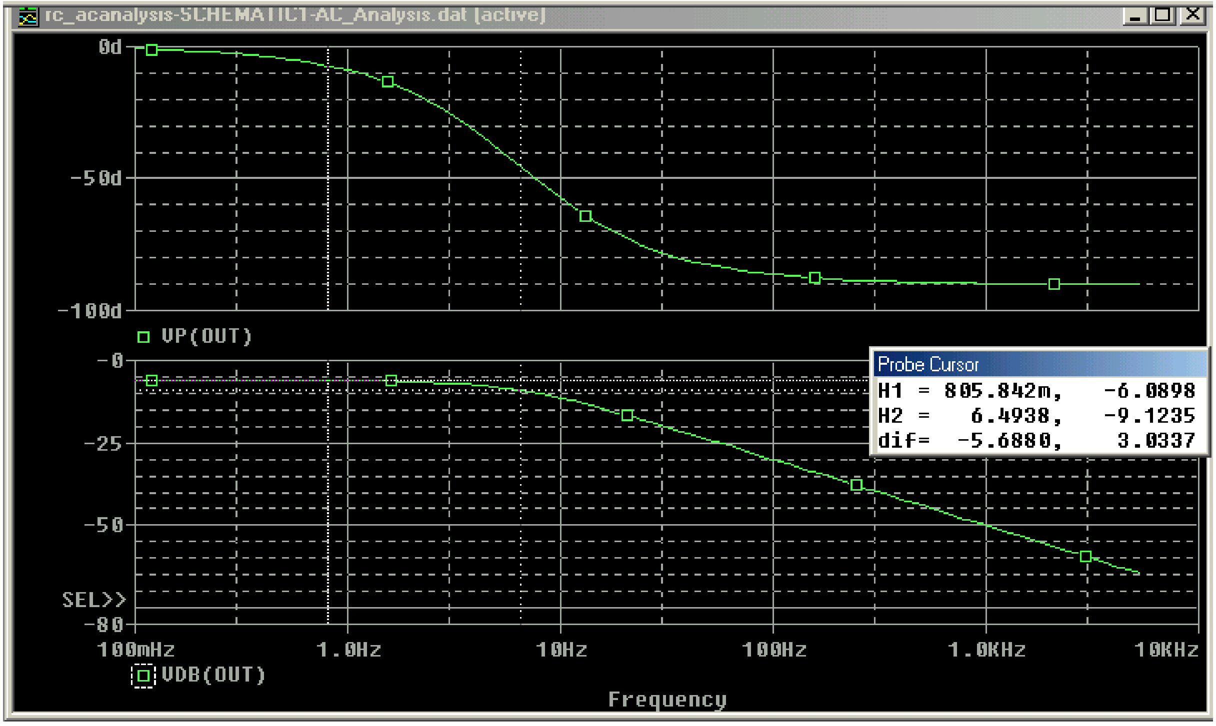


Figure 17. Circuit for the AC sweep simulation.

1. Create a new project and build the circuit
2. For the voltage source use VAC from the Sources library.
3. Make the amplitude of the input source 1V.
4. Create a Simulation Profile. In the Simulation Settings window, select AC Sweep/Noise.
5. Enter the start and end frequencies and the number of points per decade. For our example we use 0.1Hz, 10 kHz and 11, respectively.
6. Run the simulation
7. In the Probe window, add the traces for the input voltage. We added a second window to display the phase in addition to the magnitude of the output voltage. The voltage can be displayed in dB by specifying Vdb(out) in the Add Trace window (type Vdb(out) in the Trace Expression box. For the phase, type VP(out).
8. An alternative to show the voltage in dB and phase is to use markers on the schematics: PSPICE/MARKERS/ADVANCED/dBMagnitude or Phase of Voltage, or current. Place the markers on the node of interest.
9. We used the cursors in Figure 18 to find the 3dB point. The value is 6.49 Hz corresponding to a time constant of 25 ms (R1||R2.C). At 10 Hz the attenuation of Vout is 11.4db or a factor of 3.72. This corresponds to the value of the amplitude of the output voltage obtained during the transient analysis of Figure 16 above.



Additional Circuit Examples with PSpice

Transformer circuit

SPICE has no model for an ideal transformer. An ideal transformer is simulated using mutual inductances such that the transformer ratio N1/N2 = sqrt(L1/L2). The part in PSpice is called TFRM_LINEAR (in the Analog Library). Make the coupling factor K close to or equal to one (ex. K=1) and choose L such that wL >> the resistance seen be the inductor. The secondary circuit needs a DC connection to ground. This can be accomplished by adding a large resistor to ground or giving the primary and secondary circuits a common node. The following example illustrates how to simulate a transformer.

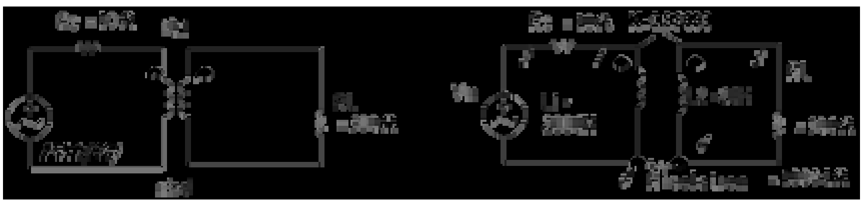


Figure 18. Circuit with ideal transformer

For the above example, lets make wL2 >> 500 Ohm or L2> 500/(60*2pi) ; lets make L2 at least 10 times larger, ex. L2=20H. L1 can than be found from the turn ratio: L1/L2 = (N1/N2)$^{\wedge 2}$. For a turn ratio of 10 this makes L1=L2x100=2000H. The circuit as entered in PSpice Capture is shown in Figure 19 and the result in Figure 20.

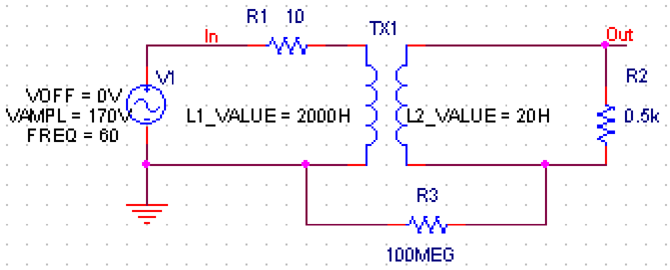


Figure 19. Circuit with ideal transformer as entered in PSpice Capture (the transformer TX is modeled by the part XFRM_LINEAR of the Analog Library).

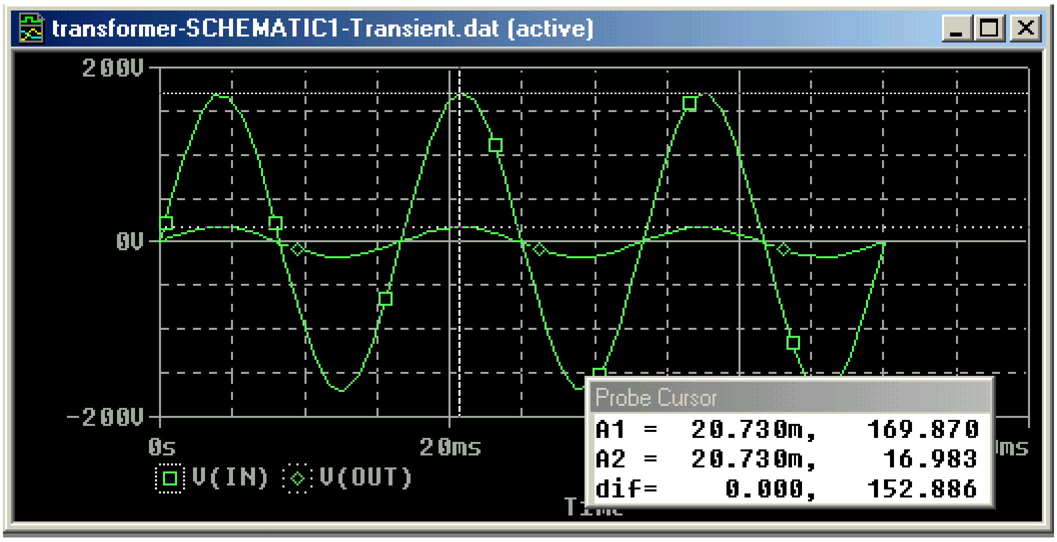


Figure 20. Results of the transient simulation of the above circuit.

AC Sweep of Filter with Ideal Op-amp (Filter circuit)

The following circuit will be simulated with PSpice.

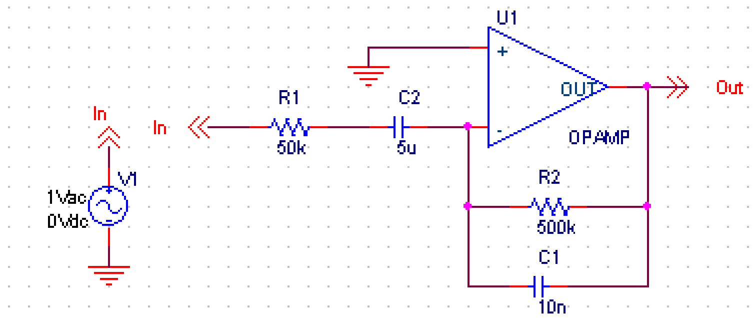


Figure 21. Active Filter Circuit with ideal op-amp.

We have used off-page connectors (OFFPAGELEFT-R from the CAPSYM library; or by clicking on the off-page icon) for the input and outputs. The name of the connectors can be changed by

double-clicking on the name of the off-page connector. By giving the same name to two connectors (or nodes), the two nodes will be connected (no wires are needed). For te voltage source we used the VAC from the SOURCE Library. We gave it an amplitude of 1V so that the output voltage will correspond to the amplification (or transfer function) of the filter. In the Simulation Analysis, select AC Sweep, and enter the starting, ending frequency and the number of points per decade.
The result is given in the figure below. The magnitude is given on the left Y axis while the phase is given by the right Y axis. The cursors have been used to find the 3db points of the bandpass filters, corresponding to 0.63 Hz and 32 Hz for the low and high breakpoints, respectively. These numbers correspond to the values of the time constants given in Figure 20. The phase at these points is -135 and -224 degrees.

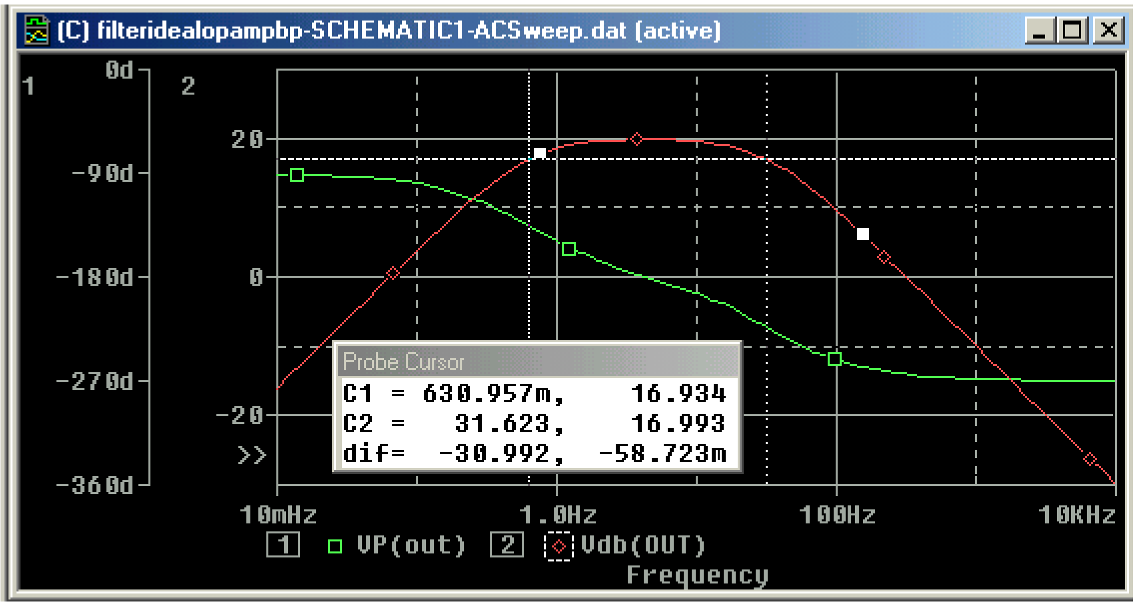


Figure 22. Results of the AC sweep of the Active Filter Circuit of the figure above.

AC Sweep of Filter with Real Op-amp (Filter circuit)

The circuit with a real op-amp is shown below. We selected the U741 op-amp to build the filter. The simulation results are shown in Figure 24. As one would expect the differences betweenthe filter with the real and ideal op-amps are minimal in this frequency range.

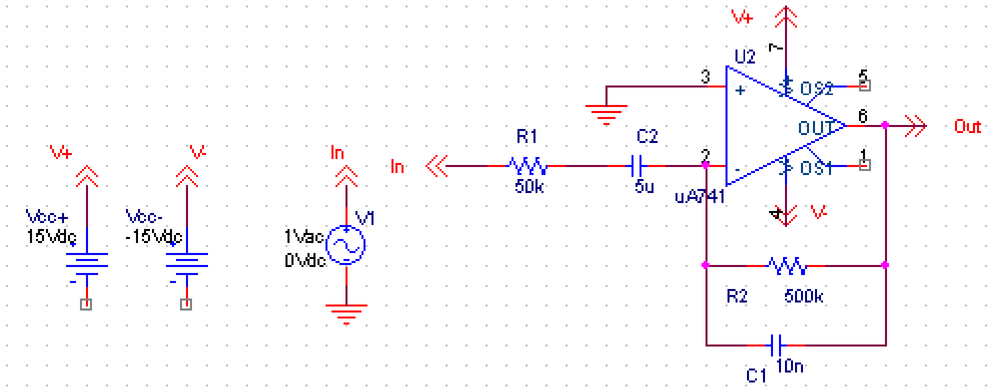


Figure 23. Active Filter Circuit with the U741 Op-amp.

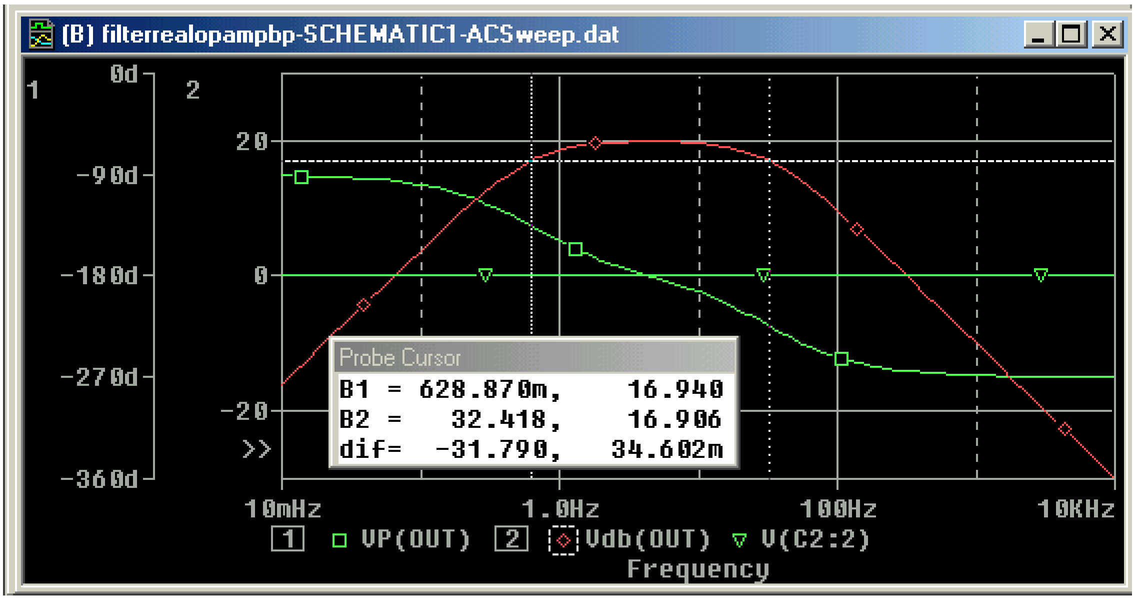


Figure 24. Results of the AC sweep of the Active Filter Circuit with real Op-amp (U741) of the figure above.

Rectifier Circuit (peak detector) and the use of a parametric sweep

Peak Detector simulation

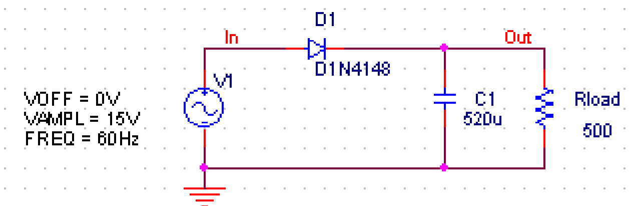


Figure 24. Rectifier circuit with the D1N4148 diode and a load resistor of 500 Ohm.

The results of the simulation are given in Figure 25. The ripple has a peak-to-peak value of 777mV as indicated by the cursors. The maximum output voltage is 13.997V which is one volt below the input of 15V.

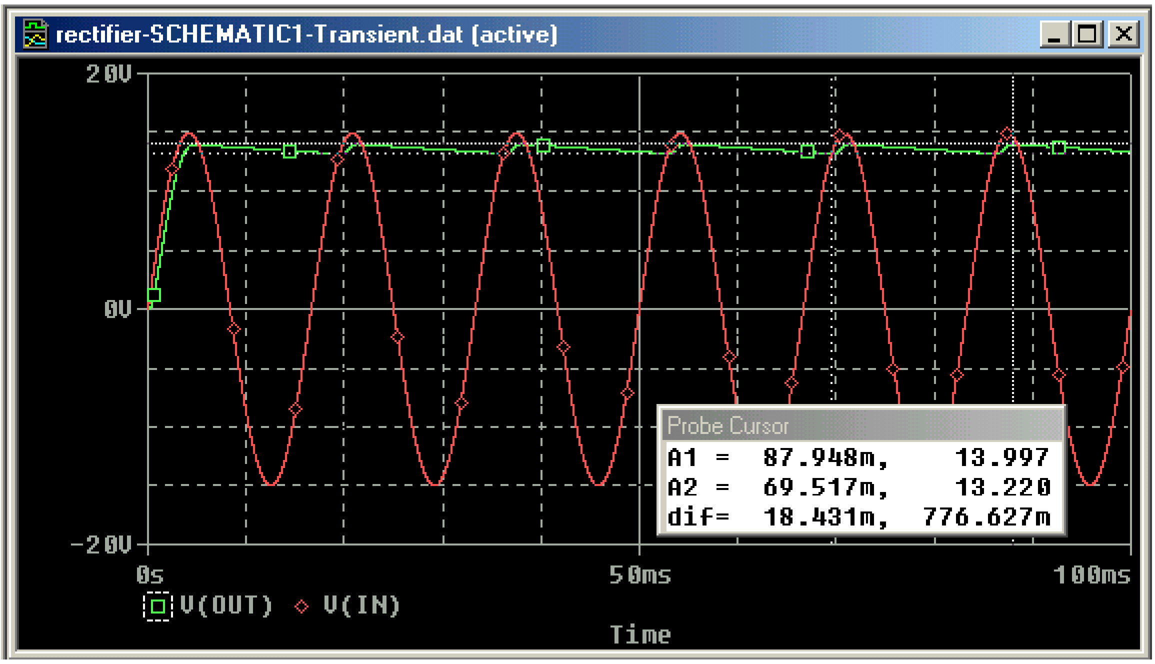


Figure 25. Simulation results of the rectifier circuit.

Parametric Sweep

It is interesting to see the effect of the load resistance on the output voltage and its ripple voltage. This can be done using the PARAM part.

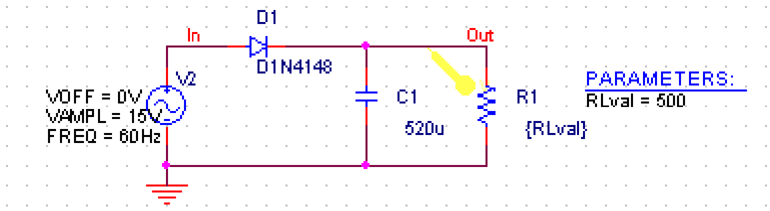


Figure 26. Circuit used for the parametric sweep of the load resistor.

<u>a. Adding the Parameter Part</u>

a. Double click on the value (500 Ohms) of the load resistor R1 to {Rval}. Use curly brackets. PSpice interprets the text between curly brackets as an expression that it evaluate to a numerical expression. Click OK when done.

b. Add the PARAM part to the circuit. You'll find this part in the SPECIAL library.

c. Double click on the PARAM part. This will open a spreadsheet like window showing the PARAM definition. You will need to add a new column to this spread sheet. Click on NEW COLUMN and enter for Property Name, Rlval (without the curly brackets).

d. You will notice that the new column Rlval has been created. Below the Rlval enter the initial value for the resistor: lets make it 500, as shown in Figure 3.4.4 below.

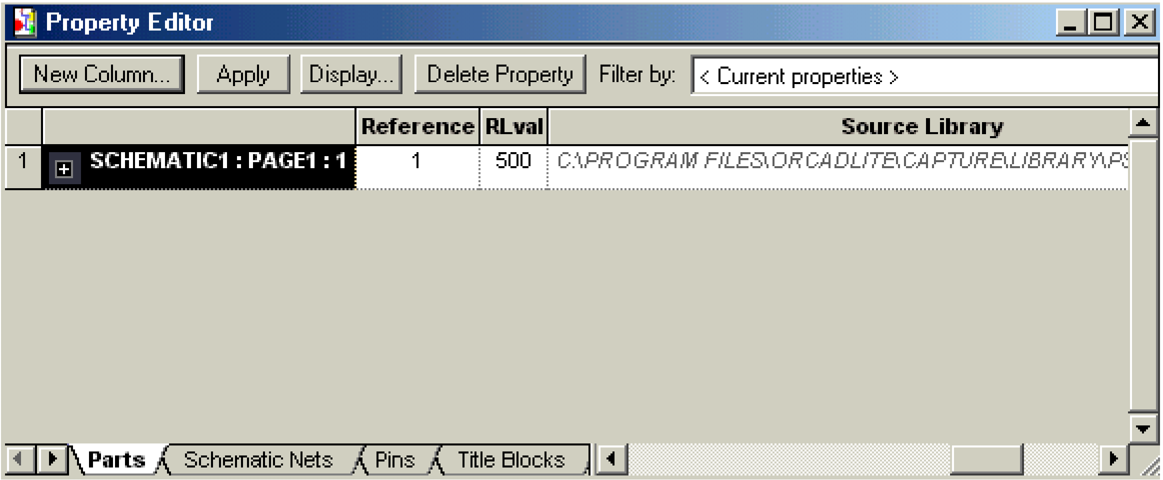


Figure 27. Property Editor window for the PARAM part, showing the newly created Rlval column.

e. While the cell in which you entered the value 500 still selected click the DISPLAY button. You can now specify what to display: select Name and Value. Click OK.

f. Click the APPLY button before closing the Property editor.

g. Save the design.

b. Create the Simulation Profile for the Parametric Analysis

a. Select PSPICE/NEW_SIMULATION_PROFILE

b. Type in the name of the profile, e.g. Parametric

c. In the Simulation Setting window, select Analysis Tab if the window does not open.

d. For the Analysis type select Transient (or the type of analysis you intend to perform; in this example we'll do a transient analysis)

e. Under Option, slect Parametric sweep as shown in Figure 28.

f. For the Sweep Variable, select Global Parameter and enter the Parameter name: Rlval. Under sweep type give the start, end and increment for the parameter. We'll used 250, 1kOhm and 250, respectively (see Figure 28).

g. Click OK

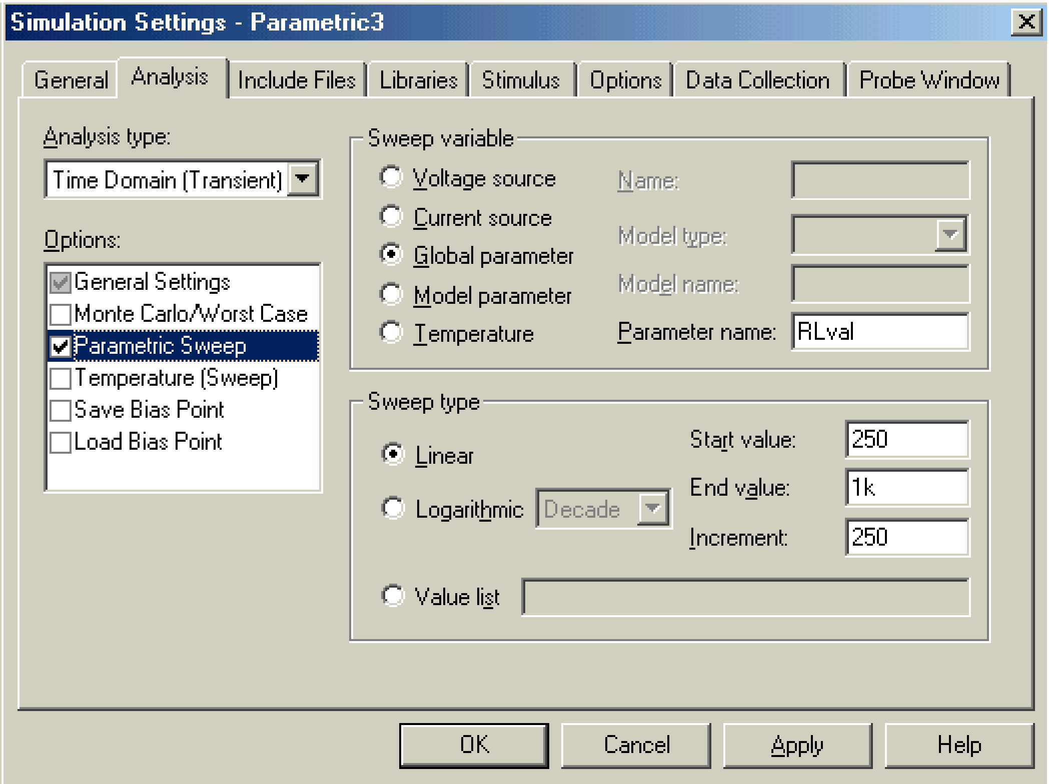


Figure 28. Window for the Simulation Settings of the Parametric Sweep.

<u>c. Run Spice and Display the waveforms.</u>

a. Run PSpice

b. When the simulation is finished the Probe window will open and display a pop up box with the Available Selection. Select ALL and OK.

c. The multiple traces will show, as given in Figure 29.

d. You can use the cursors to determined specific valueson the traces; you can also adjust the axis by double-clicking on the Y and X axes.

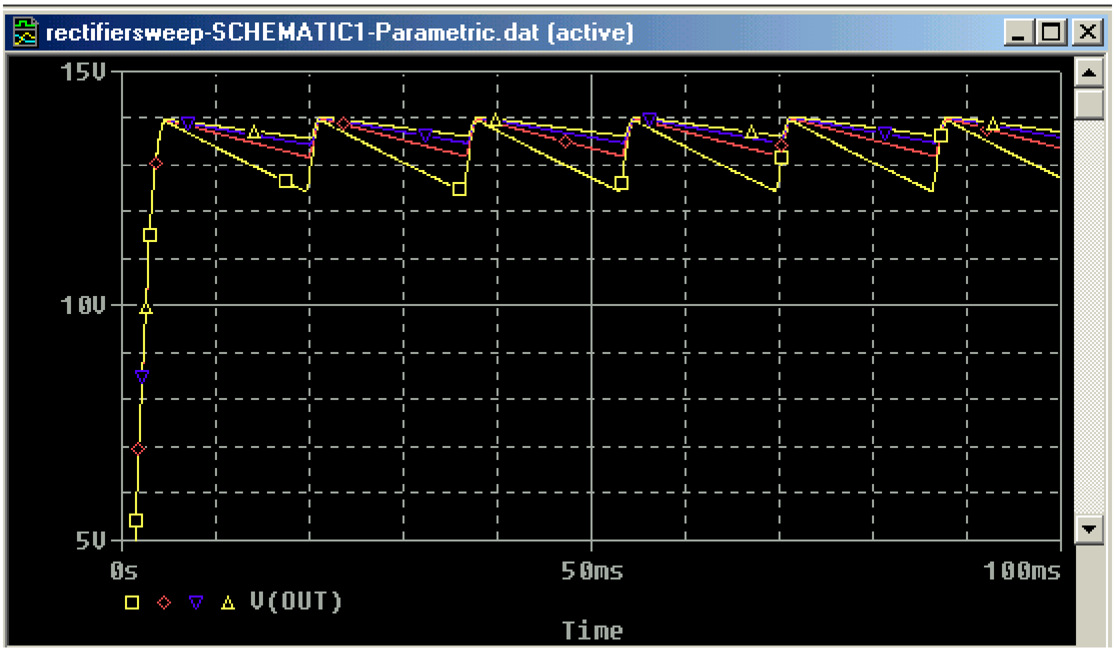


Figure 29. Results of the parametric sweep of the load resistor, varying from 250 to 1000 Ohm in steps of 250 Ohm.

AM Modulated Signal (AM Modulation)

An Amplitude modulated (AM) signal has the expression,

$v_{am}(t) = [(A + V_m \cos(2\pi f_m t)] \cos(2\pi f_c t) = A[1 + m \cos(2\pi f_m t)] \cos(2\pi f_c t)$

in which a sinusoidal high frequency carrier waveform $\cos(2\pi f_c t)$ is modulated by a sinusoidal modulating of frequency f_m. The modulating frequency can be any signal. For this example we'll assume it is a sinusoid. The modulation index is called *m*.

To generate a AM signal in PSpice we can make use of the Multiplication function MULT that can be found in the ABM library. Figure 30 shows the schematic that generates the AM signal over the resistor R1.

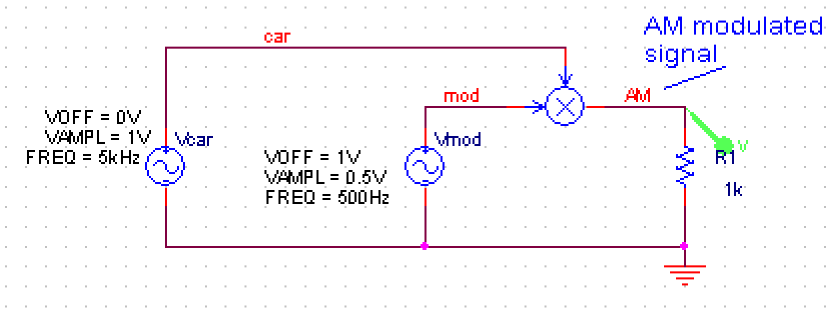


Figure 30. Schematic for the generation of an AM signal

The result of a transient simulation is shown in the figure below. One can also look at the Fourier of the simulated output signal. In the Probe window click on the FFT icon, located on the top toolbar, or go to the PSPICE/FOURIER menu. The Fourier spectrum of the displayed trace will be shown. You can change the X axis by double-clicking on it. Figure 32. gives the Fourier spectrum with the main peak corresponding to the carrier frequency of 5kHz and two side peaks at 4.5 and 5.5 kHz, indicating that the modulating frequency is 500Hz. You can use the cursors to get accurate readings.

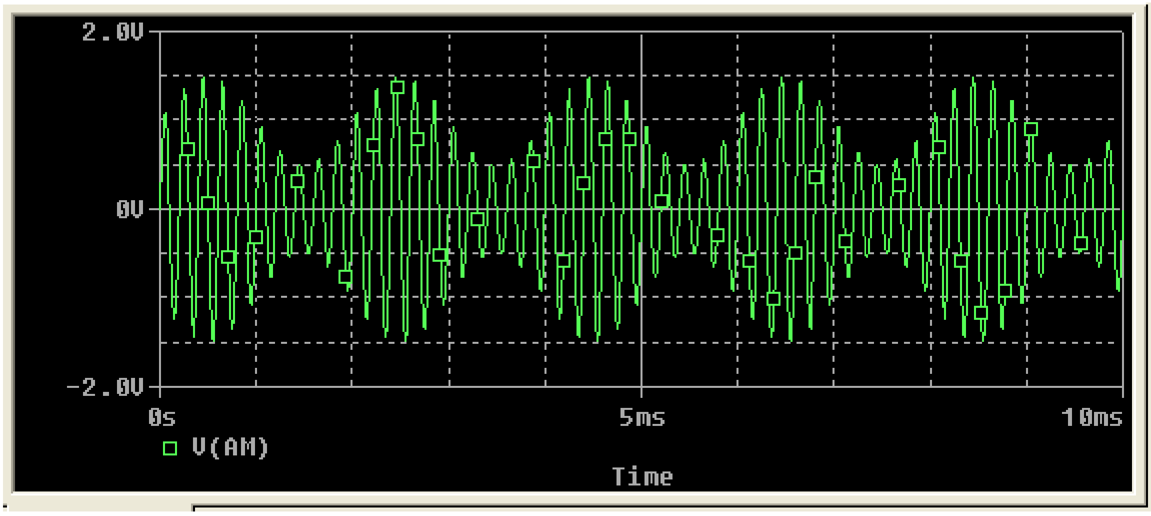


Figure 31. Simulated waveform (transient analysis) of the circuit above, with (A=1V, f_m=500 Hz, f_c=5kHz and m=0.5)

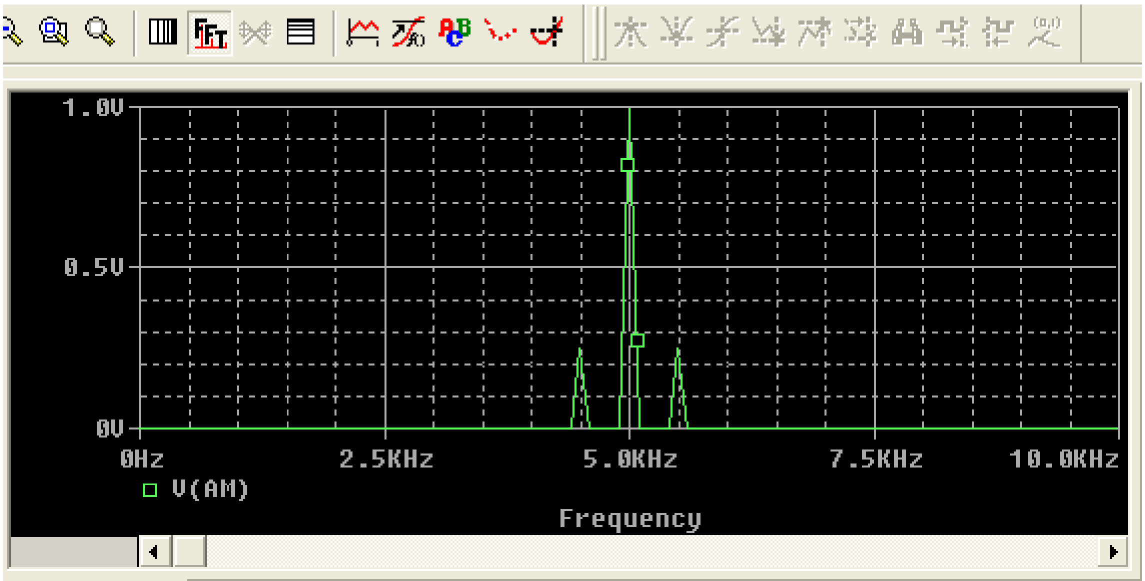


Figure 32. Fourier spectrum of the waveform of Figure 31.

Center Tap Transformer

There is no direct model in PSpice for a center tap transformer. However, one can use mutually coupled inductors to simulate a center tap transformer. Figure 33 shows the schematic of the circuit. We used one primary inductor L1 and two secondary inductors L1 and L2 put in series. In addition we added a K-Linear element.

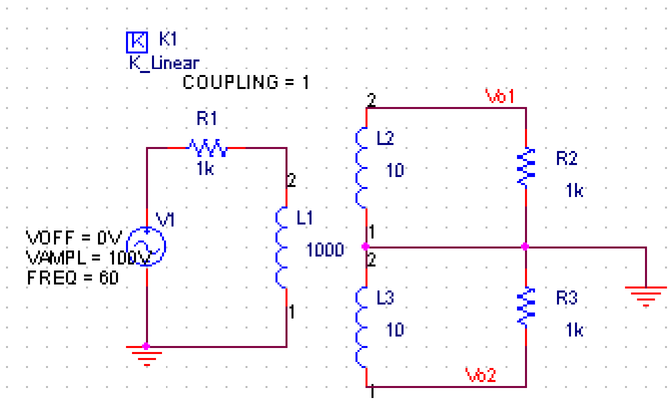


Figure 33. Circuit with Center Tap Transformer with a ratio of 10:1.

After placing the element on the schematic give each element its value. Use for the input voltage a sinusoid with amplitude of 100 V and frequency 60 Hz. Notice that we added a small resistor R1 in series with the voltage source and the inductor. This was needed to prevent a short circuit in DC (Spice would give en error without this resistor). We have kept it small equal to 1 Ohm. Assume that we want to have a step-down transformer with a ratio of 10:1 to each secondary output. The ratios of the inductors L2/L1 and L3/L1 must then be equal to $1/10^2$ (or =sqrt(L2/L1)=0.1). We made L1=1000 and L2-L3=10H.

Double-click on the K-Linear element and type under the column headings for L1, L2, L3, the values LP, Ls1, Ls2. When done, click the APPLY button and close the properties window. Go to PSpice/CREATE_NETLIST to generate the netlist. To see the list, go to the Project Manager and double-click on OUTPUTs: name.net file. The netlist looks as follows:

```
* source CENTERTAPTRANSFOR2
Kn_K1          L_Lp       L_Ls1       L_Ls2       1
L_Lp           0 N00241   1000
L_Ls1          0     VO1  10
L_Ls2          VO2   0    10
V_V1           N00203     0
+SIN 0V 100V  60   0  0    0
R_R1           N00203     N00241   1k
```

R_R2 0 VO1 1k

R_R3 VO2 0 1k

Create a new Simulation Profile (Transient) with " Time to run = 50ms". The result is shown in Figure 34. Notice that the max output is 10V as one would expect from a transformer ratio of 10:1 with an input voltage of 100Vmax.. The two outputs are 180 degrees out of phase.

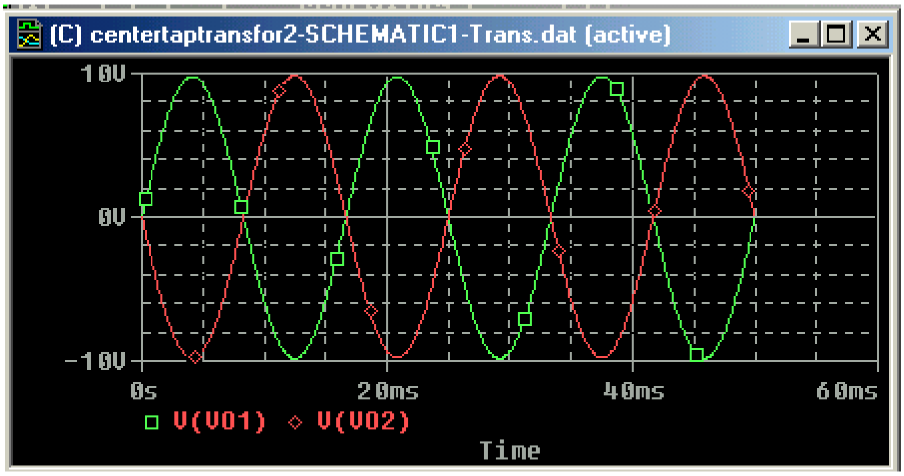


Figure 34. Output of the circuit of Figure 33.

4. Adding and Creating Libraries: Model and Parts files

4.1 Using and Adding Vendor Libraries

We assume that the model (.lib) as well as the Part Symbols files (.olb) is available from the vendor. In case only the model file is available see the next section on how to create a Part Symbol. In some cases you may want to add model libraries and symbols from vendors that contain the devices you want to use in your design. The ORCAD PSpice website list many vender-contributed models. You can download these files. You will need both the model definition file (with extension .lib) and the symbol file (extension .olb). When entering the symbols in the schematic you will need to "add the library". You need also to tell the simulator that the file exists. You do this in the Schematics when defining the Simulation Profile:

In the Simulation Setting window, select the Libraries tab. In the Filename box, enter the name of the new library (the full path name or the library name if it is located in the same folder as the standard libraries). You can make the library modesl global so that it will be available for every schematic or you can keep it local (for the current schematic only). Figure 4.1 shows who we added the library nat_semi-.lib as a global library (click on the Add as Global).

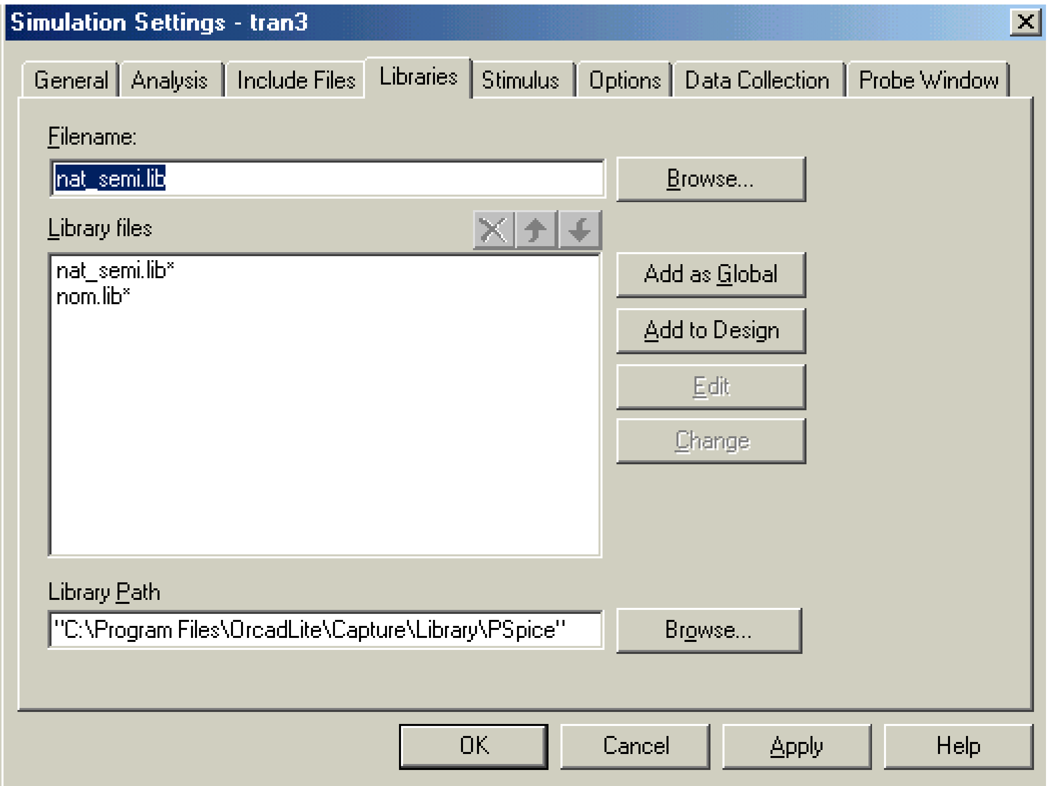


Figure 35. Adding a library

4.2 Creating Symbol Parts file from a Model file

In many cases you may have the models of devices available but not the Part Symbol that is used in PSpice Capture. In this case you need to create the symbol file (.olb). In many cases you will have a model file that contains models of many devices including subcircuits. This section describes how to use the model file to create a Part Symbol for the corresponding devices in the model file . The model file is a text file that can be read using a text editor (.e.g Notepad). In many cases existing vendor Spice files will have the extension .cir or .mod. We assume that you have such a file available but not the Part Symbol.

Open the PSpice Model Editor (this program came with the PSpice package).

a. Under the FILE Menu, select NEW

b. Next, under the MODEL menu, select IMPORT and find the model file for which you need to create the Part Symbol file. This will open the model file.

c. Save this file with the extension .lib and put it in a directory where you store the library files (you can put it in any directory; the default libraries are stored in Program Files/OrCadLite/Capture/Library/PSpice/).

d. The next step is to create the Parts for Capture. While the model file (.lib) is still open, go to FILE/CREATE_CAPTURE_FILE menu. A window (Create parts for Library) will pop us as shown below. Click on the top Browse Button and find the location of the model library is stored (.lib). This will automatically fill the Output Part Library entry with the same file name as the model library but with the .olb extension.

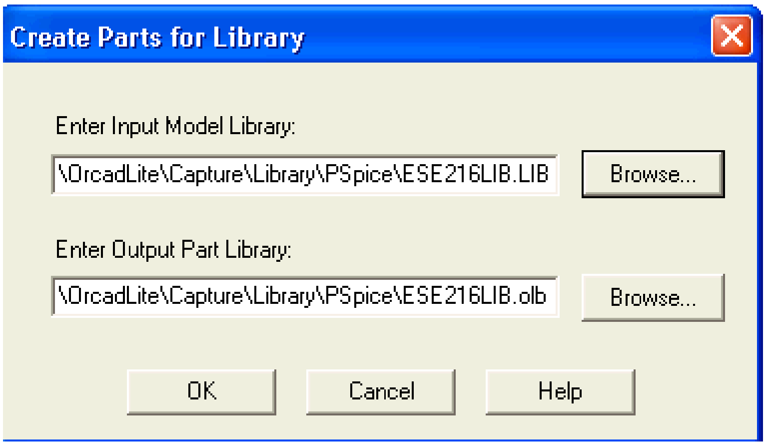


Figure 36. Create Parts for Library window. In this example we created a Model Library called ESE216LIB.lib and Parts ESE216LIB.olb

e. Click the OK button. A window will open, giving the status of the library creation. This should give you no errors.

f. Click OK in the Status window.

The next step is to edit the Part Symbols that you just created.

Editing the Part Symbol

a. Open OrCad Capture.

b. Go to the FILE/OPEN/LIBRARY menu. Browse for the location of the newly created file (e.g. ESE216LIB.OLB). Click OK. This will open the PCB window fpr the library, as shown below. In our example, our library contains two devices (NMOS and PMOS devices). In practical cases the library can contain many different devices and subcircuits. An example is the sedra_lib.lib and sedra_lib.olb that comes with the textbook [10], shown in Figure 38.

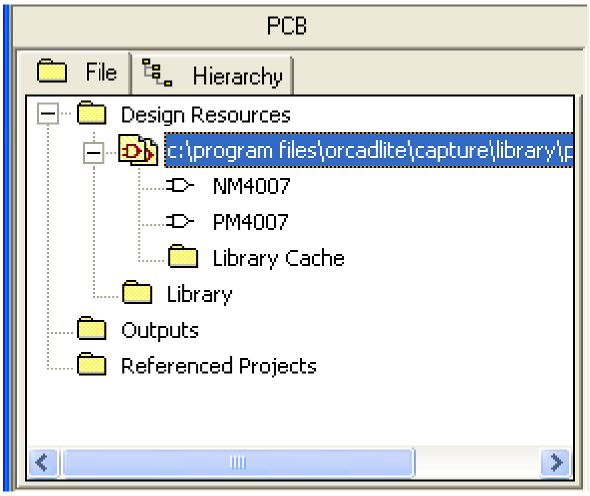


Figure 37. Library Editor Window in OrCad Capture

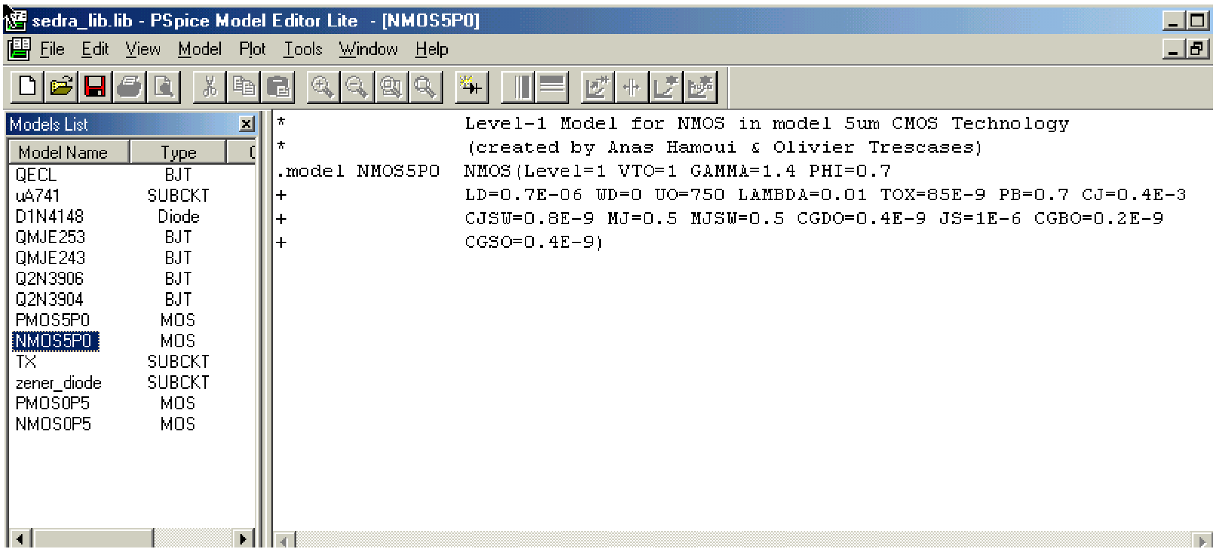


Figure 38. Library sedra_lib.lib showing the various devices in the library file. The left window pane list all the devices and subcircuits. The right pane shows the model of the NMOS5PO (highlighted on the left).

c. To edit the symbol of any of the devices double click on it in the Library Editor window. Lets select e.g. the NMOS5PO device. This will open the Part Symbol window, as shown in Figure 39. OrCad Capture is smart enough to know when a model corresponds to a transistor and will create a transistor model, as shown in Figure 39. However for subcircuits is will usually give you a generic box. You can than modify this box using the editing tools of Capture.

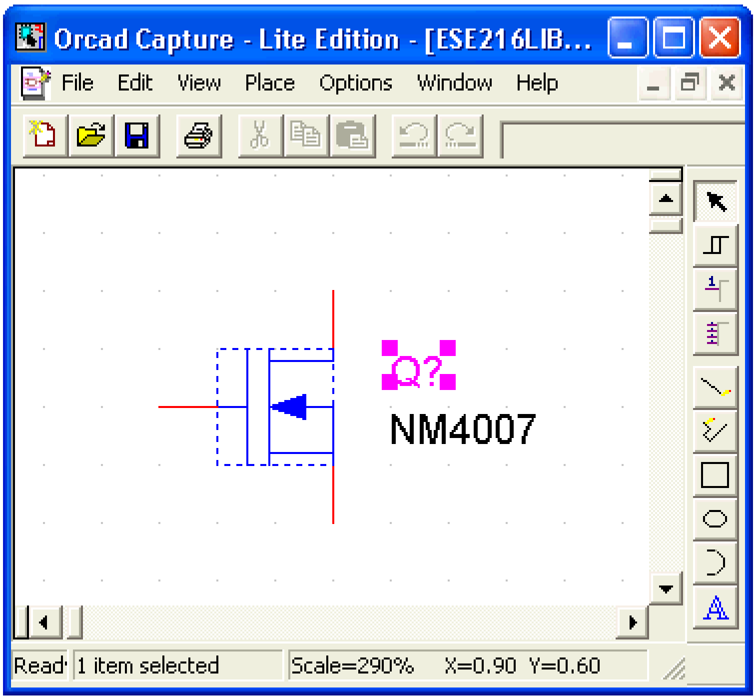


Figure 39. Part Symbol Window allowing you to edit the part. The example shown here is a NMOS symbol without a separate bulk contact. In that case the source and bulk are automatically shorted together.

d. The red line on the Parts symbol correspond to pins. These can be added by clicking on the "Place Pin" icon on the right side menu bar (or PLACE/PIN menu). This will open the Place Pin window shown below.

e. You can edit the pins by first selecting it and then right clicking and selecting Edit Properties. The pin name and type are important. In general you should not change the pin names since these relate back to the Spice model. The pin type is usually "Input" or "Output." If you make a pin a "Power" type, it will be invisible in the part symbol. For shape you can select "Line" or "Short" which corresponds to a short line. Check out the other options. In case you create a symbol for a subcircuit you can give the pin numbers that correspond to those of the datasheet.

f. When the part symbol is finished, save the library. You are now ready to use your newly created library and symbols. Before doing a simulation you need to add the library to the library path, in both the schematic and the simulator setting. See section on "Adding Vendor Libraries" above.

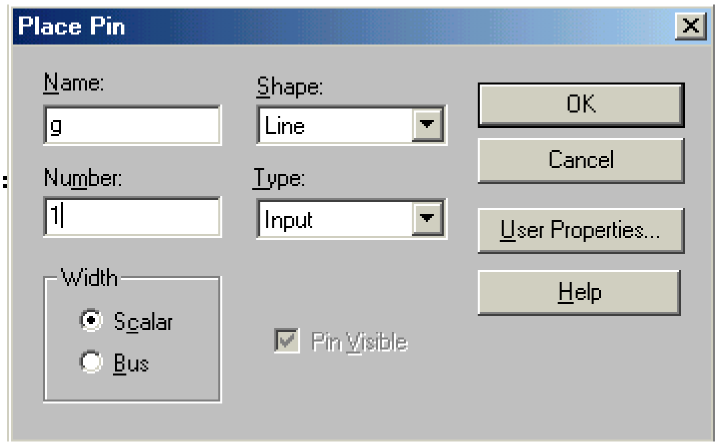
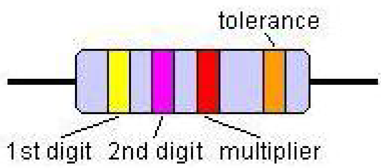


Figure 40. Place Pin window.

RESISTOR COLOR CODE

The resistor color code is a way of showing the value of a resistor. Instead of writing the resistance on its body, which would often be too small to read, a color code is used. Different colors represent the numbers 0 to 9. The first two colored bands on the body are the first two digits of the resistance, and the third band is the 'multiplier'. Multiplier just means the number of zeroes to add after the first two digits. Red represents the number 2, so a resistor with red, red, red bands has a resistance of 2 followed by 2 followed by 2 zeroes, which is 2200 ohms or 2.2 kilo Ohms. The final band is the tolerance (the accuracy ± x %). All resistors have a tolerance which is shown by the last band.



Color	1st Band	2nd Band	3rd Band	4th Band
Black	0	0	1	
Brown	1	1	10	
Red	2	2	100	
Orange	3	3	1000	
Yellow	4	4	10000	

Green	5	5	100000	
Blue	6	6	1000000	
Purple	7	7		
Grey	8	8		
White	9	9		
Red				1%
Gold				5%
Silver				10%

Examples:

*Yellow, Purple, Red, Gold = 47 x 100 = 4 700 Ω= 4.7 kΩ + 5%

*Brown, Black, Yellow, Gold = 10 x 10 000 = 100 kΩ + 5%

*Yellow, Purple, Black, Silver = 47 x 1 = 47 Ω + 10%

*Brown, Black, Red, Red = 10 x 100 = 1 000 Ω = 1 kΩ + 1%

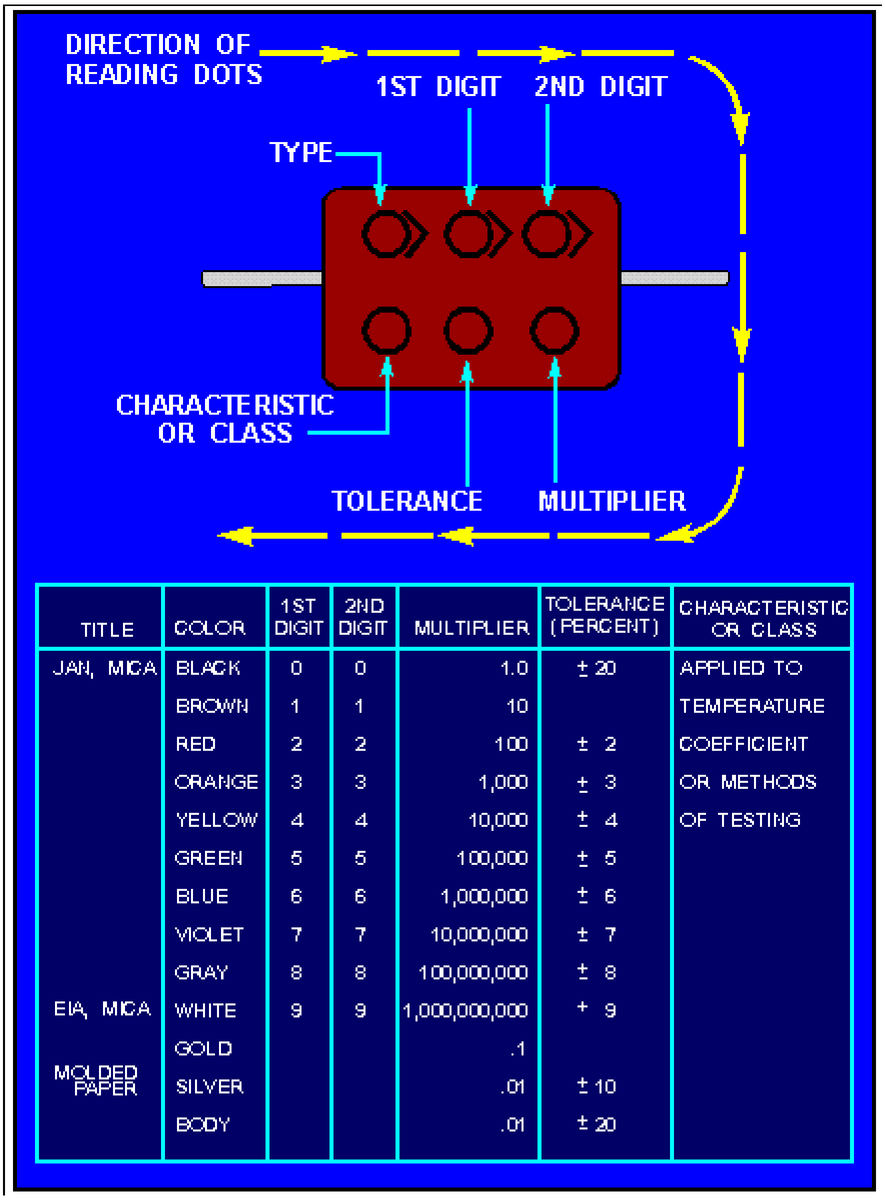
*Brown, Black, Green, Gold = 10 x 100 000 = 1 000 kΩ = 1 MΩ + 5%

COLOR CODES FOR CAPACITORS

Although the capacitance value may be printed on the body of a capacitor, it may also be indicated by a color code. The color code used to represent capacitance values is similar to that used to represent resistance values. The color codes currently in use are the Joint Army-Navy (JAN) code and the Radio Manufacturers' Association (RMA) code.

For each of these codes, colored dots or bands are used to indicate the value of the capacitor. A mica capacitor, it should be noted, may be marked with either three dots or six dots. Both the three- and the six-dot codes are similar, but the six-dot code contains more information about electrical ratings of the capacitor, such as working voltage and temperature coefficient.

The capacitor shown in figure 42 represents either a mica capacitor or a molded paper capacitor. To determine the type and value of the capacitor, hold the capacitor so that the three arrows point left to right (>). The first dot at the base of the arrow sequence (the left-most dot) represents the capacitor TYPE. This dot is either black, white, silver, or the same color as the capacitor body. Mica is represented by a black or white dot and paper by a silver dot or dot having the same color as the body of the capacitor. The two dots to the immediate right of the type dot indicate the first and second digits of the capacitance value. The dot at the bottom right represents the multiplier to be used. The multiplier represents picofarads. The dot in the bottom center indicates the tolerance value of the capacitor.



TITLE	COLOR	1ST DIGIT	2ND DIGIT	MULTIPLIER	TOLERANCE (PERCENT)	CHARACTERISTIC OR CLASS
JAN, MICA	BLACK	0	0	1.0	± 20	APPLIED TO
	BROWN	1	1	10		TEMPERATURE
	RED	2	2	100	± 2	COEFFICIENT
	ORANGE	3	3	1,000	± 3	OR METHODS
	YELLOW	4	4	10,000	± 4	OF TESTING
	GREEN	5	5	100,000	± 5	
	BLUE	6	6	1,000,000	± 6	
	VIOLET	7	7	10,000,000	± 7	
	GRAY	8	8	100,000,000	± 8	
EIA, MICA	WHITE	9	9	1,000,000,000	+ 9	
	GOLD			.1		
MOLDED PAPER	SILVER			.01	± 10	
	BODY			.01	± 20	

Figure 41. 6-dot color code for mica and molded paper capacitors.

Example of mica capacitors

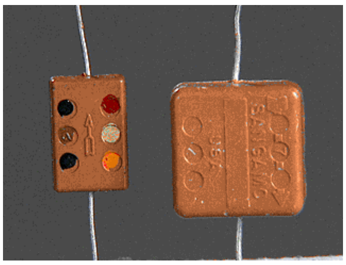


Figure 42. Mica capacitor

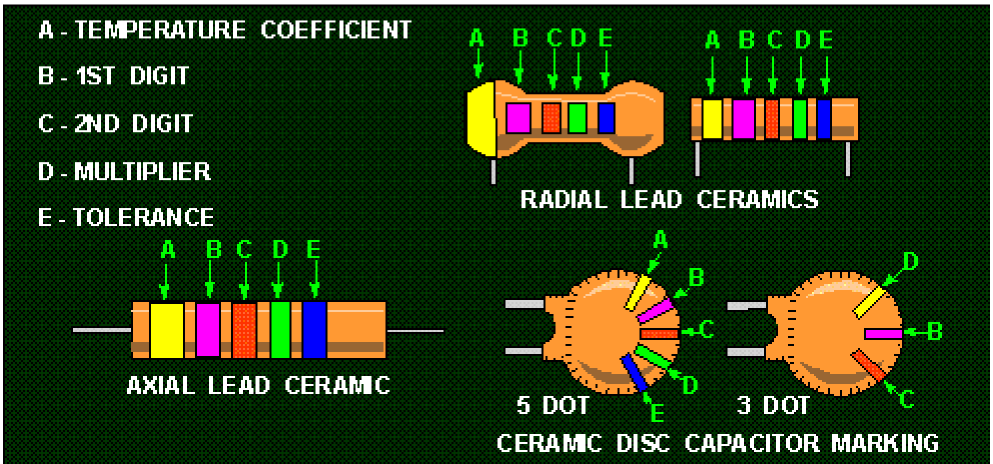
Because this type of tubular capacitor always has a paper dielectric, the type code is omitted. To read the code, hold the capacitor so the band closest to the end is on the left side; then read left to right. The last two bands (the fifth and sixth bands from the left) represent the voltage rating of the capacitor. This means that if a capacitor is coded red, red, red, yellow, yellow, yellow, it has the following digit values:

red=2
red=2
red=X 100 pF
yellow=±40%
yellow=4
yellow=4

Figure 43. 6-band color code for tubular paper dielectric capacitors.

The six digits indicate a capacitance of 2200 pF with a ±40 percent tolerance and a working voltage of 44 volts. The ceramic capacitor is color coded as shown in figure 3-23 and the mica capacitor as shown in figure 3-24. Notice that this type of mica capacitor differs from the one shown in figure 3-21 in that the arrow is solid instead of broken. This type of mica capacitor is read in the same manner as the one shown in figure 3-21, with one exception: the first dot indicates the first digit. (Note: Because this type of capacitor is always mica, there is no need for a type dot.)

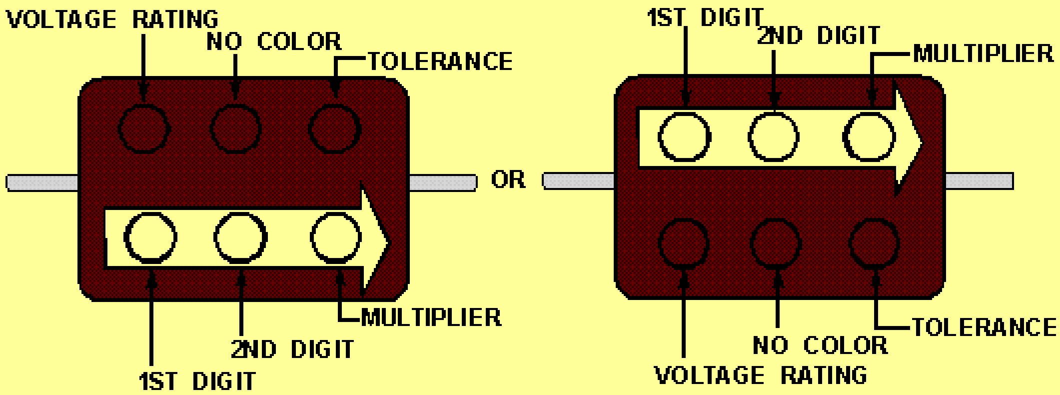
Figure 44. Ceramic capacitor color code.



COLOR	1ST DIGIT	2ND DIGIT	MULTIPLIER	TOLERANCE MORE THAN 10 pf (IN PERCENT)	TOLERANCE LESS THAN 10 pf (IN pf)	TEMPERATURE COEFFICIENT*
BLACK	0	0	1.0	±20	±2.0	0
BROWN	1	1	10	± 1		-30
RED	2	2	100	± 2		-80
ORANGE	3	3	1,000			-150
YELLOW	4	4	10,000			-220
GREEN	5	5		± 5	±0.5	-330
BLUE	6	6				-470
VIOLET	7	7				-750
GRAY	8	8	.01		±0.25	+30
WHITE	9	9	.1	±10	±1.0	+120 TO -750 (EIA)
						+500 TO -330 (JAN)
SILVER						+100 (JAN)
GOLD						BYPASS OR COUPLING

*PARTS PER MILLION PER DEGREE CENTIGRADE.

Figure 45. Mica capacitor color code.

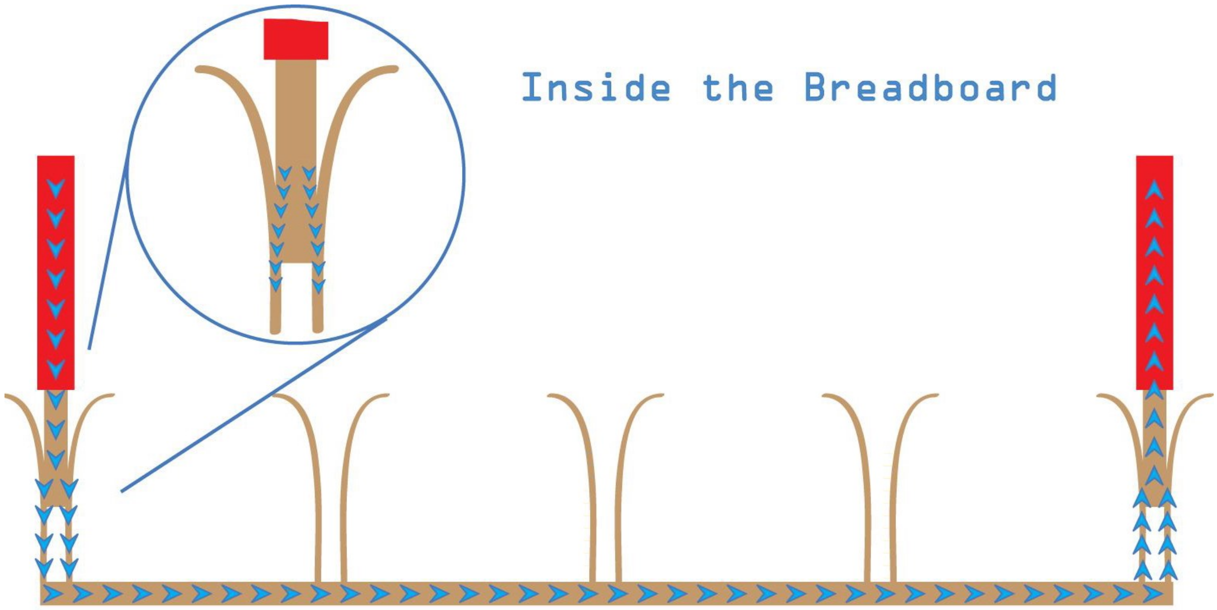


COLOR	1ST DIGIT	2ND DIGIT	MULTIPLIER	TOLERANCE (PERCENT)	VOLTAGE RATING
BLACK	0	0	1.0	± 1	100
BROWN	1	1	10	± 2	200
RED	2	2	100	± 3	300
ORANGE	3	3	1,000	± 4	400
YELLOW	4	4	10,000	± 5	500
GREEN	5	5	100,000	± 6	600
BLUE	6	6	1,000,000	± 7	700
VIOLET	7	7	10,000,000	± 8	800
GRAY	8	8	100,000,000	± 9	900
WHITE	9	9	1,000,000,000		1000
GOLD			.1	± 10	2000
SILVER			.01	± 20	
BODY					

*WHERE NO COLOR IS INDICATED, THE VOLTAGE RATING MAY BE AS LOW AS 300 VOLTS.

BASIC BREADBOARD STRUCTURE

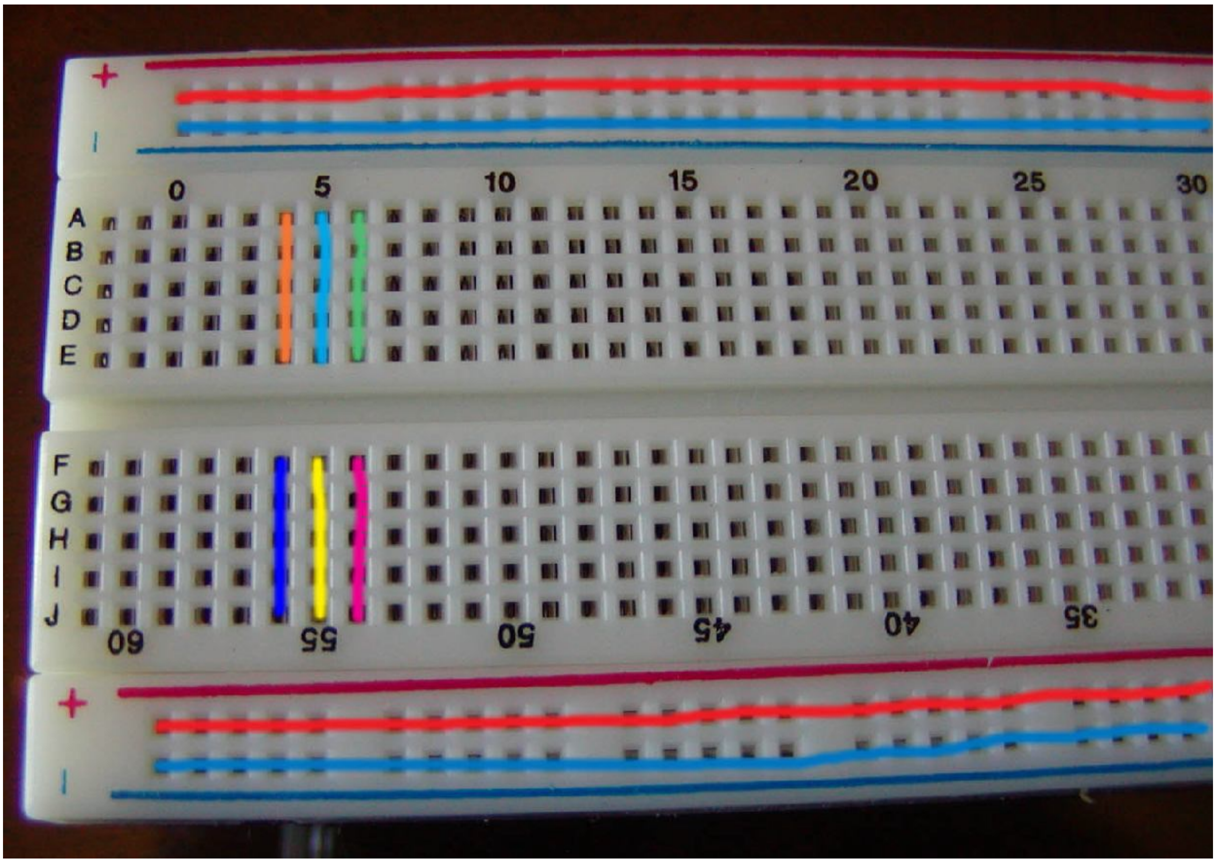
The wondrously useful breadboard is actually nothing more than several side by side rows of thin strips of metal surrounded by plastic so they are electrically insulated from each other. As it is known that electricity can flow easily through **conductors** like metal but has a hard time going through **insulators** like rubber and plastic. When two pieces of conductor material touch against each other, electricity can flow from one to the other. The small holes in the breadboard are called “Tie Points.” Under each of the tie point holes is a strip of metal that has a pair of springy flexible finger shaped pieces that squeeze against the sides of the wire when it's stuck down inside the hole. Each vertical metal strip runs underneath 5 tie point holes inside the breadboard. If two different wires are plugged into any 2 of the 5 tie points on the same metal strip the wires become electrically connected together as shown in the picture below. The blue arrows in the diagram show how the electricity can be conducted from the wire on the left over to the wire on the right by going through the strip of metal inside the breadboard. Since there is metal to metal contact, electricity can jump from the metal inside the wire on the left into the metal pincher fingers as shown in the up close view circle. It then flows on the metal strip inside the breadboard, up the fingers in the tie point on the right, jumps onto the metal part of the wire on the right and off it goes to complete the circuit back to the source...

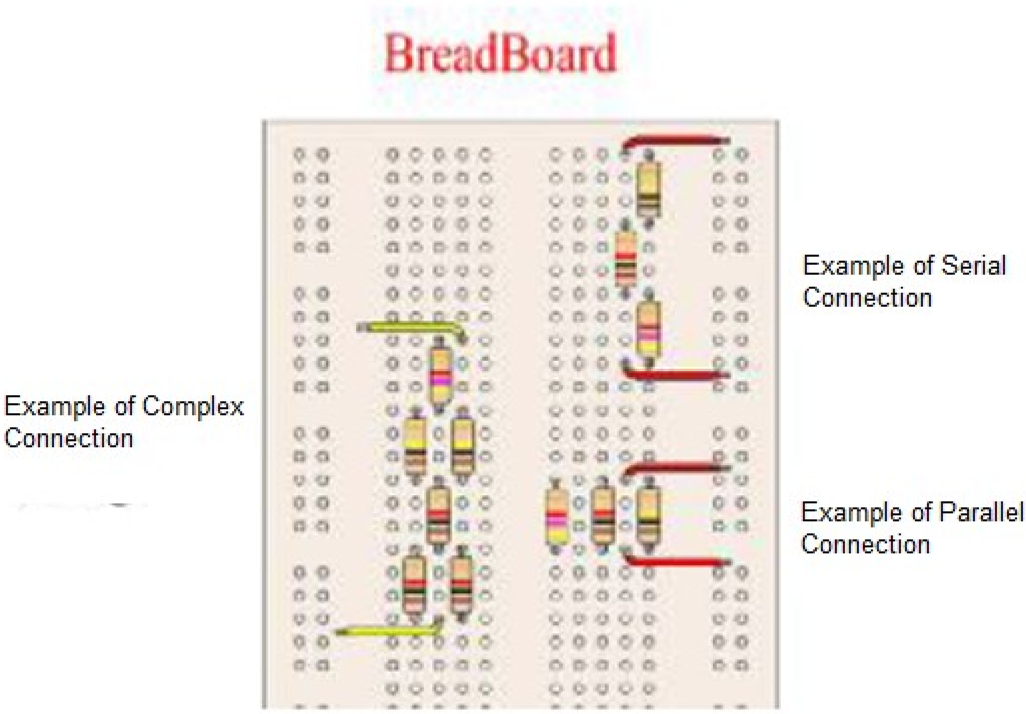


Which holes are connected together?

The photo below shows a close-up picture of a breadboard with some colored lines drawn on it to show which holes are electrically connected to each other because they are on the same metal strip together. The main center area of the board is made up of lots of vertical metal strips with 5 tie points each. The orange line in the picture shows that tie points (holes) A4,B4,C4,D4 and E4 are all electrically connected together. Notice that the points on the orange line are NOT connected to their next door neighbors on the light blue line of 5 points (A5,B5,C5,D5,E5). And that they do not connect across the gap in the center of the board. The dark blue line (F56,G56,H56,I56,J56) of connected points does not connect up with the points in the orange line on the other side of the center gap. The center gap is where the Integrated Circuit (IC) chips plug in as you'll see later on.

Above and below the main center area are the “power rails.” These are long horizontal strips of metal with lots of tie points on them. The red and blue horizontal lines on the picture show which holes are connected together.


0
5
10
15
20
25
30
A
B
C
D
E
F
G
H
I
J
60
55
50
45
40
35


BreadBoard
Example of Serial Connection
Example of Complex Connection
Example of Parallel Connection

NOTES ON OSCILLOSCOPES

Oscilloscope

In many applications, observing certain voltage waveforms in a circuit plays a crucial role in understanding the operation of the circuit. For that purpose several measurement instruments are used like voltmeter, ammeter, or the oscilloscope.

An oscilloscope (sometimes abbreviated as "scope") is a voltage sensing electronic instrument that is used to visualize certain voltage waveforms. An oscilloscope can display the variation of a voltage waveform in time on the oscilloscope's screen

Figure 46.

A probe is used to connect the oscilloscope to the circuit. Figure 46 shows an oscilloscope and a probe connected to it.

Figure 47.

Figure 47 shows a typical probe. Oscilloscope shows the potential difference between the two terminals of the probe. The terminal ending with a hook is usually connected to the node in the circuit whose voltage is of interest. The other terminal is usually (but not always) connected to the ground. The probes are attached to input channels of the oscilloscope. Most oscilloscopes have at least two input channels and each channel can display a waveform on the screen. Multiple channels are useful for comparing waveforms. For example, one can observe the voltage waveforms at the input and the output terminals of a circuit simultaneously, by using a two channel oscilloscope.

Analog and Digital

Electronic equipments can be divided into two types: analog and digital. *Analog* equipment works with continuously variable voltages, while *digital* equipment works with binary numbers (1 and 0's) that may represent voltage samples. For example, a conventional cassette player is an analog device; a compact disc player is a digital device.

Oscilloscopes also come in analog and digital types. An analog oscilloscope works by directly applying a voltage being measured to an electron beam moving across the oscilloscope screen. The voltage deflects the beam up and down proportionally, tracing the waveform on the screen. This gives an immediate picture of the waveform. In contrast, a digital oscilloscope samples the waveform and uses an analog-to-digital converter (or ADC) to convert the voltage being measured into digital information. It then uses this digital information to reconstruct the waveform on the screen.

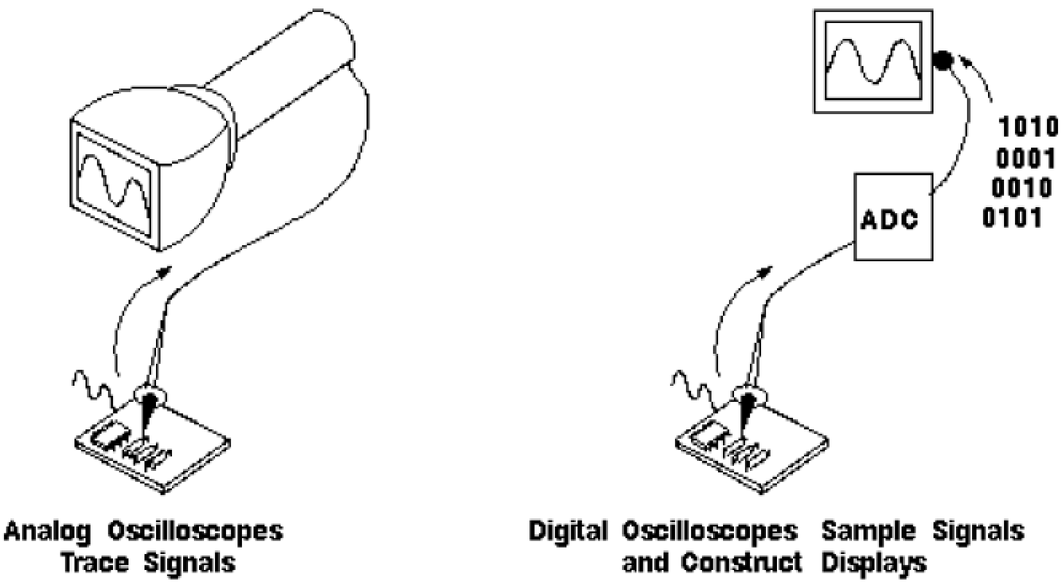


Figure 48. Digital and Analog Oscilloscopes Display Waveforms.

Analog Oscilloscopes

An analog oscilloscope displays the voltage waveforms by deflecting an electron beam generated by an electron gun inside a cathode-ray tube on to a fluorescent coating. Because of the use of the cathode ray tube, analog oscilloscopes are also known as cathode ray oscilloscopes. To understand how an analog scope displays the voltage waveforms, it is necessary to understand what is inside the unit. The following section describes the general principles of the operation of cathode ray oscilloscopes.

Cathode Ray Oscilloscope Principles

Figure 49 shows the structure, and the main components of a cathode ray tube (CRT). Figure 50 shows the face plane of the CRO screen.

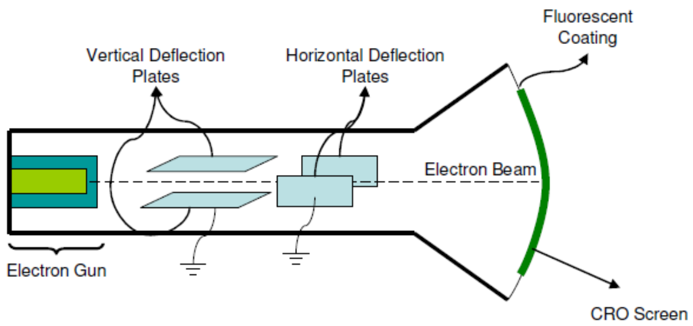


Figure 49.

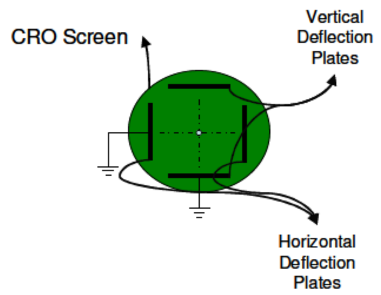


Figure 50.

Electron beam generated by the electron gun first deflected by the deflection plates, and then directed onto the fluorescent coating of the CRO screen, which produces a visible light spot on the face plane of the oscilloscope screen.

A detailed representation of a CRT is given in Figure 51. The CRT is composed of two main parts,

· Electron Gun

· Deflection System

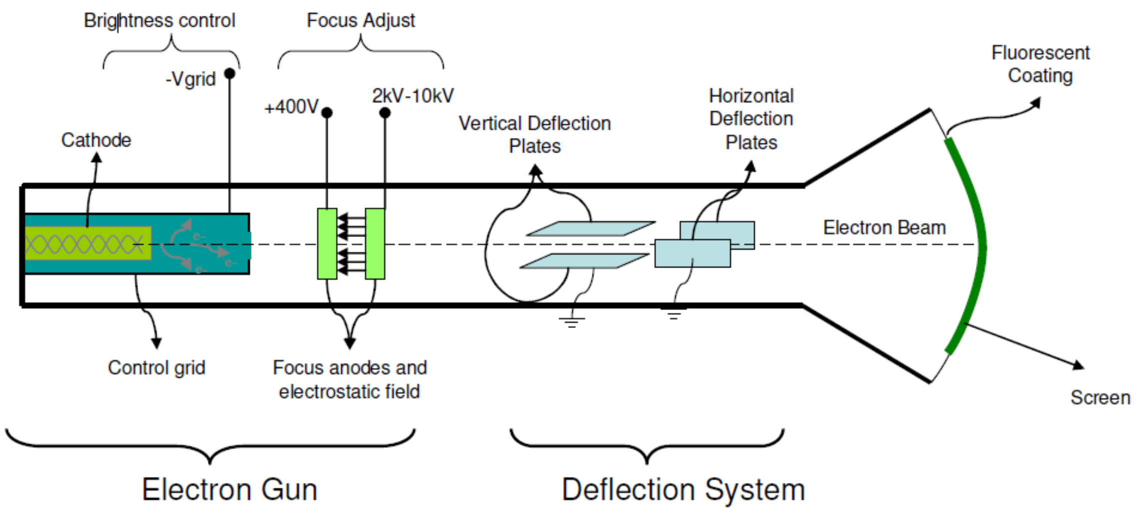


Figure 51.

Electron Gun

Electron gun provides a sharply focused electron beam directed toward the fluorescent-coated screen. The thermally heated cathode emits electrons in many directions. The control grid provides an axial direction for the electron beam and controls the number and speed of electrons in the beam.

The momentum of the electrons determines the intensity, or brightness, of the light emitted from the fluorescent coating due to the electron bombardment. Because electrons are negatively charged, a repulsion force is created by applying a negative voltage to the control grid, to adjust their number and speed. A more negative voltage results in less number of electrons in the beam and hence decreased brightness of the beam spot.

Since the electron beam consists of many electrons, the beam tends to diverge. This is because the similar (negative) charges on the electrons repulse each other. To compensate for such repulsion forces, an adjustable electrostatic field is created between two cylindrical anodes, called the focusing anodes. The variable positive voltage on the second anode cylinder is therefore used to adjust the focus or sharpness of the bright spot.

The Deflection System

The deflection system consists of two pairs of parallel plates, referred to as the vertical and horizontal deflection plates. One of the plates in each set is permanently connected to the ground (zero volt), whereas the other plate of each set is connected to input signals or triggering signal of the CRO.

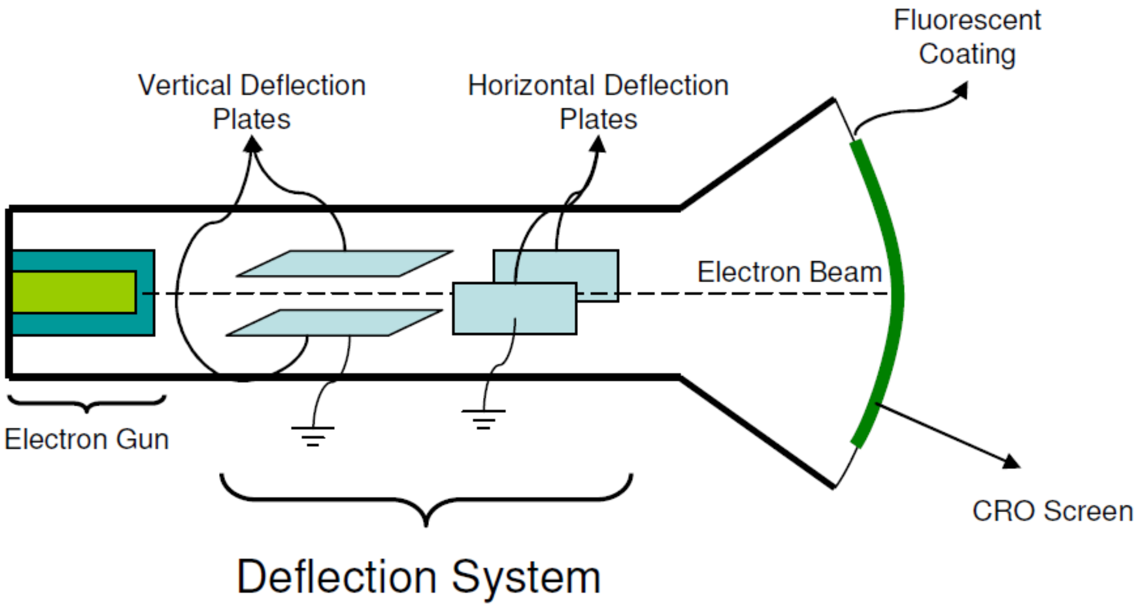


Figure 52.

As shown in Figure 52, the electron beam passes through the deflection plates. In reference to the schematic diagram in Figure 8, a positive voltage applied to the Y input terminal causes the electron beam to deflect vertically upward, due to attraction forces, while a negative voltage applied to the Y input terminal causes the electron beam to deflect vertically downward, due to repulsion forces. Similarly, a positive voltage applied to the X input terminal will cause the electron beam to deflect horizontally toward the right, while a negative voltage applied to the X input terminal will cause the electron beam to deflect horizontally toward the left of the screen.

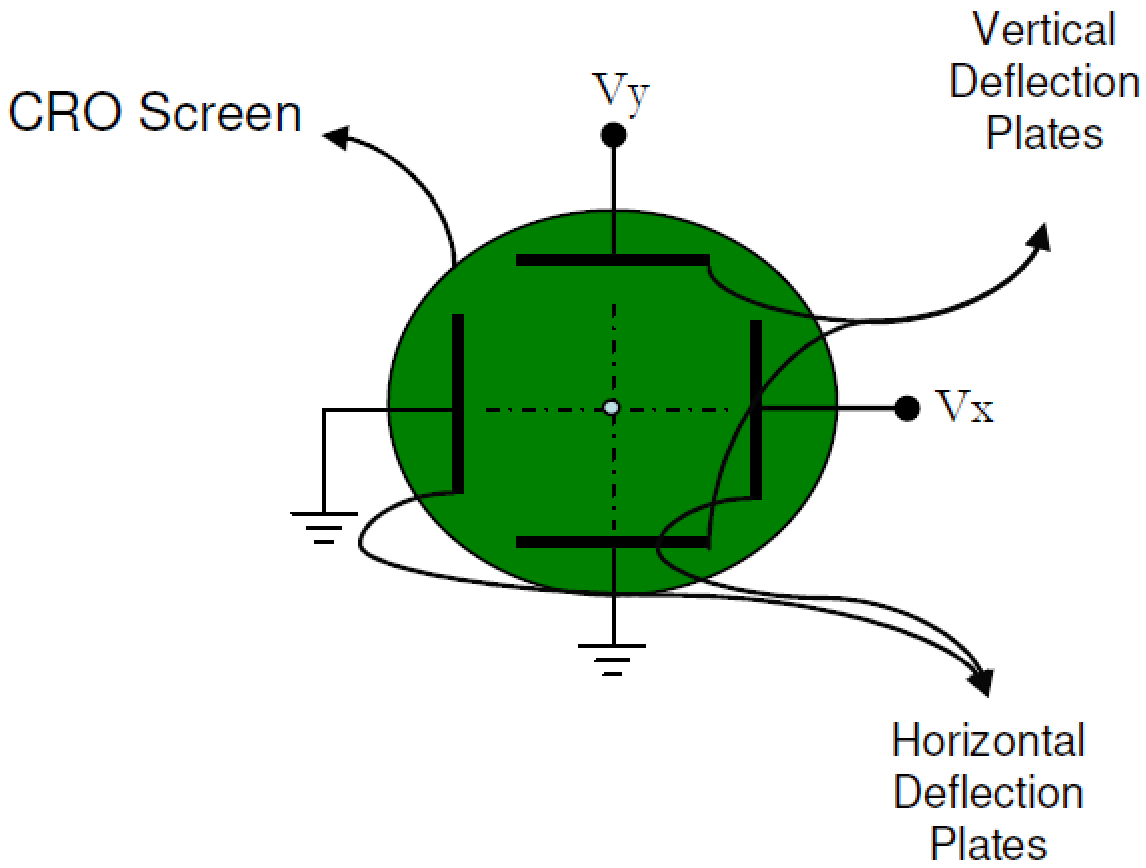


Figure 53.

The amount of vertical or horizontal deflection is directly proportional to the corresponding applied voltage. When the electrons hit the screen, the phosphor emits light and a visible light spot is seen on the screen.

Since the amount of deflection is proportional to the applied voltage, actually the voltages Vy and Vx determine the coordinates of the bright spot created by the electron beam.

Example 1:

Suppose $V_x = \sin(t)$, $V_y = \cos(t)$ are applied to the horizontal and vertical deflection plates respectively. Then the bright spot would follow a circular path on the CRO screen.

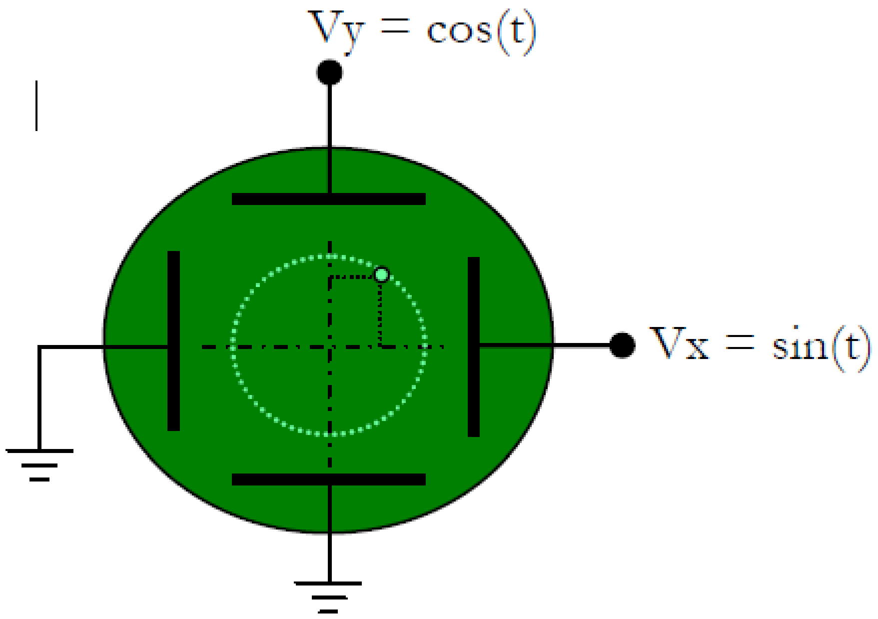


Figure 54.

Example 2:

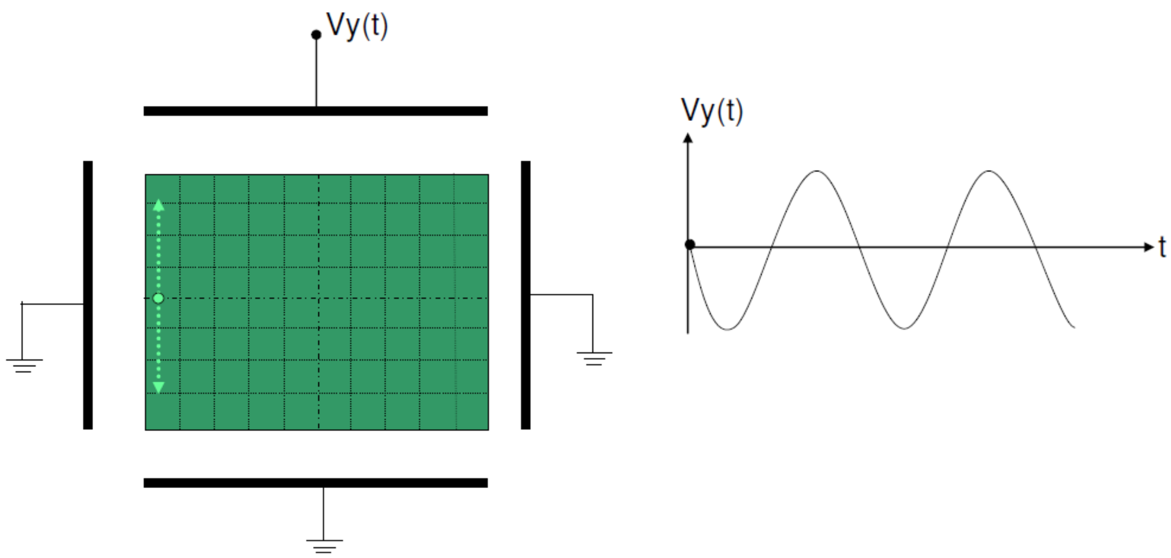


Figure 55-a.

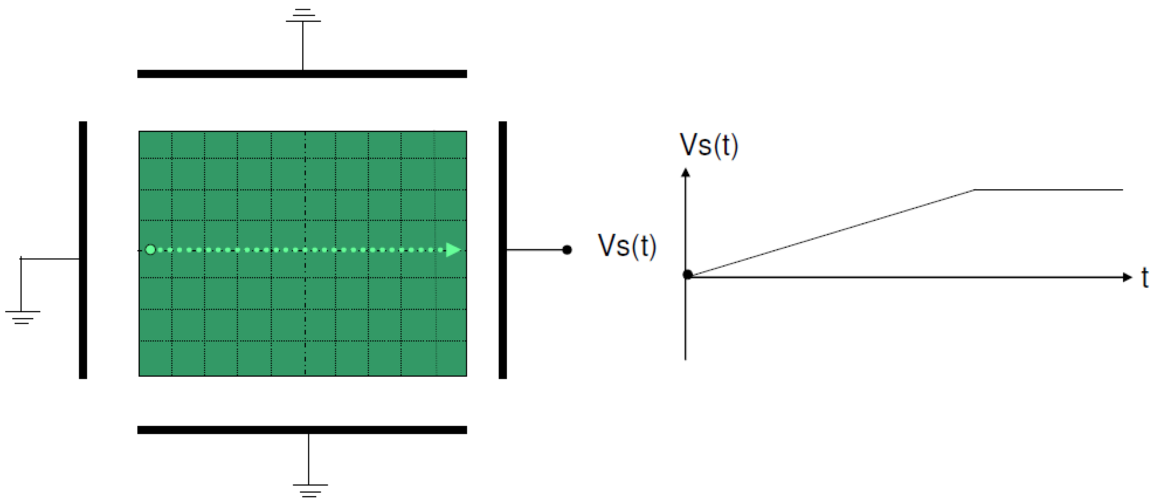


Figure 55-b.

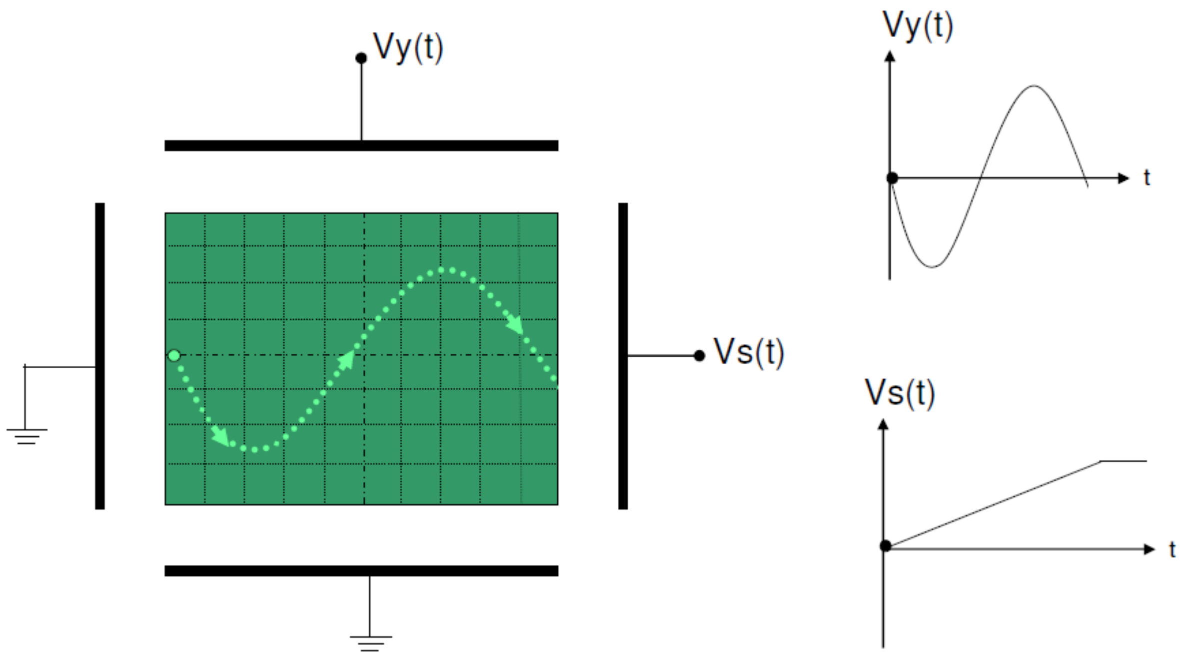


Figure 55-c.

In Figure 55-a, the input signal Vy(t) is applied to the vertical deflection plates, whereas the horizontal deflection plates are connected to ground. It is assumed that the electron beam is kept at the extreme left position when the horizontal deflection plates are connected to ground. Under this configuration, the bright spot in the CRO screen will follow a vertical path (will go up and down) at the extreme left position of the screen.

In Figure 55-b, the input signal Vs(t) is applied to the horizontal deflection plates, whereas the vertical deflection plates are connected to ground. This time, the bright spot will travel from extreme left to extreme right end of the screen and will stop there.

In Figure 55-c, the signals Vy(t) and Vs(t) are applied to the vertical and the horizontal deflection plates respectively. This time the bright spot will follow a sinusoidal path, resulting a visualization of the input signal Vy(t) on the CRO screen. Actually the bright spot must follow the same path fast and repetitively (at least 30 times in a second) so that the human eye can perceive the motion of the bright spot as a continuous curve. Therefore, in order to display the waveform on the CRO screen for the example in figure 10-c, the signals Vy(t) and Vs(t) should be applied to the vertical and the horizontal deflection plates periodically and in synchronization. The next section discusses the details of this procedure and depicts how CRO handles this problem.

Displaying a Voltage Waveform

In numerous applications it will be required to display a periodical voltage waveform as a function of time. By applying the voltage to be displayed on the CRO, to the vertical deflection plates (Vy), the vertical deflection of the beam spot will be proportional to the magnitude of this voltage. It is then necessary to convert the x axis (horizontal deflection) into
a time axis. Notice that, in the example given in figure 55-c, the voltage waveform Vs(t) (which varies linearly in time before the bright spot reaches the extreme right end of the screen) is used for this purpose and the bright spot have traveled the path determined by Vy(t).

If the signal to be observed is periodic, then a periodic voltage waveform that varies linearly with time, as shown in figure below, is applied to the horizontal deflection plates. This type of waveform is called the sawtooth waveform.

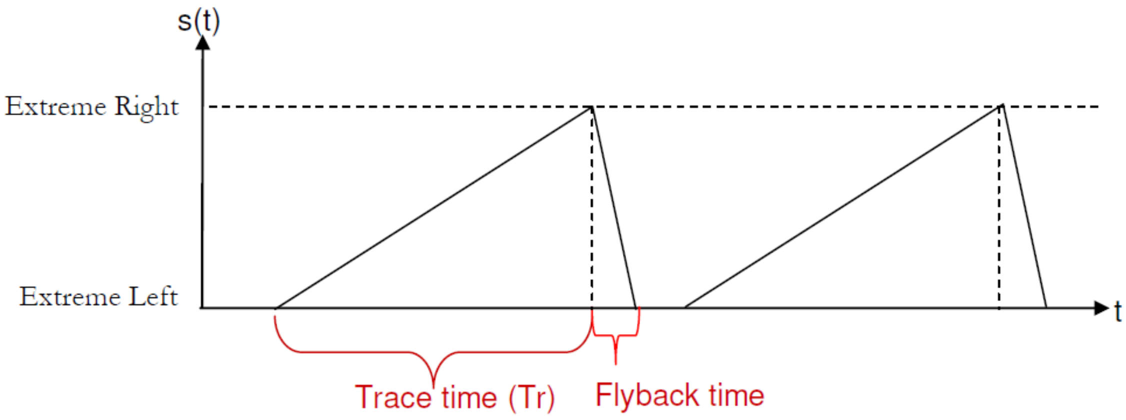


Figure 56.

When s(t) is zero volt, the bright spot is at the extreme left-hand position, and when s(t) is maximum, the bright spot is at the extreme right position. Therefore, the bright spot travels from extreme left to extreme right in a time equal to the trace time. During the flyback time, which is usually very short compared to trace time, a high negative voltage pulse is applied to the control grid of the electron gun to prevent electron beams reaching the CRO screen. This action is called blanking and prevents any reverse retrace (or shadow) as the beam is going back to the extreme left-hand position. The time period including the trace time and the flyback time is called the sweep period.

The period of the sawtooth waveform plays a crucial role in obtaining a steady waveform on the CRO screen. The following section discusses the requirements on the period of the sawtooth waveform and the need of a synchronization between the sawtooth waveform and the input waveform.

Triggering

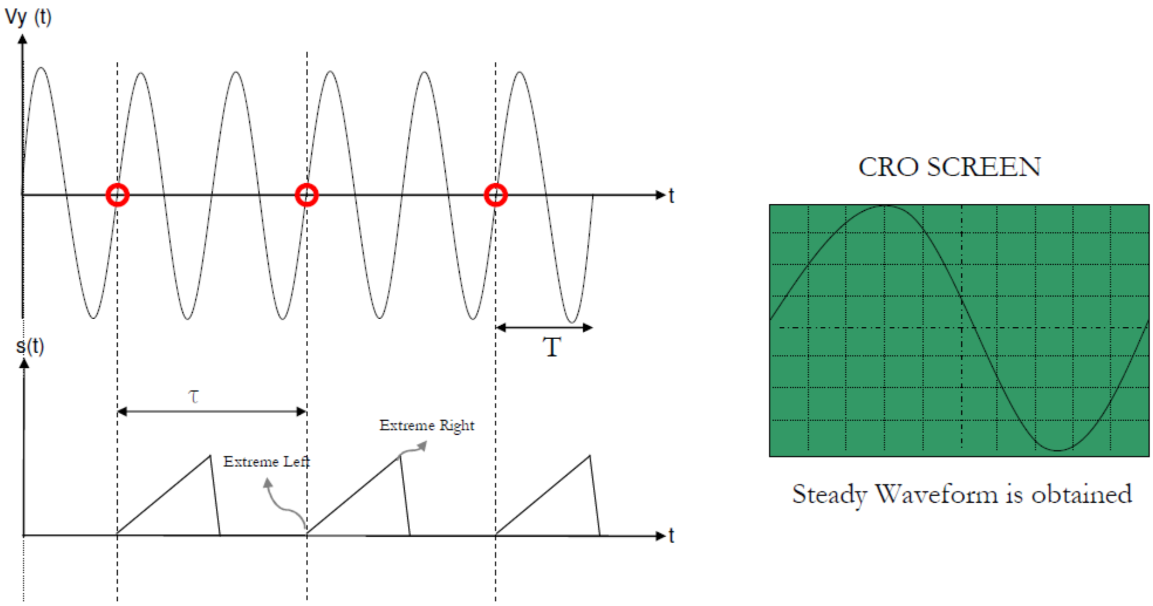


Figure 57a.

Suppose the input Vy(t) and s(t) shown in Figure 57.a are applied to the vertical and horizontal deflection plates of the CRO respectively. Note that, At the beginning of each sweep cycle, (i.e when the bright spot is at extreme left) Vy(t) gets exactly the same value (The points indicated by red circles). Therefore the bright spot is following exactly the same path in each sweep cycle. Thus, we can observe a steady waveform on CRO screen. Notice that the time between the beginning of two consecutive sweep cycles is a multiple of input signal period. (i.e. = n T. T, are shown on the figure 57b, n is a positive integer.)

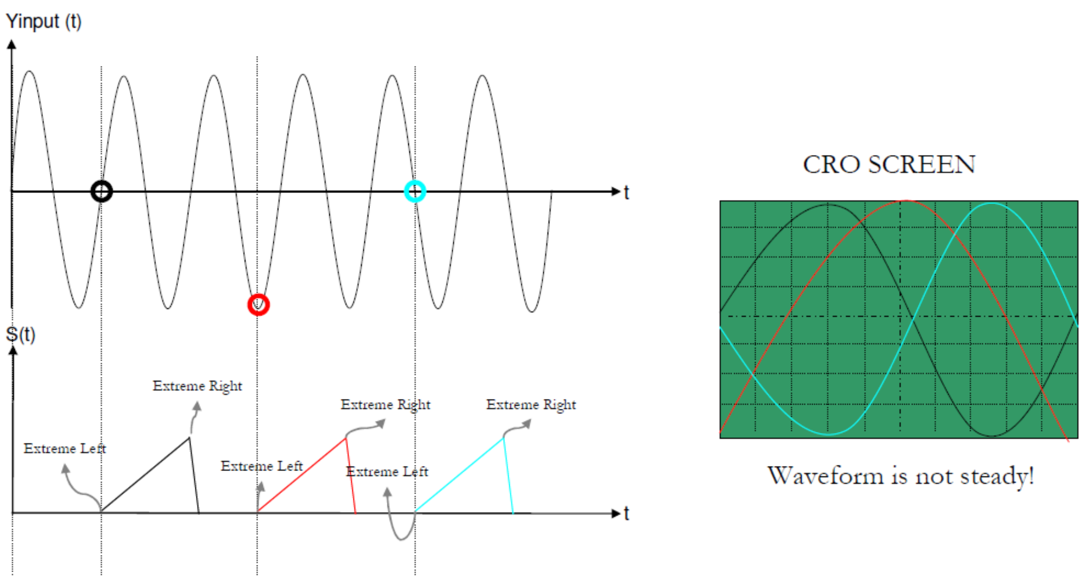


Figure 57.

For the given case, the bright spot is following different paths in different sweep cycles, therefore we can not obtain a steady waveform on CRO screen.

In order to obtain stable and stationary waveform displays, the sawtooth signal should be applied to the horizontal deflection plates, in synchronism with the waveform being displayed. CRO handles this synchronization problem by using the following structure.

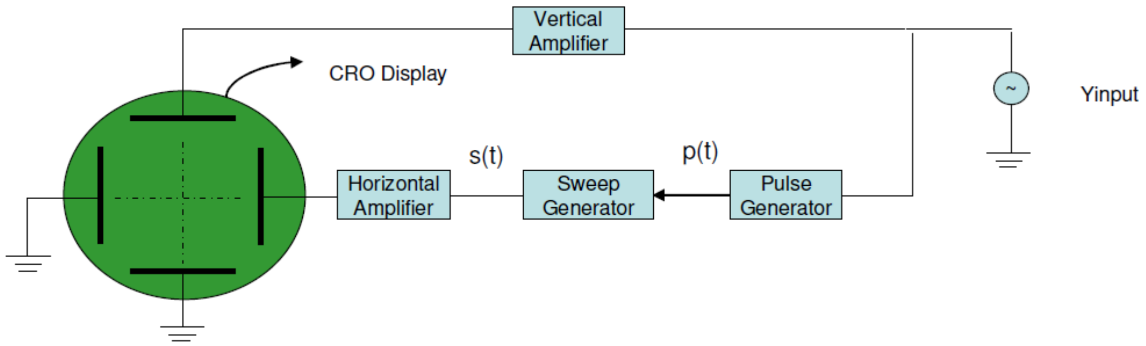


Figure 58.

Notice that, the voltage waveform which is to be displayed on the CRO screen (Yinput in this case) is applied to the vertical amplifier. In the amplification stage, only the amplitude of the input waveform is changed. After the amplification stage, the output of the vertical amplifier is applied to the vertical deflection plates. Then in order to obtain a steady waveform on the CRO screen, a sawtooth waveform having a period which is an integer multiple of the period of the input voltage waveform should be applied to the horizontal deflection plates.

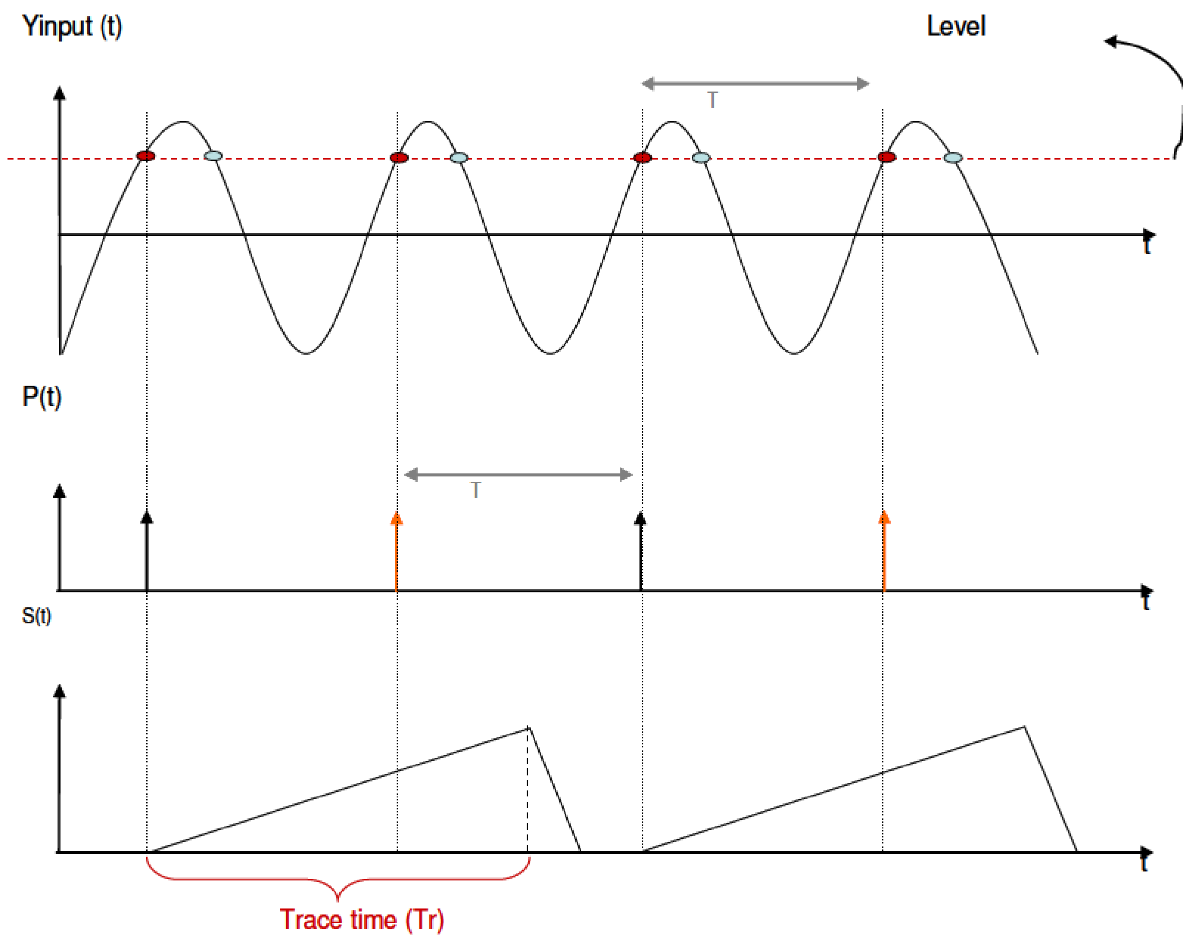


Figure 59.

Pulse Generator

The main function of the pulse generator (PG) (see figure 58.) is to produce periodical pulses with a period of T, which is equal to the period of the input signal. For that purpose, the input signal is compared to a certain voltage level ('Level' on the figure 59). Producing pulses each time the input voltage is equal to that certain voltage level, may seem to result pulses which are periodic with period T. But this is not the case. Notice that, the 'Level' intersects the input signal more than once in one period. Therefore, one of the intersection points is neglected in each period. The decision on which intersection point to neglect is made by inspecting the slope of the signal at the intersection point.

The pulse generator produces pulses each time the input voltage level is same as the 'Level', after checking the slope of the signal at that time instant. In the example given in figure 59, there are two intersection points at each period, and the one with the negative slope (blue points) are neglected. (The selected slope is positive for that case). Actually, the sign of the slope can be selected by using the +/- button of the CRO. Also the voltage level that the signal is being compared to, can be adjusted by using the level button on the oscilloscope.

Sweep Generator

The main function of the sweep generator is to produce one cycle of a sawtooth waveform, when it receives a pulse at its input. If the sweep generator receives a trigger pulse during its sweep cycle (i.e., during the trace period Tr), it will simply ignore the pulse and continue with the completion of its sweep cycle.

Depending on the selected level and the slope of the input signal, the output of the pulse generator will consist of narrow trigger pulses separated from each other by one period T. Each time the input signal crosses a preselected level (and a preselected slope), the pulse generator emits one narrow trigger pulse. The emitted pulse triggers the sweep generator to begin producing one cycle of the sweep waveform; its duration is the trace period Tr. At the end of each sweep cycle, the sweep generator stops its output and awaits the arrival of the next trigger pulse before producing a new sweep cycle.

Notice that if the sweep generator receives a trigger pulse during its sweep cycle (i.e., during the trace period Tr), it will simply ignore the pulse and continue with the completion of its sweep cycle. The trigger pulse received after the completion of the trace period will initiate the new sweep cycle. This allows the scope to display more than one cycle, of period T, of the signal connected to its vertical deflection plates.

The following figure illustrates an example.

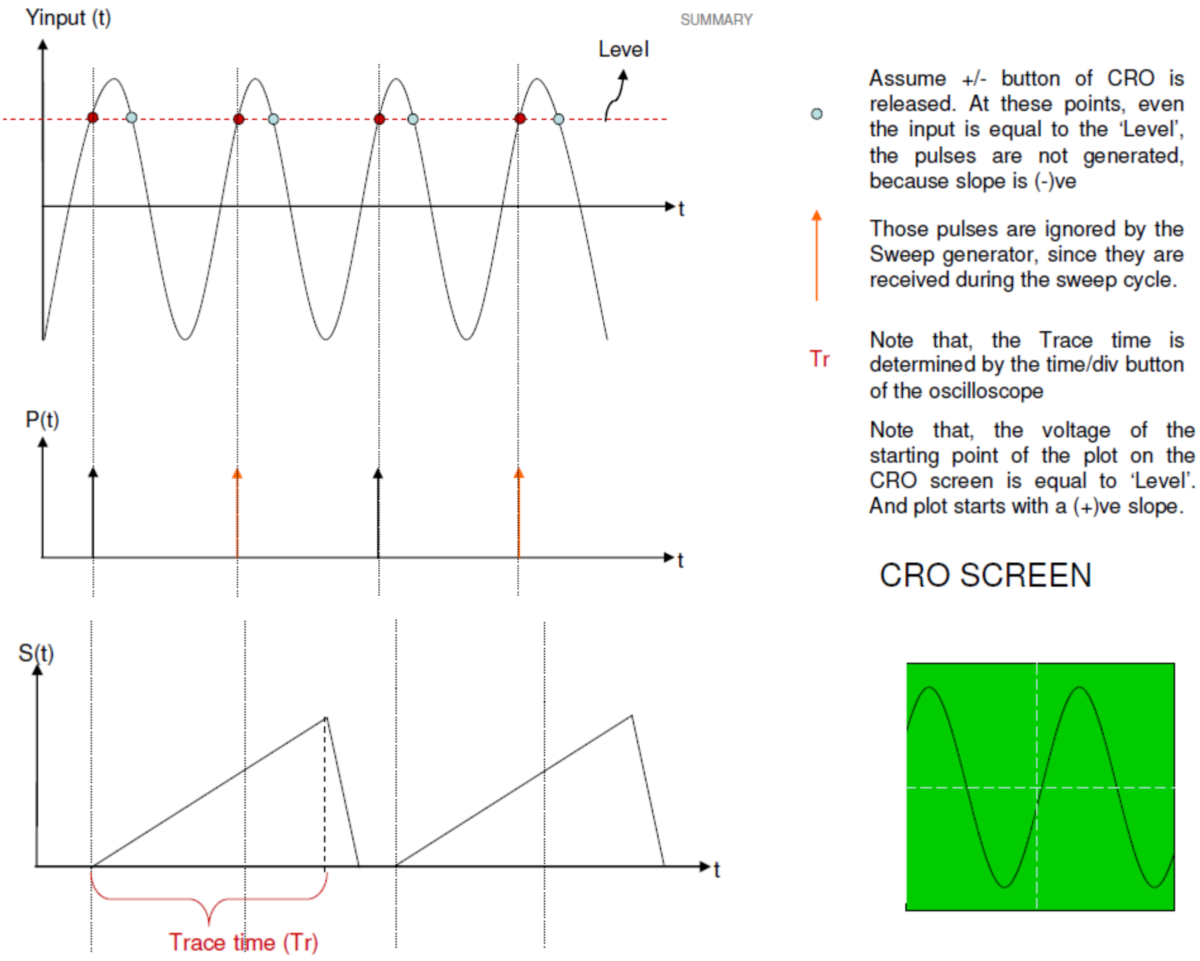


Figure 60.

In the given example, first the voltage waveform to be displayed on the CRO screen is compared with a voltage level. The blue and the red points show the intersection of the input signal with the level. Assuming the positive slope is selected, pulse generator produces pulses at each time the input signal is equal to 'level', and its slope is positive. The pulses generated by the pulse generator, trigger the sweep generator, which produces one cycle of the sawtooh waveform. The trace time of the sweep generator is adjusted by the time/div button which is available on the front panel of the oscilloscope. The resulting sawtooth waveform is applied to the horizontal deflection plates, which leads to a steady display of the input signal on the oscilloscope screen. Notice that, at the beginning of each sweep period (when the bright spot is at the extreme left), input signal voltage is equal to \`level\` and has a positive slope.

Therefore, the waveform shown on the CRO screen starts with a positive slope at the extreme left and its value is equal to the 'level'. One can change these settings by varying the 'level' control or the +/- button of the oscilloscope.

The whole process is called triggering because, obtaining a steady plot on the CRO screen can only be achieved by producing pulses at the input of the Sweep Generator at the correct time instances. (i.e. triggering the Sweep Generator at the correct time instances.)

X-Y Operation

When the variation of one voltage waveform, Vy(t), as a function of another, Vx(t), eliminating the parameter time, t, is desired, X-Y mode of operation is used. In X-Y mode, one signal is applied to the vertical deflection plates whereas the other signal is applied to the horizontal deflection plates. The XY button on the front panel of the oscilloscope disconnects the triggering signal from the horizontal deflection system, and connects the second input signal instead. This process is done by using a switch shown as 'X-Y button' on the figure below.

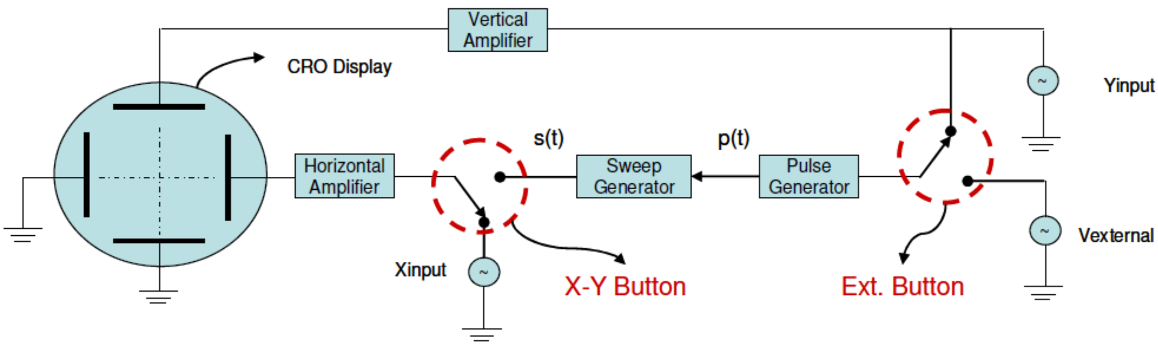


Figure 61.

External Triggering

Rather than the input signal itself, an external signal can also be used for triggering. For that purpose multi-positional switch, which corresponds to Ext. Button of the CRO, should be set to position 2, as shown in Figure 61. The external signal should satisfy certain conditions in order to obtain a steady waveform on the CRO screen. Keeping in my that the period of the sawtooth waveform, s(t), should be an integer multiple of the period of the input signal, can you find the conditions needed on the frequency of the external triggering signal?

Digital Storage Oscilloscopes (DSO)

The concept behind the digital oscilloscope is somewhat different to an analogue scope. Rather than processing the signals in an analogue fashion, the DSO converts them into a digital format using an analogue to digital converter (ADC), then it stores the digital data in the memory, and then processes the signals digitally, finally it converts the resulting signal in a picture format to be displayed on the screen of the scope.

Since the waveform is stored in a digital format, the data can be processed either within the oscilloscope itself, or even by a PC connected to it. One advantage of using the DSO is that the stored data can be used to visualize or process the signal at any time. The analogue scopes do not have memory therefore the signal can be displayed only instantaneously. The transient parts of the signal (which may vanish even in milliseconds or microseconds) can not be observed using an analogue oscilloscope.

The DSO's are widely used in many applications in view of their flexibility and performance.

Measurement Techniques

The major concern in observing a signal on the oscilloscope screen is to make voltage and time measurements. These measurements may be helpful in understanding the behavior of a circuit component, or the circuit itself, depending on what you measure. Except for the X-Y mode of operation, the oscilloscope displays the voltage value of the waveform as a function of time. The oscilloscope screen is partitioned into the grids, which divides both the horizontal axis(voltage) and the vertical axis(time) into divisions which will be helpful in making the measurements. See Figure 62.

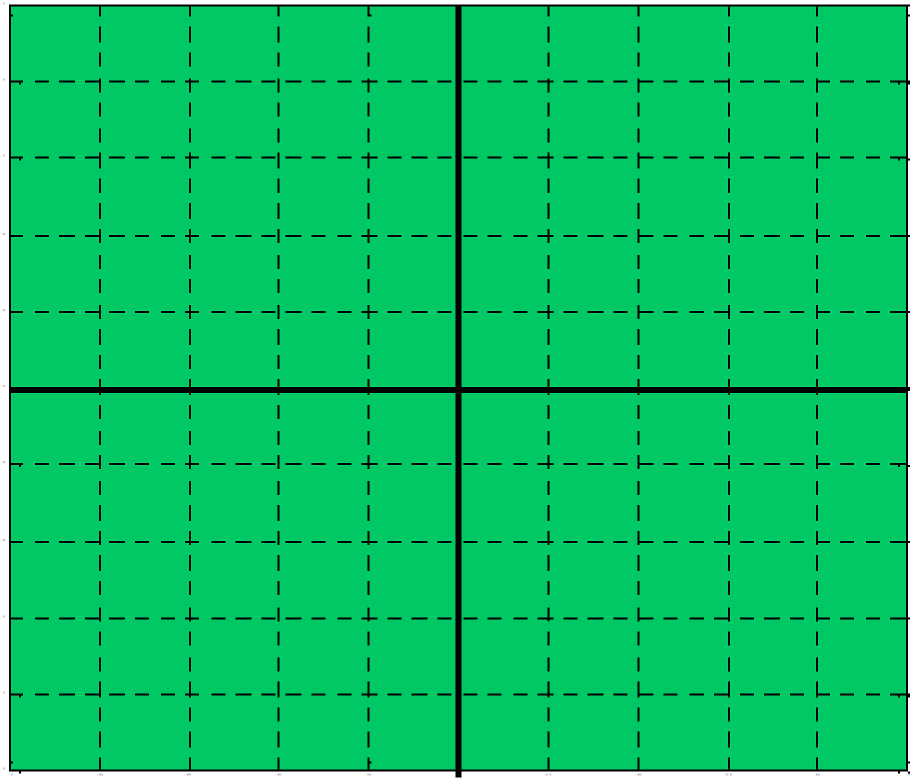


Figure 62. Oscilloscope Screen.

Obviously one needs to know the time or the voltage values corresponding to each division, in order to make accurate calculations. These values are determined by two variables namely the **time/div** and the **volt/div** both of which can be adjusted from the relevant buttons available on the front panel of the oscilloscope . Also note that, the time/div buton controls the trace time of the sweep generator, whereas the volt/div button controls the `gain` in the vertical amplifiers in the vertical deflection system.

Typical quantities, which are of primer interest when observing a signal with the scope, are shown in Figure 63.

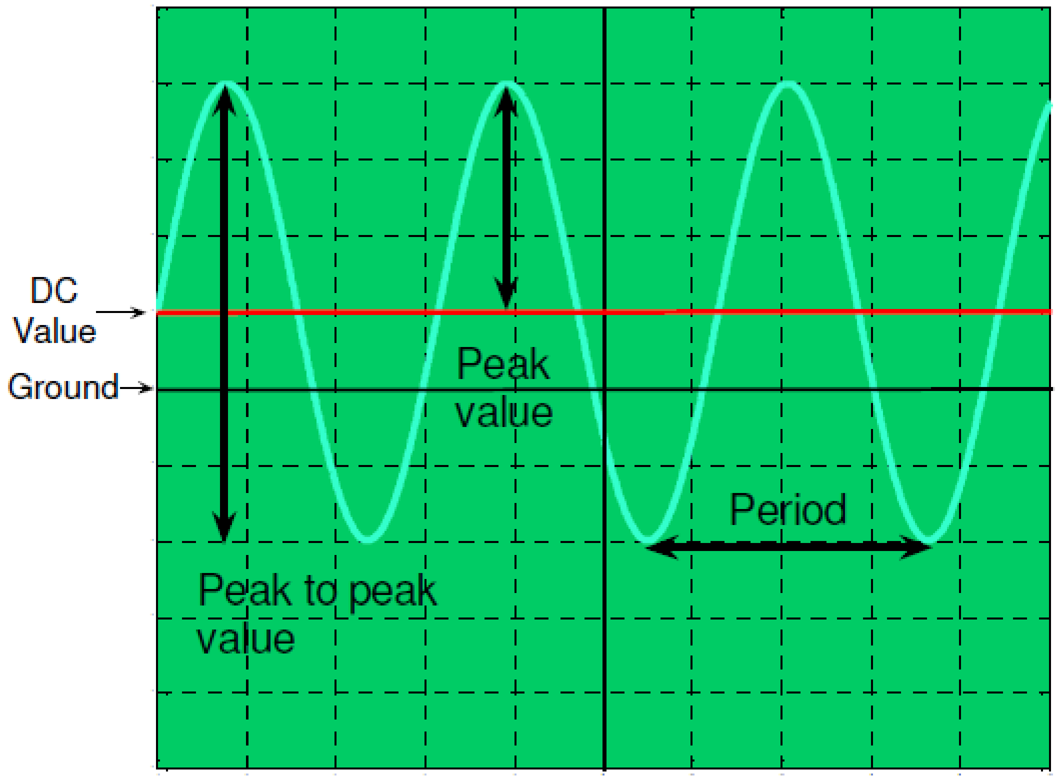


Figure 63. Sinusoidal Signal on Oscilloscope Screen.

For the given figure, suppose that the variables **volt/div** and **time/div** are set to:

volt/div = 2Volts/div.

time/div = 1millisecond/div

Then the corresponding values shown on the figure are calculated to be;

Peak Value = 6volts

Peak to peak value = 12 Volts

DC Value (Average Value) = 2 Volts

Period = 3 milliseconds

Frequency = 1/ *Period* = 333 Hz.

Note that the signal s(t), shown on the oscilloscope screen can be expressed as,

$s(t)= V_{peak} \sin(2\pi ft)+V_{DC}$

$= 6\sin(2\pi 333t)+2$

$= 6\sin(666\pi t)+2$ Volts.

Phase difference

In some applications, one may need to monitor or compare two or more signals simultaneously. A typical example can be the comparison of the input voltage with the output voltage of a two-port (input and output ports) circuit. If the signals that are being monitored have the same frequency, a time delay may occur between the signals (i.e. one signal may lead the other or vice versa). Two waves that have the same frequency, have a phase difference that is constant (independent of t). When the phase difference (modulo 2) is zero, the waves are said to be **in phase** with each other. Otherwise, they are **out of phase** with each other. If the phase difference is 180 degrees (radians), then the two signals are said to be ***in anti-phase***. If the peak amplitudes of two anti-phase waves are equal, their sum is zero at all values of time, t.

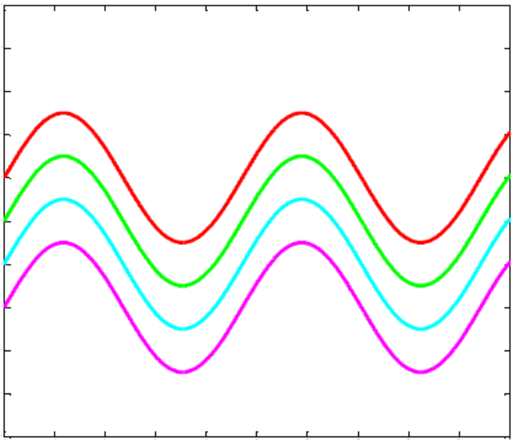


Figure 64. In-phase Waves

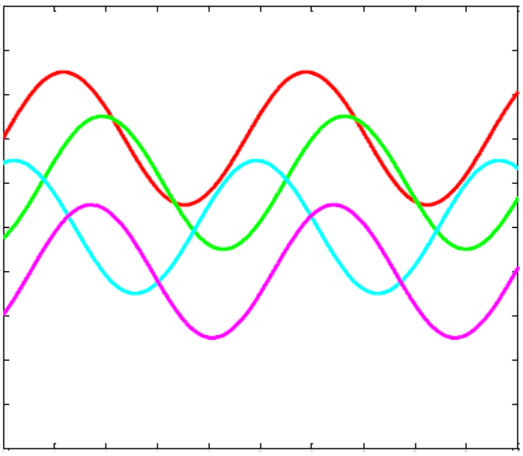


Figure 65. Out of phase Waves

The phase difference is expressed in terms of radians or degrees. In **Dual Mode** of the oscilloscope the phase difference can be calculated easily as follows.

Given the two signals having the same frequency, as shown in Figure 66,

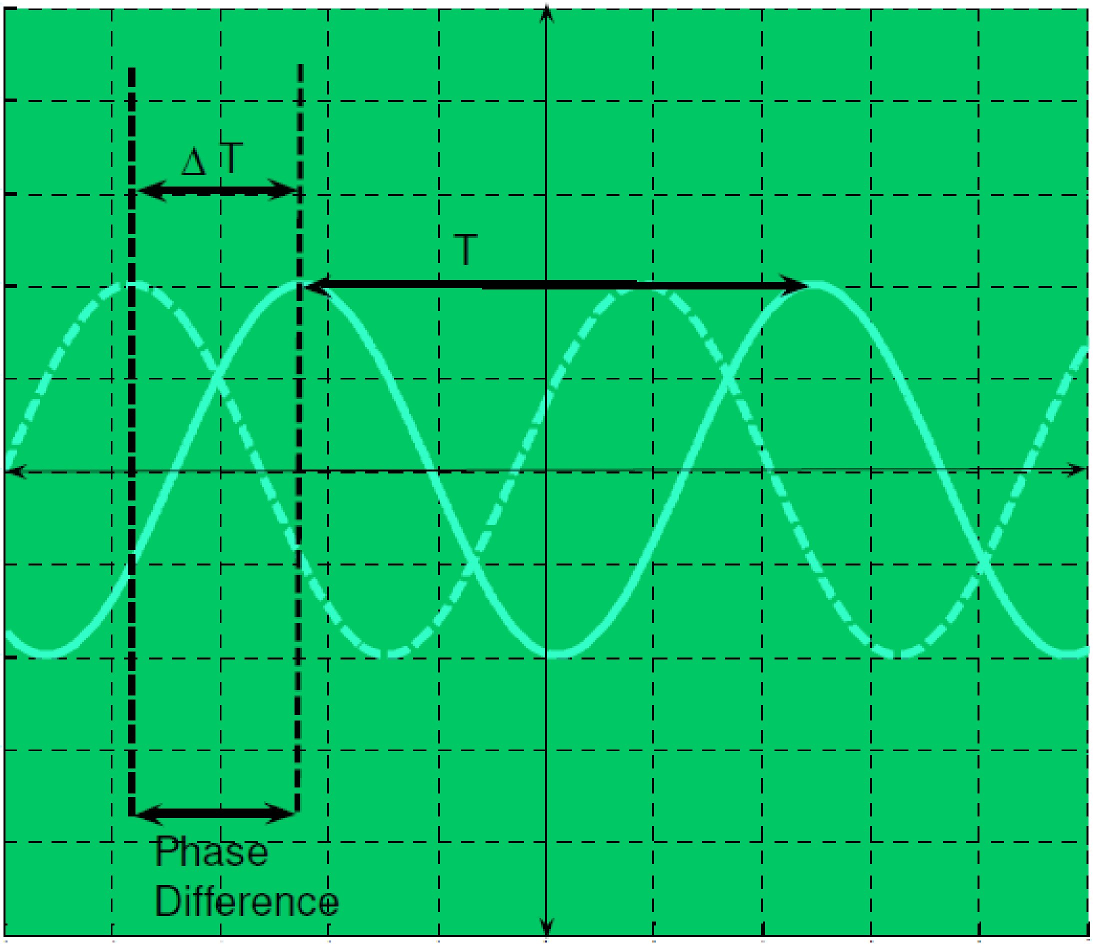


Figure 66. Two Signals Displayed in DUAL Mode

define,

ΔT = horizontal spacing of the peak values (or the zero crossings) of the two signals.

T = horizontal spacing for one period.

Then the phase difference, is;

$\theta = (\Delta T/T)x360^o$ *in degrees*

$\theta = (\Delta T/T)x2\pi$ *in radians*

Note that, one has to specify the leading or the lagging signal in order to fully describe the time difference between the two signals. In the figure above, the signal represented with dashed curve leads the other. Suppose that the signal represented by the dashed curve is connected to Channel I of the oscilloscope, and the other one is connected to Channel II. In such a case Channel I is **leading** the Channel II with phase difference equal to , and Channel II is **lagging** the Channel I with phase difference equal to . Determining the leading or the lagging signal may be frustrating at first, but note that the dashed curve reaches its maximum value before the other does.

The phase difference between the signals can also be determined in **XY mode** of the oscilloscope. In the **XY mode**, the **x-axis** data is taken from one channel, **y-axis** data is taken from the other. In that way, **Channel I** vs **Channel II** graph can be obtained, so that the variation of a signal with respect to another can be observed. Figure 67 shows a typical graph in **XY mode**, of two signals having a constant phase difference.

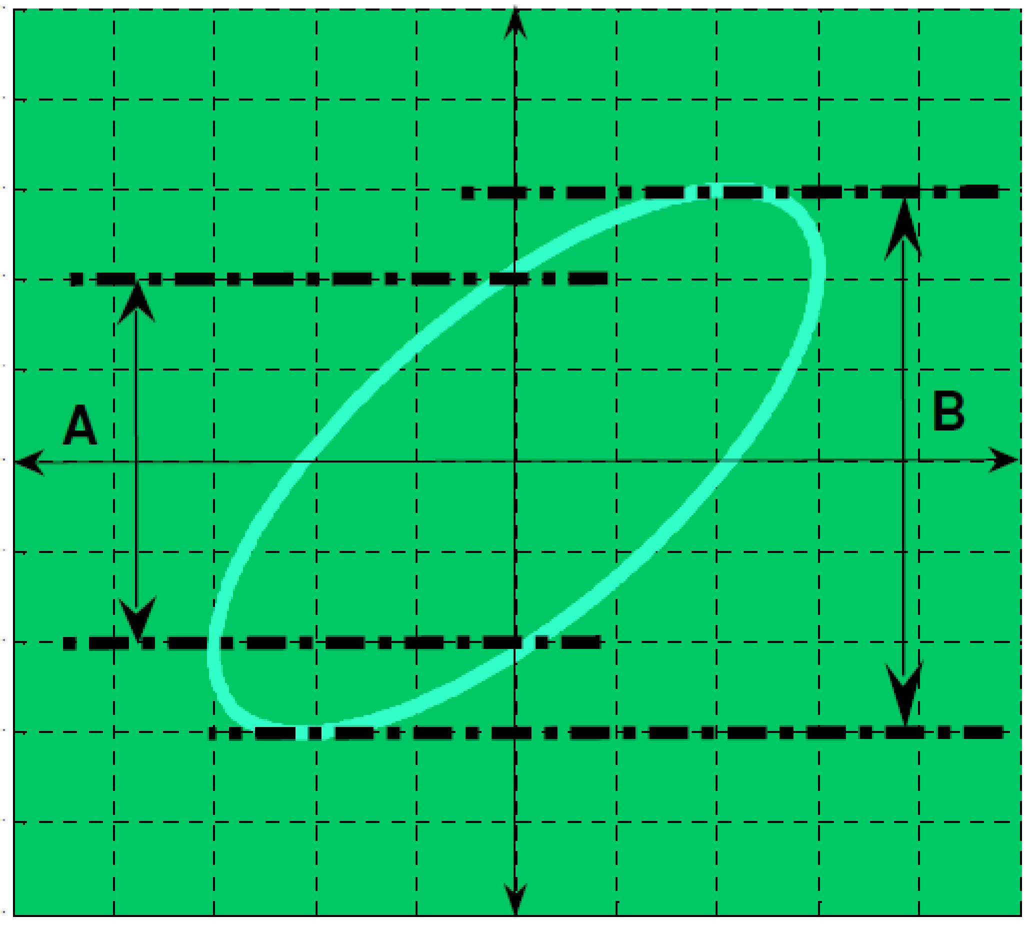


Figure 67. Phase Difference Calculation in XY Mode

Phase difference is equal to,

$\theta = \sin^{-1}(A/B)$.

One can show this relation by expressing one signal as, $y(t)= (B/2) \sin(wt\pm \theta)$ and the other signal as, $x(t)= (C/2) \sin(wt)$. Then consider the value of y(t) when x(t) is zero volts. It should be noted that, the center of the ellipsoidal shape (sometimes circular or linear shapes) on the screen should be at the origin of CRO unless any DC component is added to one of the signals.

In XY mode, the leading or the lagging signal can not be determined. One has to switch to **DUAL mode** in order to specify the leading signal.

Figure 68 shows typical graphs in XY mode corresponding to different values of phase difference.

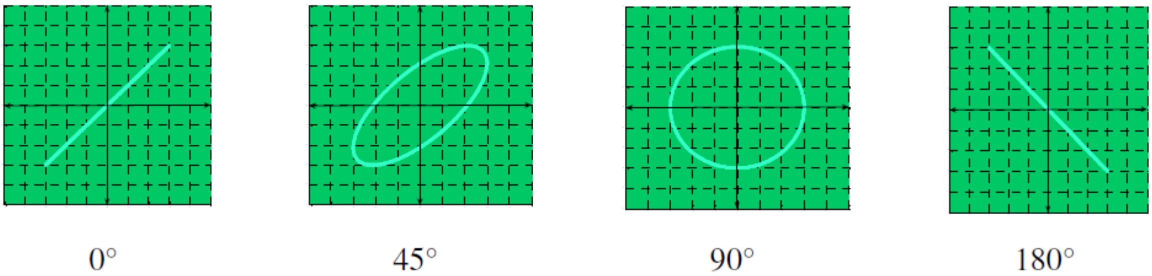


Figure 68. The Graphs in XY Mode for Different Phase Difference Values

Controls

Display Controls

Display systems may vary between analog and digital oscilloscopes. Common controls include:
· An intensity control to adjust the brightness of the waveform. As you increase the sweep speed of an analog oscilloscope, you need to increase the intensity level.
· A focus control to adjust the sharpness of the waveform. Digital oscilloscopes may not have a focus control.
· Other display controls may let you adjust the intensity of lights and turn on or off any on-screen information (such as menus).

Vertical Controls

Vertical controls are used to position and scale the waveform vertically. Oscilloscopes also have controls for setting the input coupling and other signal conditioning, described in this section.

Position and Volts per Division Settings

The position knob moves the waveform vertically. The scale knob varies volts per division (usually written volts/div), which determines the voltage value corresponding to each vertical division on the oscilloscope's screen. As the volt/div value is altered, the size of the waveform on the screen changes.

The volts/div setting is a scale factor. For example, If there are ten vertical divisions on the oscilloscope screen and if the volts/div setting is 5 volts, then each of the vertical divisions represents 5 volts and the entire screen can show 50 volts from bottom to top. If the setting is 0.5 volts/div, the screen can display 5 volts from bottom to top, and so on. The maximum voltage you can display on the screen is the volts/div setting times the number of vertical divisions.

Often the volts/div scale has either a variable gain or a fine gain control for scaling a displayed signal to a certain number of divisions. Figure 69 shows the vertical controls

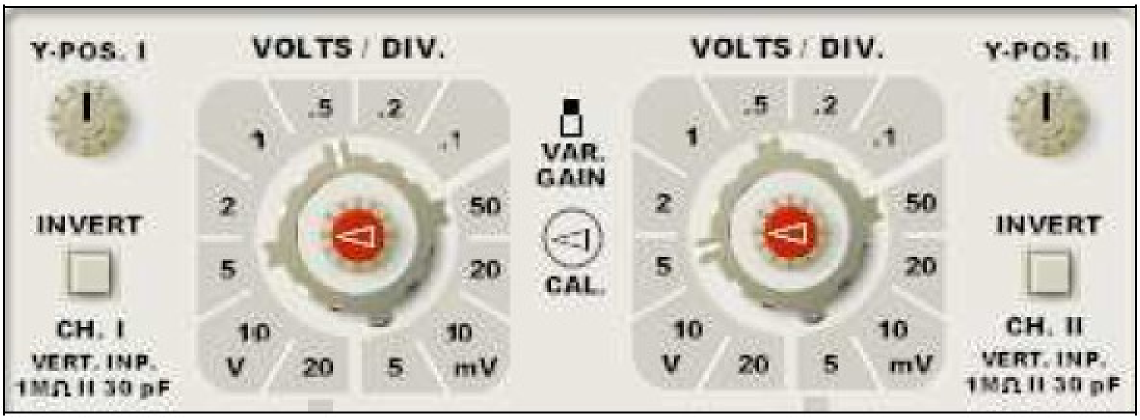


Figure 69. Vertical Controls

Horizontal Controls

Horizontal controls are used to position and scale the waveform horizontally. Figure 70 shows typical front panel for the horizontal controls.

Figure 70. Horizontal Controls

The horizontal position control (x-pos.) is used to move the waveform from left and right to exactly where you want it on the screen.

The time per division (time/div) setting lets you select the rate at which the waveform is drawn across the screen (also known as the time base setting or sweep speed). This setting is a scale factor. For example, if the setting is 1 ms, each horizontal division represents 1 ms and the total screen width represents 10 ms (ten divisions). Changing the time/div setting lets you look at longer or shorter time intervals of the input signal.

As with the vertical volts/div scale, the horizontal sec/div scale may have variable timing, allowing you to set the horizontal time scale in between the discrete settings. Also note that, the time/div button actually controls the trace time of sawtooth waveform in the sweep generator. When sawtooth waveform is zero volt, the bright spot is at the extreme left-hand position, and when it is maximum, the bright spot is at the extreme right position. Therefore, the bright spot travels from extreme left to extreme right in a time equal to the Trace time. Assume that the CRO screen is divided into N equal horizontal divisions. The bright spot travels the N divisions in Tr seconds. Therefore each division corresponds to (Tr/N) seconds. If the Trace time is changed, the corresponding time for each division is changed. Time per division controls can be used to select the appropriate time/div (i.e., the Trace time of the sawtooth waveform).

Input Coupling

Coupling means the method used to connect an electrical signal from one circuit to another. In this case, the input coupling is the connection from your circuit to the oscilloscope. The coupling can be set to **DC**, **AC**, or **ground (GND)**. By setting the coupling control to **AC,** the **DC offset** voltage is removed form the input waveform, so that you see the waveform centered at zero volts. When **DC** coupling is selected, both **AC** and **DC** components of the input waveform are passed to the oscilloscope. Figure 28 illustrates the difference. The signal in Figure 27 is $y(t) = 3 + \sin(wt)$ where 3 Volts is DC component and sin(wt) is AC component. By selecting **AC** coupling, DC component is eliminated and only the signal of sin(wt) is shown on the screen (Figure 71-b). The AC coupling setting is useful when the entire signal (alternating plus constant components) is too large for the volts/div setting.

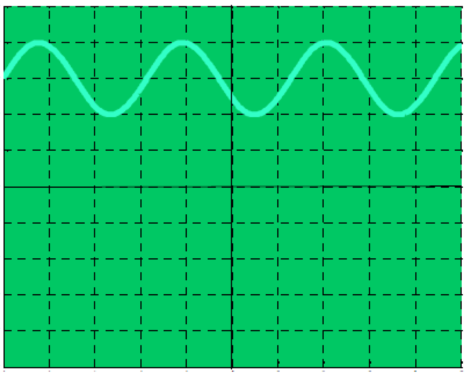


Figure 71-a. 2V peak to peak sinusoidal with 3 Volts offset, shown in **DC** mode.

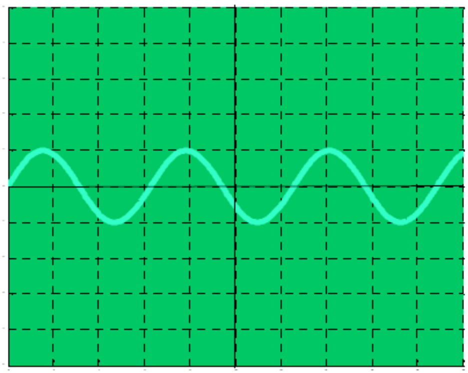


Figure 71-b. 2V peak to peak sinusoidal with 3 Volts offset, shown in **AC** mode.

The ground setting disconnects the input signal from the vertical system, which lets you see where zero volts is on the screen. With grounded input coupling and auto trigger mode, you see a horizontal line on the screen that represents zero volts. Switching from DC to ground and back again is a handy way of measuring signal voltage levels with respect to ground.

X-Y Button

Most oscilloscopes have the capability of displaying a second channel signal along the X-axis (instead of time). This is called XY mode. Pressing the **X-Y** button the oscilloscope is used in XY mode.

DUAL Button

The oscilloscopes have the capability of displaying both channel signals on the screen at the same time. This is called the Dual Mode. This mode is usually used to measure phase difference between two signals

Alternate and Chop Buttons

On analog scopes, multiple channels are displayed using either an alternate or chop mode. (Digital oscilloscopes do not normally use chop or alternate mode.) Alternate mode draws each channel alternately - the oscilloscope completes one sweep on channel 1, then one sweep on channel 2, a second sweep on channel 1, and so on. Use this mode with medium- to high-speed signals, when the time/div scale is set to 0.5 ms or faster. Alternate mode is available when only **DUAL** button is depressed. Chop mode causes the oscilloscope to draw small parts of each signal by switching back and forth between them. The switching rate is too fast for you to notice, so the waveform looks whole. You typically use this mode with slow signals requiring sweep speeds of 1 ms per division or less. Chop mode is available when both **DUAL** and **ADD** button are depressed. Figure 28 shows the difference between the two modes. It is often useful to view the signal both ways, to make sure you have the best view.

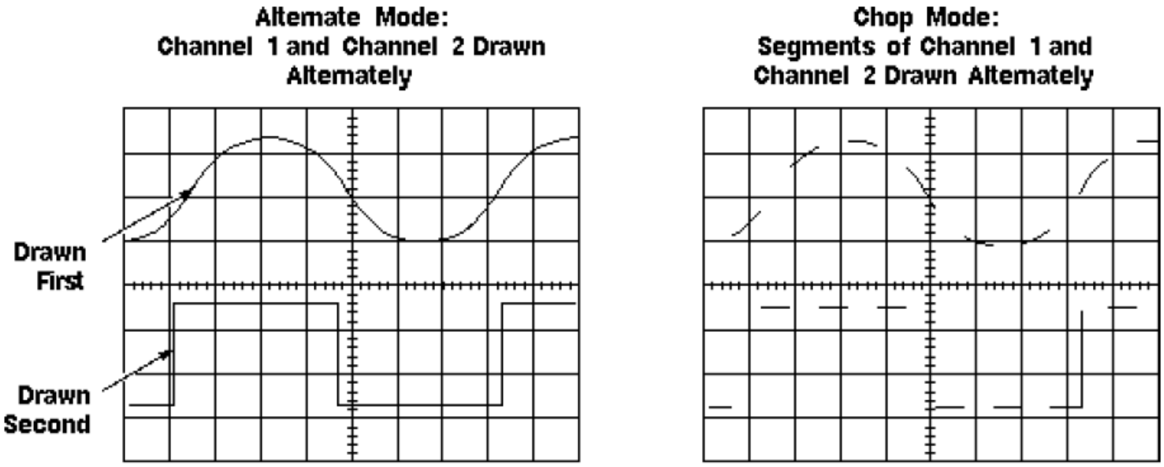


Figure 72. ALT and CHOP modes

ADD Button

When ADD button is depressed, the signals of both channels are algebraically added and the result is displayed on the screen. Volt/div scales of two channels should be the same in order to appropriately see the summation of the signals. When the volt/div scales of the channels are not the same, the signals are summed up as they are displayed on the screen (i.e. graphically). Assume a signal 2sin(wt) is connected to Channel I and a signal 2sin(wt) is connected to channel II. CH I is set to 2 volts/div (Figure 73-a) and CH II is set to 1 volt/div (Figure 73-b). When the ADD button is depressed, the resulting signal on the screen is shown in Figure 30.

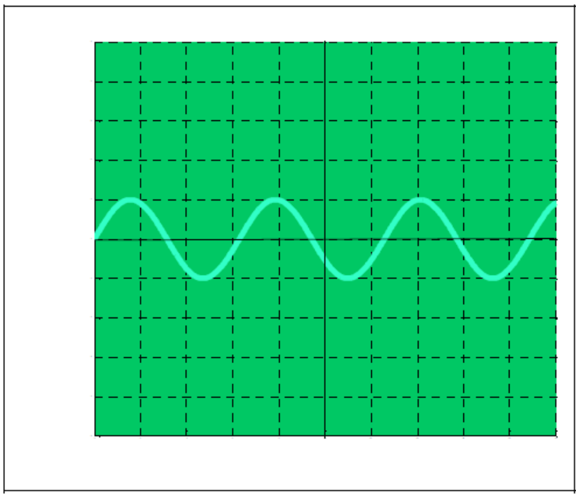


Figure 73-a. The first signal seen on the on the oscilloscope with 2 volt/div scale.

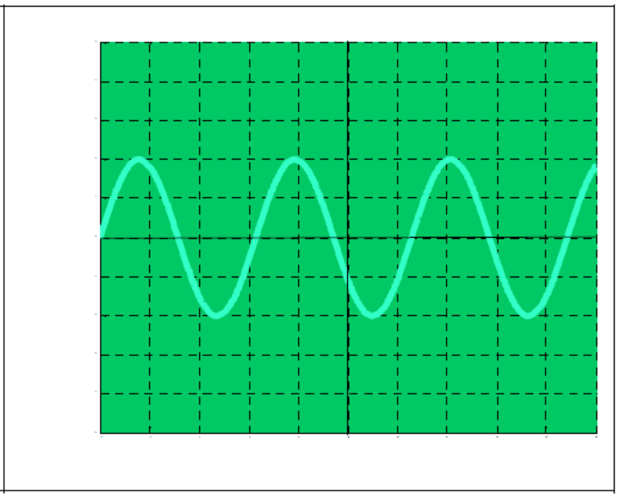


Figure 73-b. The second signal seen oscilloscope with 1 volt/div scale.

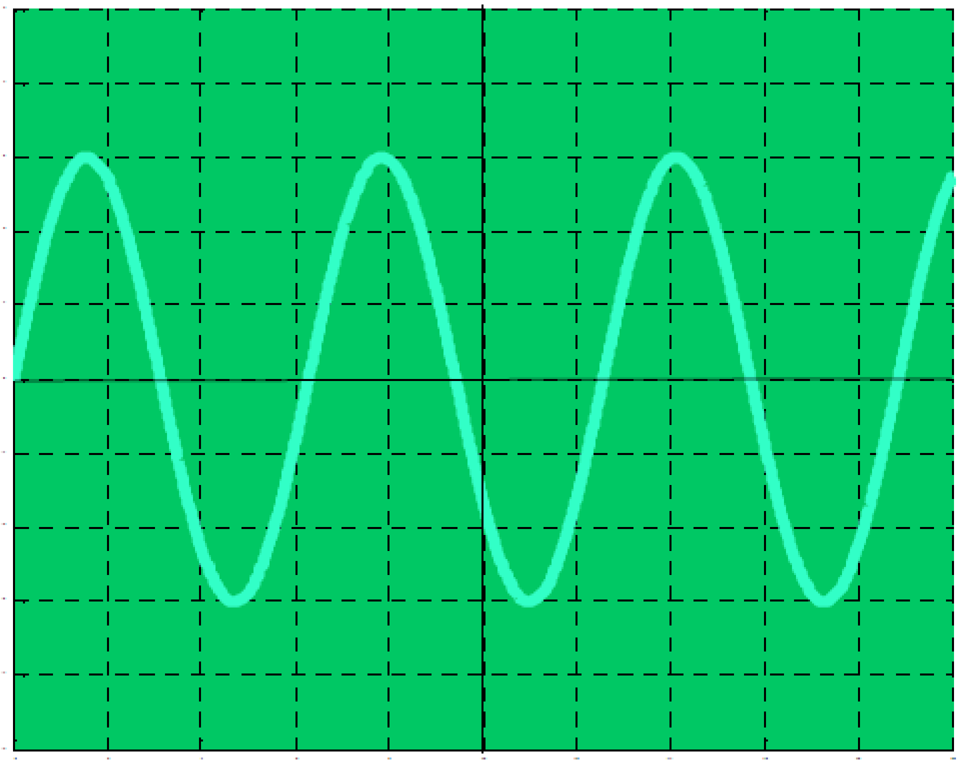


Figure 74. The sum of two signals in Figure 73-a and 73-b when ADD button is depressed.

INVERT Button

When the **INVERT** button of a channel is depressed, negative of the signal is displayed on the CRO screen.

EXT Button

When the **EXT** button is depressed, the oscilloscope is used in external triggering mode.

AT/NORM Button

Using the AT/NORM button you can switch between automatic trigger level selection (AT) and manual trigger level selection (NORM). When the AT/NORM button is released, the automatic trigger level is selected as zero volts, so that the value of the signal on the extreme left of the screen is equal to zero. When the AT/NORM button is depressed, the user can determine the trigger voltage level (the voltage on the extreme left) manually by adjusting LEVEL knob.

LEVEL and +/- Buttons

The trigger level can be set using the **LEVEL** knob when the **AT/NORM** button is depressed. Using the **LEVEL** knob, the trigger voltage level can be set to values different than zero. However, if the trigger level is set to a voltage value that is higher/lower than the positive/negative peak of the signal, the signal can not be triggered and therefore can not be displayed on the CRO screen (Figure 75-d). The **+/-** button is used to determine whether an increasing signal passing from trigger voltage, starts the sawtooth waveform (**+/-** buton released) or viceversa. To be familiar with these buttons, the signals seen on the oscilloscope with various button configurations for the signal in Figure 73-b (1.5sin(wt)) are given in Figure 75.

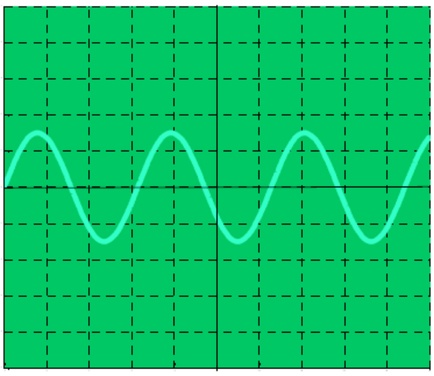


Figure 75-a. The signal when **AT/NORM** button is released. (**LEVEL** is automatically set to 0 volt.)

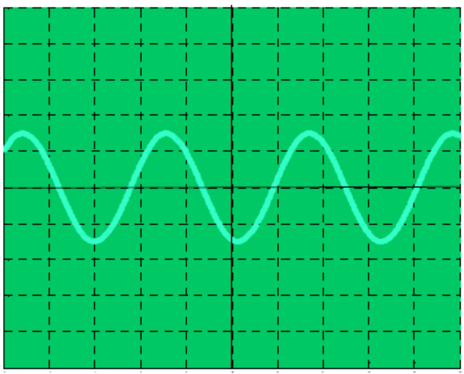


Figure 75-b. The signal when **AT/NORM** button is depressed, **LEVEL** is set to 1 Volt and +/- button is released.

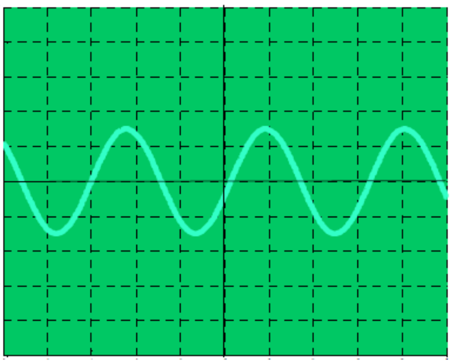


Figure 75-c. The signal when **AT/NORM** button is depressed, **LEVEL** is set to 1 Volt and +/- button is depressed.

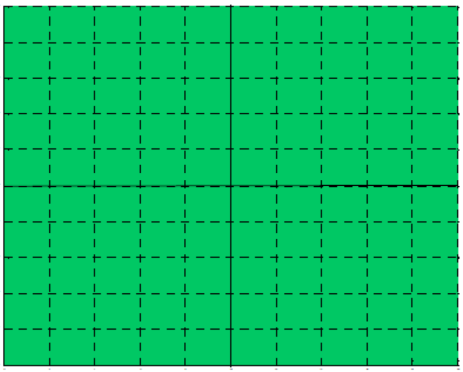


Figure 75-d. The signal when **AT/NORM** button is depressed, **LEVEL** is set to 2 Volt.

Summary of Controls, Connectors, and Indicators

Title	Function	Recommended Use
INTENSITY	Adjusts trace brightness.	Compensate for ambient lighting, trace speed, trigger frequency.
BEAM FIND	Compresses display to within CRT limits.	Locate off-screen phenomena.
FOCUS	Adjusts for finest trace thickness.	Optimize display definition.
TRACE ROTATION	Adjusts trace parallel to centerline.	Compensate for earth's field.
POWER	Turns power on and off.	Control power to the instrument.

Power Indicator	Illuminates when power is turned on.	Know power condition.
POSITION	Moves trace up or down screen.	Position trace vertically and compensate for dc component of signal.
TRACE SEP	Moves the magnified trace vertically with respect to the unmagnified trace when HORIZONTAL MODE is set to ALT.	Position unmagnified and horizontally magnified traces for convenient viewing and measurement.
CH 1-BOTH -CH2	Selects signal inputs for display.	View either channel independently or both channels simultaneously
NORM- INVERT	Inverts the Channel 2 signal display.	Provide for differential (CH 1 - CH 2) or summed (CH 1 + CH 2) signals when ADD is selected.
ADD-ALTCHOP	ADD shows algebraic sum of CH 1 and CH 2 signals. ALT displays each channel alternately. CHOP switches between CH 1 and CH 2 signals during the sweep at 500 kHz rate.	Display summed or individual signals.
VOLTS/DIV	Selects vertical sensitivity.	Adjust vertical signal to suitable size.
Variable (CAL)	Provides continuously variable deflection factors between calibrated positions of the VOLTS/DIV switch. Reduces The CAL control can be pulled out to vertically magnify the trace by a factor of 1 0. Limits bandwidth	Match signals for common mode readings. Adjust height of pulse for risetime calculations. Inspecting small signals.

	gain by at least 25:1. to 5 MHz.	
MAG(X5-X10-X50)	Selects degree of horizontal magnification.	Examine small phenomena in detail.
PROBE ADJUST	Provides approximately 0.5-V,1-kHz square wave.	Match probe capacitance to individual circuit. This source may be used to check the basic functioning of vertical and horizontal circuits but is not intended to check their accuracy.
SLOPE	Selects the slope of the signal that triggers the sweep.	Provide ability to trigger from positive-going or negativegoing signals.
LEVEL	Selects trigger-signal amplitude point.	Select actual point of trigger.
RESET	Arms trigger circuit for SQL SWP.	
HOLDOFF	Varies sweep holdoff time 1 0:1 .	Improve ability to trigger from aperiodic signals.
SOURCE	CH 1,CH2, and EXT trigger signals are selected directly. In VERT MODE, trigger source is determined by the VERTICAL MODE switches as follows: CH 1 : trigger comes from Channel 1 signal. CH 2: trigger comes from Channel 2 signal. BOTH -ADD and BOTH CHOP: trigger is algebraic sum of Channel 1 and Channel 2 signals. BOTH-ALT: trigger comes from Channel 1 and Channel 2 on alternate sweeps.	Select source of signal that is coupled to the trigger circuit.

COUPLING	AC blocks dc components and attenuates signals below 15 Hz. LF REJ blocks dc components and attenuates signals below about 30 kHz. HF REJ blocks dc components and attenuates signals above about 30 kHz. DC couples all signal components.	Select how the triggering signal is coupled to the trigger circuit.
EXT INPUT	Connection for Connection for applying external applying external signal that can be signal that can be used as a trigger, used for intensity modulation.	Trigger from a Provide source other reference blips than vertical by intensity signal. Also modulation from used for single- independent shot application. source.

Measurement Techniques

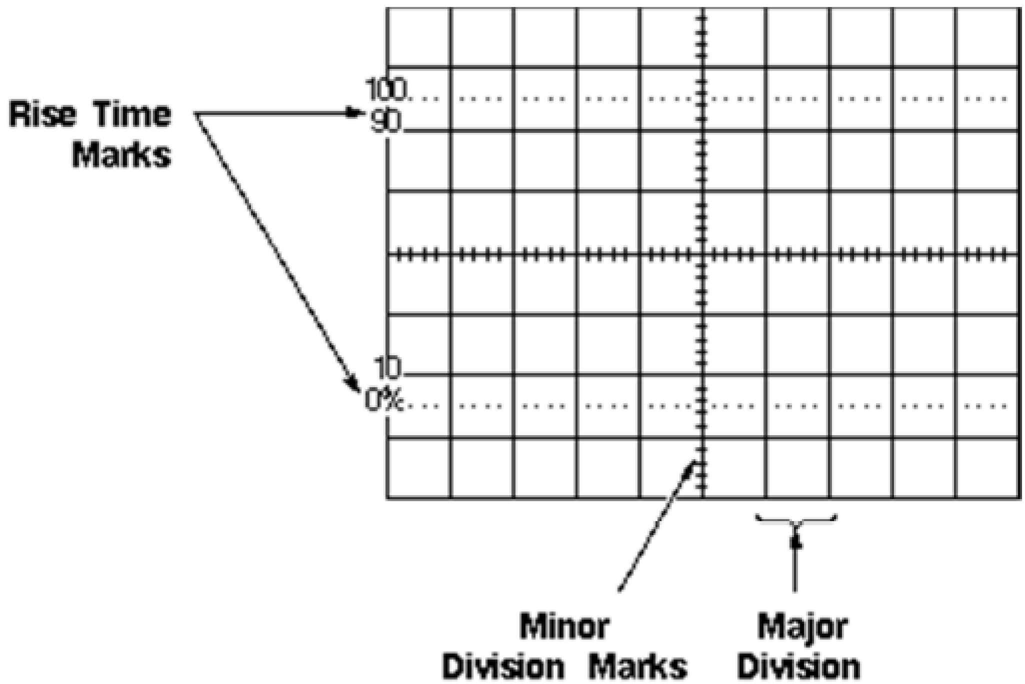
This section teaches you basic measurement techniques. The two most basic measurements you can make are voltage and time measurements. Just about every other measurement is based on one of these two fundamental techniques.

This section discusses methods for taking measurements visually with the oscilloscope screen. Many digital oscilloscopes have internal software that will take these measurements automatically. Knowing how to take the measurements manually will help you understand and check the automatic measurements of the digital oscilloscopes.

The Display

Take a look at the oscilloscope display. Notice the grid markings on the screen - these markings create the *graticule*. Each vertical and horizontal line constitutes a *major division*. The graticule is usually laid out in an 8-by-10 division pattern. Labeling on the oscilloscope controls (such as

volts/div and sec/div) always refers to major divisions. The tick marks on the center horizontal and vertical graticule lines (see Figure 1) are called minor divisions.
Many oscilloscopes display on the screen how many volts each vertical division represents and how many seconds each horizontal division represents. Many oscilloscopes also have 0%, 10%, 90%, and 100% markings on the graticule to help make rise time measurements, described later. An oscilloscope graticule is given below.



Voltage Measurements

Voltage is the amount of electric potential, expressed in volts, between two points in a circuit. Usually one of these points is ground (zero volts) but not always. Voltages can also be measured from peak-to-peak - from the maximum point of a signal to its minimum point. You must be careful to specify which voltage you mean. The oscilloscope is primarily a voltage-measuring device. Once you have measured the voltage, other quantities are just a calculation away. For example, Ohm's law states that voltage between two points in a circuit equals the current times the resistance. From any two of these quantities you can calculate the third. Another handy formula is the power law: the power of a DC signal equals the voltage times the current.

Calculations are more complicated for AC signals, but the point here is that measuring the voltage is the first step towards calculating other quantities.

Ohm’s Law:

Voltage=CurrenttimesResistance

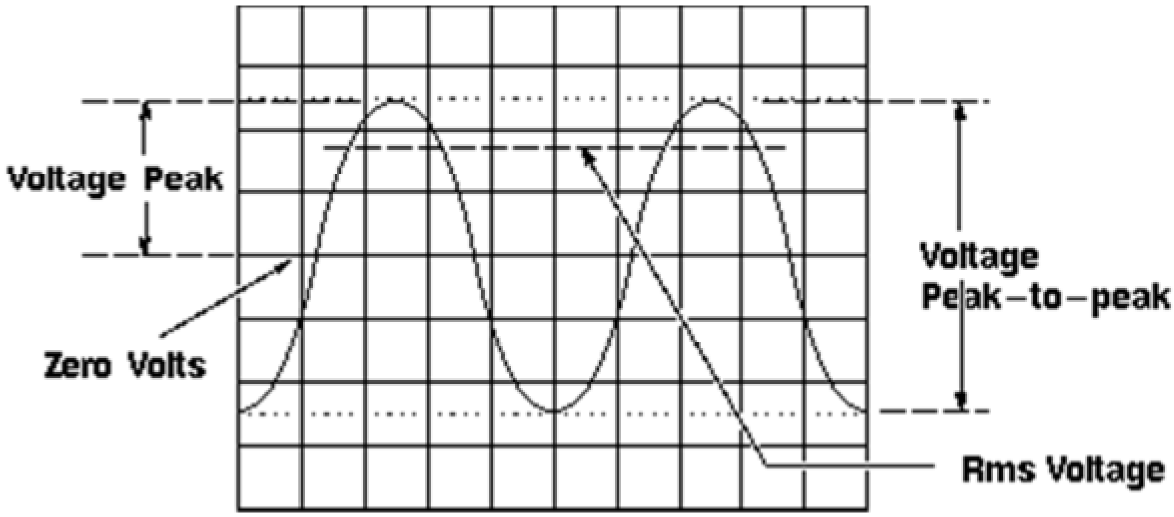
Current=Voltage/Resistance

Resistance=Voltage/Current

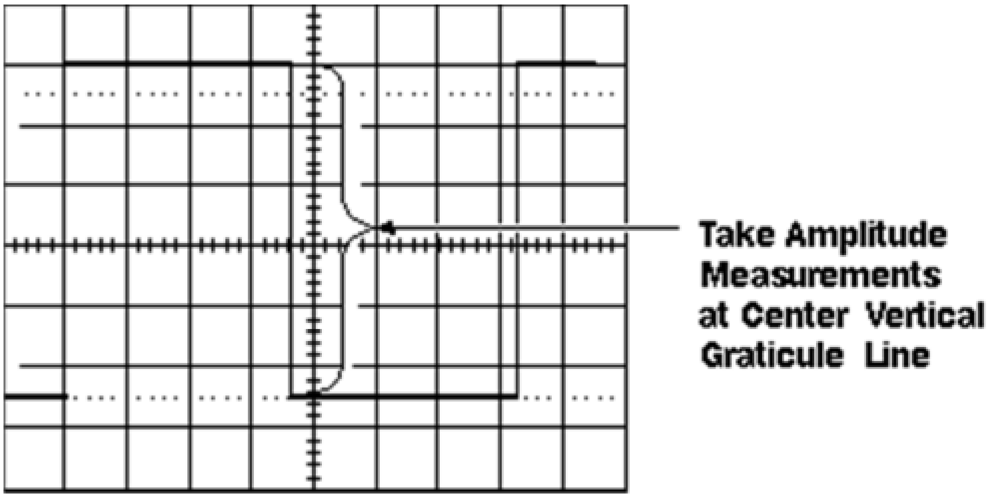
Power Law:

Power=VoltagetimesCurrent

Figure given below shows the voltage of one peak - V[p] - and the peak-to-peak voltage - V[p-p] -, which is usually twice V[p]. Use the RMS (root-mean-square) voltage - V[RMS] - to calculate the power of an AC signal.



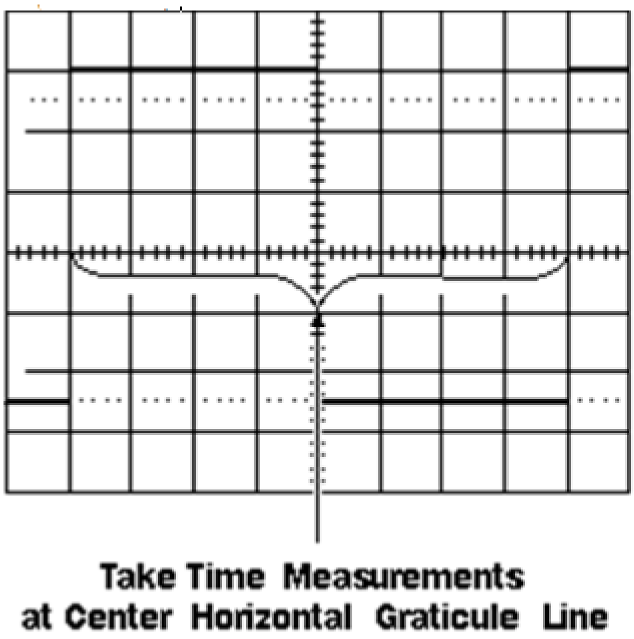
You take voltage measurements by counting the number of divisions a waveform spans on the oscilloscope's vertical scale. Adjusting the signal to cover most of the screen vertically, then taking the measurement along the center vertical graticule line having the smaller divisions makes for the best voltage measurements. The more screen area you use, the more accurately you can read from the screen.



Many oscilloscopes have on-screen *cursors* that let you take waveform measurements automatically on-screen, without having to count graticule marks. Basically, cursors are two horizontal lines for voltage measurements and two vertical lines for time measurements that you can move around the screen. Readout shows the voltage or time at their positions.

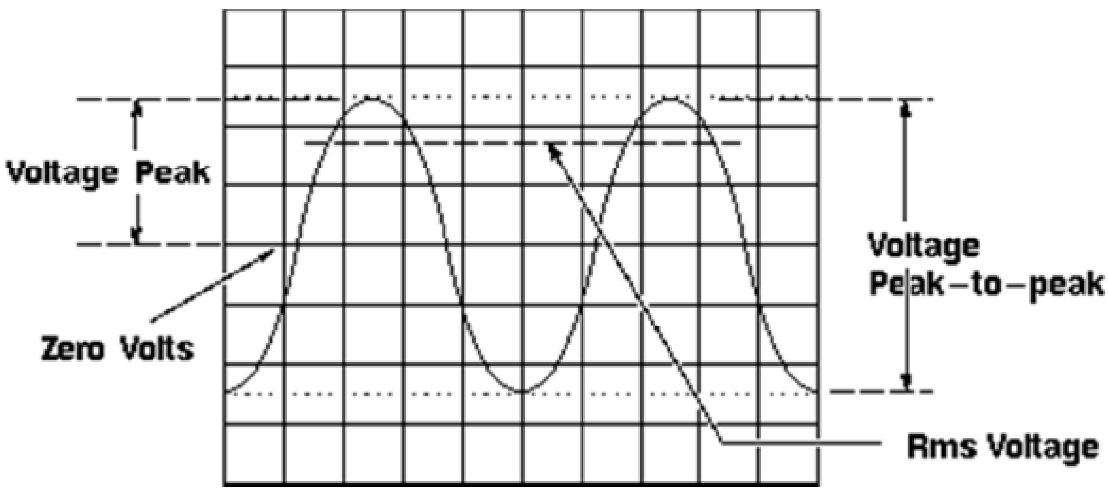
Time and Frequency Measurements

You take time measurements using the horizontal scale of the oscilloscope. Time measurements include measuring the period, pulse width, and timing of pulses. Frequency is the reciprocal of the period, so once you know the period, the frequency is one divided by the period. Like voltage measurements, time measurements are more accurate when you adjust the portion of the signal to be measured to cover a large area of the screen. Taking time measurement along the center horizontal graticule line, having smaller divisions, makes for the best time measurements.



Pulse and Rise Time Measurements

In many applications, the details of a pulse's shape are important. Pulses can become distorted and cause a digital circuit to malfunction, and the timing of pulses in a pulse train is often significant. Standard pulse measurements are *pulse width* and *pulse rise time*. Rise time is the amount of time a pulse takes to go from the low to high voltage. By convention, the rise time is measured from 10% to 90% of the full voltage of the pulse. This eliminates any irregularities at the pulse's transition corners. This also explains why most oscilloscopes have 10% and 90% markings on their screen. Pulse width is the amount of time the pulse takes to go from low to high and back to low again. By convention, the pulse width is measured at 50% of full voltage.



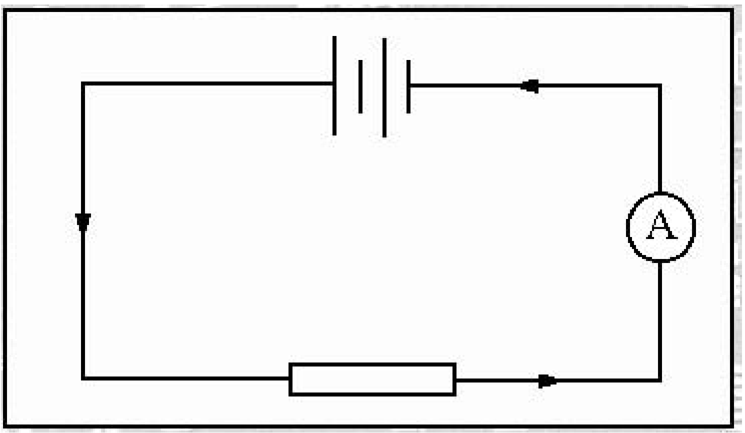
Pulse measurements often require fine-tuning the triggering. To become an expert at capturing pulses, you should learn how to use trigger holdoff and how to set the digital oscilloscope to capture pretrigger data, as described earlier in the Controls section. Horizontal magnification is another useful feature for measuring pulses, since it allows you to see fine details of a fast pulse.

DIGITAL MULTIMETER

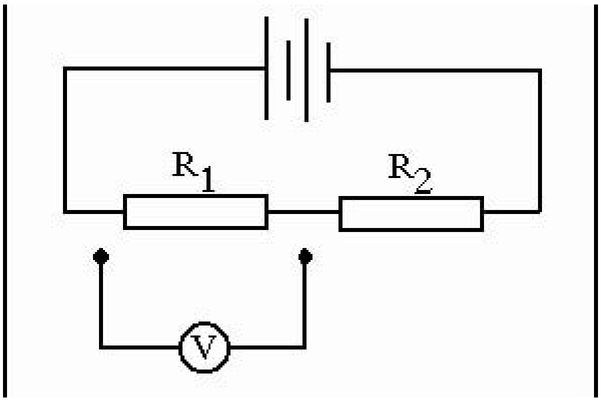
Multimeter is the measuring instrument use to measure voltage, current and resistance of the electronics and electrical circuit. Multimeter is basically an integration of Ammeter, Voltmeter and Ohm-meter. Some of the modern digital Multimeter also contains Frequency meter. Ammeter is used to measure the current. Since current flows through the component, the ammeter must go in series with the component. This makes sure the same current flows through the meter. Current is measured in Amperes (A). Often in electronics we use large resistors which only allow very small current to pass. Therefore we used two other small units.

mA (milliamperes)

µA (microamperes)



Voltmeter is used to measure the voltage and potential difference across the component. Therefore the voltmeter must go in parallel. If the internal resistance of voltmeter is quite small then the loading effect causes the problem.



The unit for measuring the voltage is volt. Small signals such as bio-signals are generally measure in millivolts (mV).

SIGNAL GENERATORS

Principles of Signal Generators

A signal generator is an electronic instrument that generates repeating voltage waveforms. An ideal signal generator can simply be modeled as a voltage source as shown in Fig 76 (a).

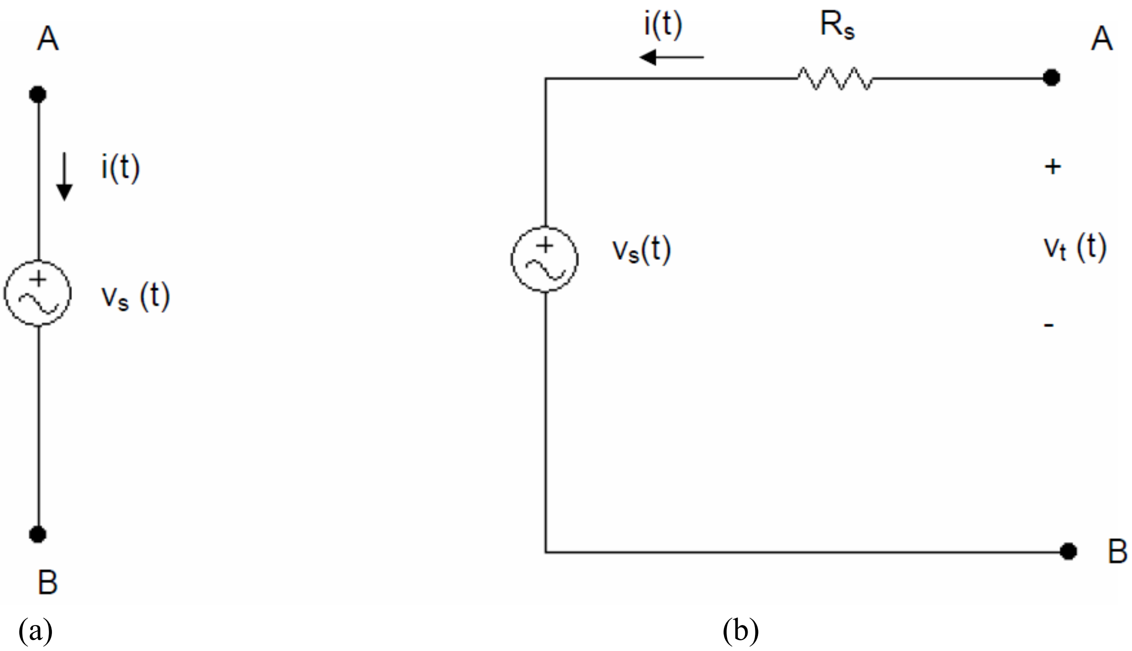


Figure 76

where Vs(t) is a specified function of time. A practical signal generator is modeled as an ideal signal generator connected to a series source resistance (output resistance) Rs as shown in Fig

76 (b). The terminal voltage, v(t), is the output of the signal generator and depends on the terminal current, i(t), and Rs. Vs(t) can be, in general, a sine wave, a square wave, a triangular wave or a pulse train. The first three are characterized by three parameters: frequency (or period), amplitude, and DC (Offset) value. The pulse train is associated with frequency, amplitude and pulse duration.

These parameters can be set to any value in the operation range of the signal generator, using the external controls. In general, amplitude ranges of signal generators vary from 10 mV to 20 V, and frequency ranges vary from 1μHz to 40 MHz. This means signals for which the amplitude and frequency can be set to any value in these ranges can be generated using these signal generators.

Signal generators usually produce more than one type of signals. Different signal types can be obtained by proper connection and/or switching.

A simplified block diagram of a sine and square wave generator is given in Figure 77.

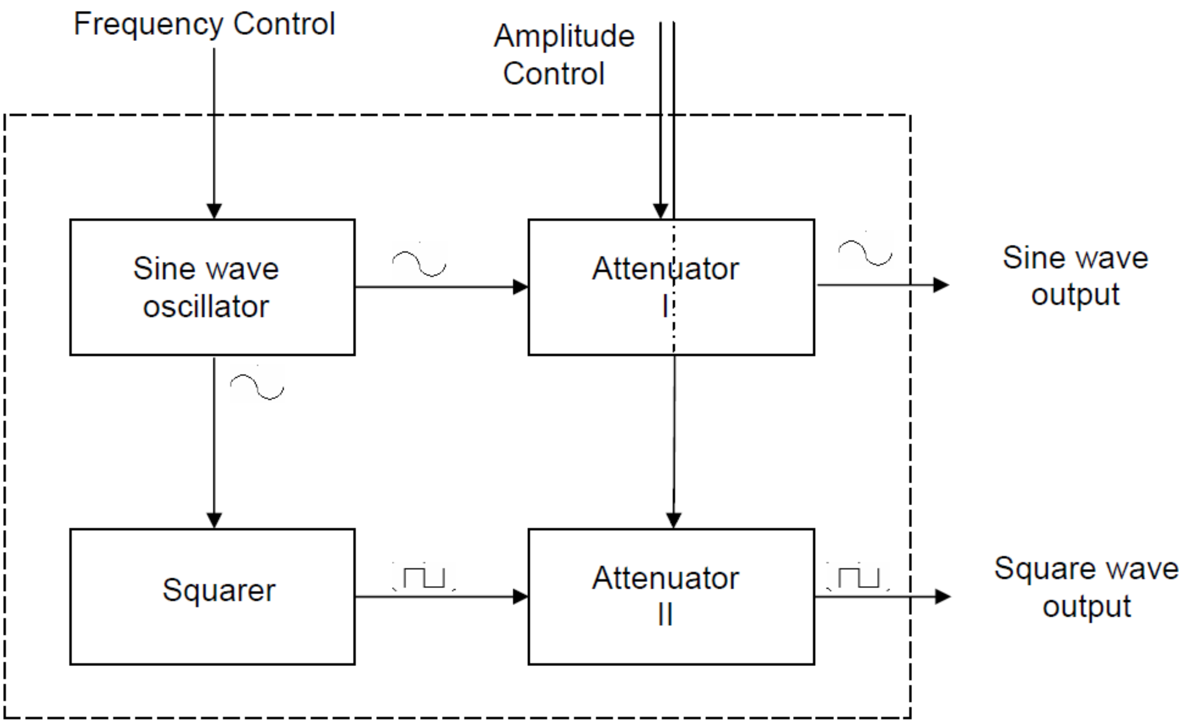


Figure 77.

A sine wave oscillator is the heart of the signal generator. It generates a sine wave of fixed amplitude and adjustable frequency, which is set by the external frequency control. This signal is fed to both attenuator I and squarer. The signal amplitude is set to the desired value determined by external controls, by attenuator I. The output of attenuator I is a sine wave with desired amplitude and frequency.

The squarer generates a square wave of fixed amplitude and at the same frequency with the sine wave. The output of the squarer is fed to attenuator II which acts similar to attenuator I. The output is the square wave with amplitude and frequency which are determined externally.

Effect of Output resistance

As explained above, terminal voltage depends on source voltage (Vs(t)) and terminal current (i(t)). For example given Vs(t)=20sin(wt) Volts, and Rs=50 Ohm, terminal voltage for a 10 KOhm load is 19.9 sin(wt) Volts, but if a load of 50 Ohm is connected, terminal voltage drops to 10sin(wt) Volts. The terminal voltage will be closer to the Vs(t) if the resistance of the load is higher. The value of the output resistance is mostly written on the front panel of the signal generator, and its effects should be considered when dealing with low circuits with low resistance.

In most of the digital signal generators the user can specify the resistance of the circuit to be connected to the signal generator (default value for Agilent 33220A is 50 Ohm). However, changing this value does not affect the output resistance of the signal generator, but rather informs the signal generator, about the circuit in order to modify its output so that the user selected waveform properties (amplitude) can be seen between the output terminals. Also when dealing with circuits with high resistance, the existence of the output resistance of the signal generator as a very small effect on the value of the terminal voltage as seen in the previous example. In such cases the signal generator can be used in High-Z mode. In this case, if the connected circuit is specified to have a large resistance value, source voltage is directly set to the desired terminal voltage.

For example if a terminal voltage of 5sin(wt) Volts is desired when connecting the signal generator (output resistance of the signal generator is 50 ohm) to a 25 Ohm circuit. When the resistance value of the circuit is specified, the signal generator adjusts its source voltage, Vs(t) to 15sin(wt) Volts, in order to output the desired voltage between the terminals. Using the same settings, if the signal generator is connected to a 10 KOhm circuit the voltage between the output terminals will be approximately 15sin(wt) Volts. If the signal generator is set to its High-Z mode the voltage between the output terminals will be approximately 5sin(wt) Volts.

Grounded Output – Floating Output

The signal generators can be separated into two categories depending on their output characteristics: Floating output signal generators and grounded output signal generators.

Grounded Output Signal Generators: One of the output terminals is ground, and should be connected to the ground of the circuit (or ground probes of other instruments in the circuit). The value of the other output terminal is the desired output voltage with respect to the ground.

Floating Output Signal Generators: There is no necessity to connect one of the output terminals to ground. Actually floating output signal generators can be connected in series to other signal generators or power supplies. The voltage in one output terminal is the desired voltage with respect to the other terminal, whatever the value of the other terminal is.

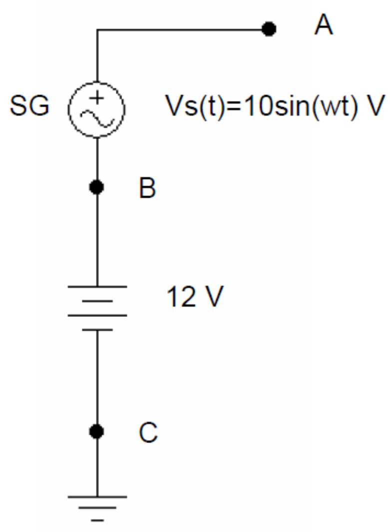


Figure 78.

Consider the circuit shown in Figure 78. A signal generator (SG) is connected in series to a 12 Volt power supply. If the signal generator is a grounded output signal generator, one of the output terminals should be ground terminal. If the ground terminal is connected to B then VBC=0 as both B and C are grounded (0 V). And VAC= 10sin(wt) Volts, the power supply has no function. But if a floating output signal generator is used VAC would be 12+10sin(wt) Volts.

LAB EXPERIMENTS

EXPERIMENT - I	**:** Serial and Parallel Connected Resistor Applications
EXPERIMENT - II	**:** Serial and Parallel Connected Resistor Applications
EXPERIMENT - III	**:** Node and Mesh Current Analysis Methods
EXPERIMENT - IV	**:** Thevenin-Norton Teorems and Maximum Power Transfer
EXPERIMENT - V	**:** Superposition Principle
EXPERIMENT - VI	**:** Natural Responses of RL and RC Circuits
EXPERIMENT - VII	**:** Step Responses of RL and RC Circuits
EXPERIMENT - VIII	**:** Natural Response of Parallel R-L-C Circuit
EXPERIMENT - IX	**:** Step Response of Parallel R-L-C Circuit
EXPERIMENT - X	**:** Natural and Step Responses of Serial R-L-C Circuit
EXPERIMENT-XI	**:** Design Experiment

EXPERIMENT I : Serial and Parallel Connected Resistor Applications

INTRODUCTION: As discrete components, resistors come in various sizes and shapes depending on their power rating and use. The resistive element material may also vary: metallic wire, carbon, etc. The resistor most commonly used in the laboratory is made of carbon. Some resistors may have their nominal ohmic value stamped on the body of the resistor. More often, however, color code is used to indicate the value between 0 and 9.

MEASURING RESISTANCE WITH OHMMETER

1. Isolate the resistance.
2. Select the ohm function.
3. Connect the test leads across the component.
4. If the ohmmeter is not auto-ranging, start on the highest rank and switch down to lower
5. ranges until an in-range reading is obtained.
6. Do not leave your lingers on the resistor lead while you are making measurements.
7. Doing so will cause readings of high value resistors to be lower than they really are.

PURPOSE:

1. Study types of resistor connections

2. Study the limitations of a Voltage Source.

PRELIMINARY STUDY:

Formulate the values requested for circuits below and calculate each values. Compare the calculated values and obtained ones from the measurements during the experiment. Explain and interpret the reasons of differences between the values calculated and measured in the experiment report.

EXPERIMENT STUDY:

1. Determine the voltage, current and power values for each elements in the circuit given in Figure 1.

R1	R2	V1	I1	V2	I2	Pin	P1	P2
1K	1K							
1K	2K							
2K	1K							
10K	100							
1K	1M							

2. Determine the voltage, current and power values for each elements in the circuit given in Figure 2.

R1	R2	V1	I1	V2	I2	Pin	P1	P2
1K	1K							
1K	2K							
2K	1K							
10K	100							
1K	1M							

3. Determine the voltage, current and power values for each elements in the circuit given in Figure 3.

R1	R2	V1	I1	V2	I2	Pin	P1	P2
1K	1K							
1K	2K							
2K	1K							
10K	100							
1K	1M							

4. Determine the voltage, current and power values for each elements in the circuit given in Figure 4.

R1	R2	V1	I1	V2	I2	Pin	P1	P2
1K	1K							
1K	2K							
2K	1K							
10K	100							
1K	1M							

CIRCUIT DIAGRAMS

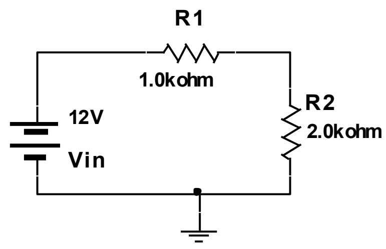


Figure 1- Resistors Series Connected to Voltage Source

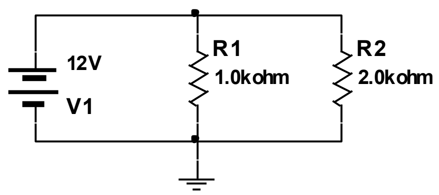


Figure 2 - Resistors Parallel Connected to Voltage Source

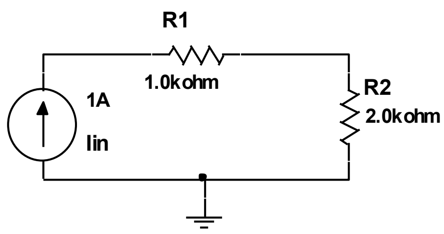


Figure 3 – Resistors Series Connected to Current Source

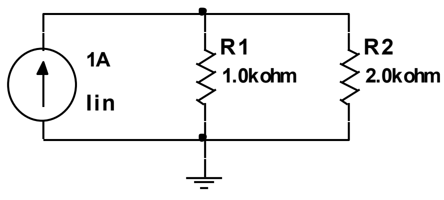


Figure 4 – Resistors Parallel Connected to Current Source

CONCLUSION AND COMMENT:

1. Report all measurements and operations done in the experiment section. Add your comments.
2. Compare the values obtained from measurements with theorical calculations.

EXPERIMENT II : Serial and Parallel Connected Resistor Applications

PURPOSE: Study complex connections of resistor

PRELIMINARY STUDY:

Formulate the values requested for circuits below and calculate each values. Compare the calculated values and obtained ones from the measurements during the experiment. Explain and interpret the reasons of differences between the values calculated and measured in the experiment report.

EXPERIMENT STUDY:

1. Determine the values of I_1, I_2, I_3, V_1, V_2 and V_3 using resistor values given in the table below for V_{in} = 10.

R_1	R_2	R_3	V_1	V_2	V_3	I_1	I_2	I_3
1k	2k	2k						
1k	2k	4k						
4k	2k	2k						
2k	2k	2k						
1k	4k	2k						
4k	4k	2k						

Draw the circuit diagram you formed using Pspice for obtaining the measurements.

2. For given circuit in Figure-2, find the power consumed on the R_3 resistor with using the values of V_{in} = 12V, I_y = 1A, R_1 = 1Ω, R_2 =1Ω and R_3 = 2 Ω. Draw the Pspice circuit diagram you used for determining this power.

CIRCUIT DIAGRAMS :

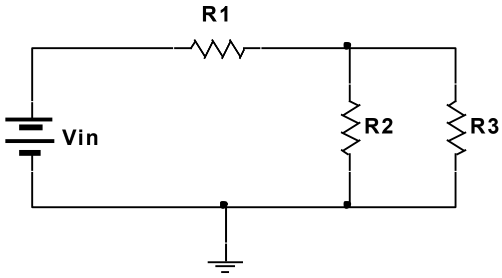


Figure 1- Resistors Complex Connected to Voltage Source

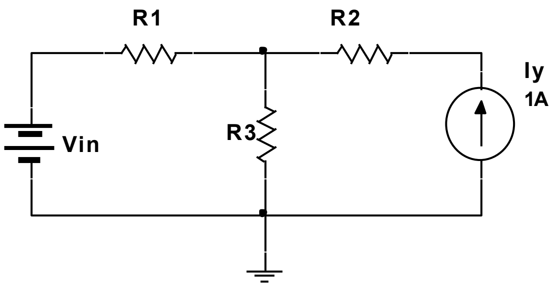


Figure 2- Usage of Voltage and Current Sources Together

CONCLUSION AND COMMENT:

1. Report all measurements and operations done in the experiment section. Add your comments.
2. Compare the values obtained from measurements with theorical calculations.

EXPERIMENT III: Node and Mesh Current Analysis Methods

INTRODUCTION:

Nodes

A node is a section of a circuit which connects components to each other. All of the current entering a node must leave a node, according to Kirchoff's Current Law (KCL). Every point on the node is at the same voltage; no matter how close it is to each component, because the connections between components regarded as perfect conductors. This voltage is called the node voltage, and is the voltage difference between the node and an arbitrary reference point. The reference point is a node which is defined as having zero voltage. The ground node should be chosen carefully for convenience. Note that the reference node does not necessarily represent an actual connection to ground. For example, if a node has a voltage of 5 Volts, then the voltage drop between that node and the reference node will be 5 Volts.

Nodal Analysis

Nodal analysis is a formalized procedure based on KCL equations.

Steps:

1. Identify all nodes.
2. Choose a reference node. Identify it with reference symbol. A good choice is the node with the most branches, or a node which can immediately give you another node voltage (e.g., below a voltage source).
3. Assign voltage variables to the other nodes (these 3. are node voltages.)
4. Write a KCL equation for each node (sum the currents leaving the node and set equal to zero). Rearrange these equations into the form A*V1+B*V2=C (or similar for equations with more voltage variables.)
5. Solve the system of equations from step 4. There are a number of techniques that can be used:

Also to solve an N mesh circuit, a set of N simultaneous equations are needed. There are several ways to derive a solution (i.e. Matrix algebra).

PURPOSE:

1. Solve a circuit using mesh analysis.
2. Solve a circuit using mesh analysis.

PRELIMINARY STUDY:

- **Node Analysis Method**

  Apply node analysis method for the circuit given in Figure 1. Assume that V_{s1}=20V and V_{s2}=9.6V. Resistor values are given in the table.

  1. Find V_4 and V_5 node voltages.
  2. Find the values of I_1, I_2, I_3, I_4 and I_5 currents using the node voltages that you found in step 1 and voltage sources values.
  3. Determine if the voltage sources give power for the circuit.

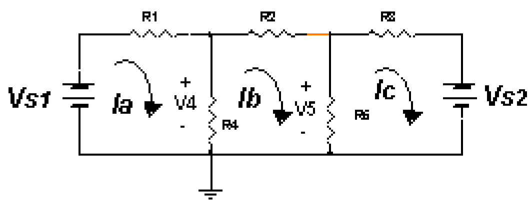


Figure 1- Experiment Circuit

- **Mesh Current Method**

  Apply mesh current analysis method for the circuit given in figure. Assume that V_{s1}=20V and V_{s2}=9.6V. Resistor values are given in the table.

  1. Find the I_a, I_b, and I_c mesh currents.
  2. Find I_1, I_2, I_3, I_4 and I_5 currents using the values you found in step 1.
  3. Determine if the voltage sources give power for the circuit.

EXPERIMENT STUDY:

Set up the circuit diagram in the figure for resistor values given in the table. Measure the V_4 and V_5 node voltages and I_1, I_2, I_3, I_4 and I_5 currents. Compare the values that you obtained from measurement with the values you obtained from theorical calculations. If there are differences, interpret them.

Table: Resistor values

Resistors	Values
R_1	2kΩ
R_2	5kΩ
R_3	2kΩ
R_4	20kΩ
R_5	10kΩ

Draw the circuit diagram you formed using Pspice for obtaining the measurements.

<u>CONCLUSION AND COMMENT:</u>

1. Report all measurements and operations done in the experiment section. Add your comments.
2. Compare the values obtained from measurements with theorical calculations.

EXPERIMENT IV: Thevenin-Norton Theorems and Maximum Power Transfer

PURPOSE:

To understand how to analyze a circuit using Thevenin's Theorem in a circuit, also the relation between Thevenin and Norton Theorems.

PRELIMINARY STUDY:

- **Thevenin – Norton Equivalent Circuits**

  Apply the Thevenin-Norton analysis methods for the circuit given in figure. Assume that V_{s1}=15V and V_{s2}=10V. Resistor values are given in the table.

  1. Find the Thevenin equivalent voltage (V_T) between A – B.
  2. Find the Thevenin equivalent resistor (R_T) between A – B.
  3. Find Norton equivalent current by using short circuit (I_{KD}) between A – B.

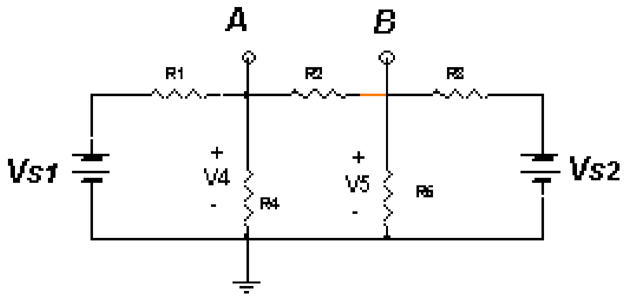


Figure 1- Experiment Circuit

- **Maximum Power Transfer**

  For the circuit shown in the figure if we defined a load using the Thevenin equivalent circuit as a R_{load} which can be installed between the ends of A-B, find the value of R_{load} to get maximum power of R_{load}. Assuming that a variable resistor is connected to circuit, for the values of R_{load} between 1kΩ to 10 kΩ increased by 1kΩ intervals and show on the table.

EXPERIMENT STUDY:

1. Set up the circuit diagram given in figure by using resistor values in the table.
2. Measure the open circuit voltage between A – B by using voltmeter (V_T).
3. Measure the short circuit current between A – B by using ampermeter. (I_{KD})
4. Be sure that the voltage the sources are short circuit and current sources are open circuit in the circuit. Then measure the resistor between A-B by using ohmmeter. Observe that the value of equivalent resistor (R_T) is (V_T) / (I_{KD}).
5. Place an adjustable resistor between points A – B. Set the value of this resistor from 1 kΩ to 10 kΩ and measure the voltages between A – B. Determine the power transmitted to these resistor values by using voltage values you found and resistor values you used.
6. Draw the resistor-power graphic according to the resistor values.
7. Compare the values that you obtained from measurement with the values you obtained from theorical calculations. If there are differences, interpret them.

Table: Resistor values

Resistors	**Values**
R_1	4.7 kΩ
R_2	20 kΩ
R_3	10 kΩ
R_4	36 kΩ
R_5	5.1 kΩ

8. Draw the circuit diagram you formed using Pspice for obtaining the measurements.

CONCLUSION AND COMMENT:

1. Report all measurements and operations done in the experiment section. Add your comments.
2. Compare the values obtained from measurements with theorical calculations.

EXPERIMENT V: Superposition Principle

PURPOSE:

1. To apply the superposition principle to a linear circuit.
2. To learn how to obtain equivalent circuits.

THEORY:

A linear circuit is composed entirely of independent sources, linear dependent sources, and linear elements. A linear circuit element is one which has a linear voltage-current relationship. The superposition principle is expressed as follows:

In any linear circuit containing several sources, the voltage across (or current through) any circuit element may be obtained by adding algebraically the individual voltages (or currents) caused by each independent source acting alone, with the other independent voltages deactivated (voltage sources replaced by short circuits and current sources replaced by open circuits).

PRELIMINARY STUDY:

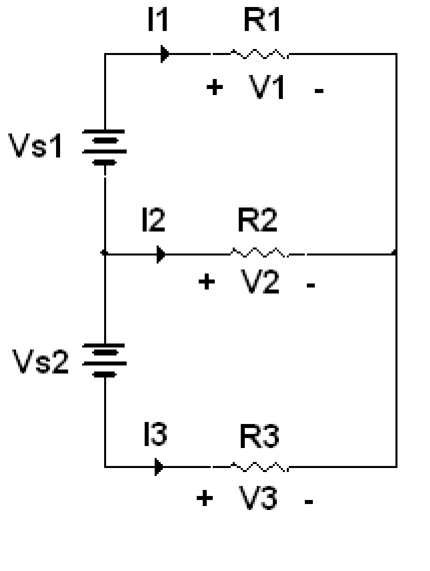


Figure 1 – Circuit of Experiment 6.

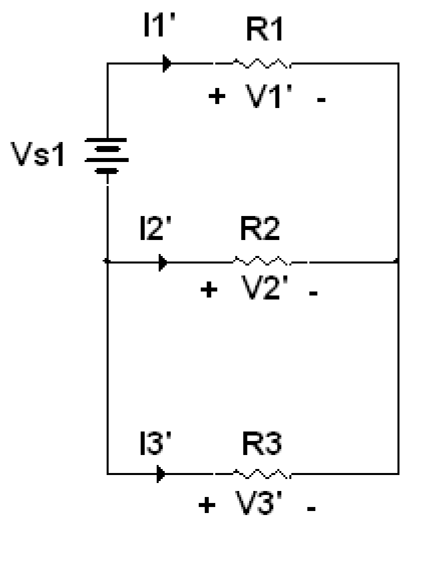


Figure 2 – Circuit without Vs2.

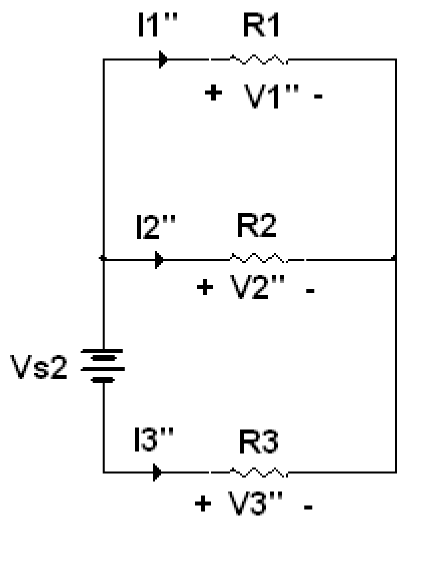


Figure 3– Circuit without Vs1.

Table: Resistor values

Resistors	Values
R_1	5 kΩ
R_2	10 kΩ
R_3	30 kΩ

Superposition principle will be applied for circuit given in Figure 1. V_{s1} and V_{s2} are assumed to be 15V and 10V respectively. Resistor values are given in the table above.

1. Calculate the V_1, V_2 and V_3 voltages and I_1, I_2 and I_3 currents given in Figure 1. For this calculations you can use one of the methods which are node or mesh current analyses except superposition principle.
2. Replace the Vs2 voltage source with a short circuit in Figure 1 and therefore obtain Figure 2. Calculate the V_1', V_2' and V_3' voltages and I_1', I_2' and I_3' currents for the circuit given in Figure 2.
3. Replace the Vs1 voltage source with a short circuit in Figure 1 and therefore obtain Figure 3. Calculate V_1'', V_2'' ve V_3'' voltages and I_1'', I_2'' ve I_3'' currents given in Figure 3.
4. Compare the values found in second and thirth steps with the values found in first step. (X ?= X' + X'')

EXPERIMENT STUDY:

Application Circuit

1. Implement the circuit diagram given in Figure 1 using the resistor values. Take the values of V_{s1} and V_{s2} voltage sources as given prelab.
2. Measure the voltages V_1, V_2 and V_3 and currents I_1, I_2 and I_3.

Application Circuit without V_{s2}

1. Implement the circuit diagram given in Figure 2 using the resistor values in table. Take the values of V_{s1} voltage source as given prelab.
2. Measure the voltages V_1', V_2' and $V_3'$and currents I_1', I_2' and I_3' .

Application Circuit Without V_{s1}

1. Implement the circuit diagram given in Figure 3 using the resistor values in table. Take the values of V_{s2} voltage source as given prelab.
2. Measure the voltages V_1'', $V_2''$and V_3'' and currents I_1'', I_2'' ve I_3''.

Analysis

1. Observe that sum of the values obtained from step B and step C is equal to the values obtained from step A.
2. Compare the total values from theorical calculations with the values obtained from measurements.
3. Interpret if superposition principle is applicable for circuit analysis.
4. Draw the circuit you formed by Pspice for obtaining these measurements.

CONCLUSION AND COMMENT:

1. Report all measurements and operations done in the experiment section. Add your comments.
2. Compare the values you obtained from theorical calculations with measurements.

EXPERIMENT VI. : Natural Response of RL and RC Circuits

INTRODUCTION: Natural response of RL and RC circuits (suddenly disconnected from a DC source) : Consider current and voltage that arise when stored energy in an inductor or capacitor is suddenly released to a resistive network.

PURPOSE: The purpose of this experiment, to examine the behavior of RL and RC circuits under DC voltage. In this experiment, practice-oriented techniques for the detection of an unknown circuit parameters will be introduced.

PRELIMINARY STUDY:

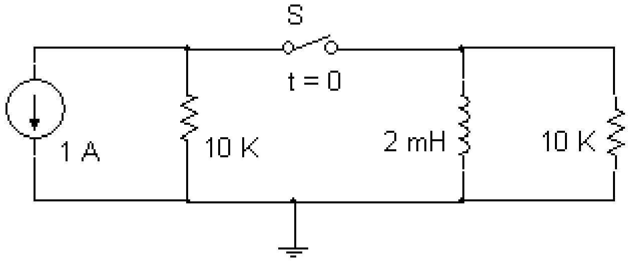


Figure 1– RL Application Circuit

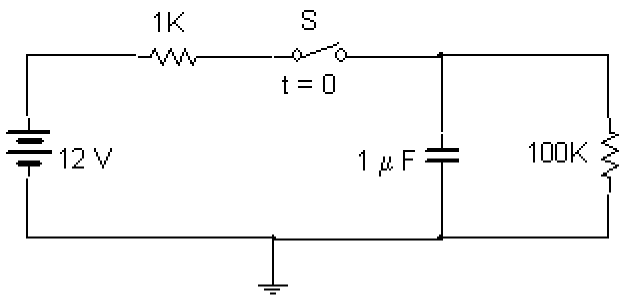


Figure 2– RC Application Circuit

At Figure 1 and Figure 2, S switches are shown in the circuits, on closed position for a long time, and at the time of (t = 0) assumed that switches are opening.

- For the RL circuit,
  1. For ($t = 0^-$) find the values of the voltage and current to the coil.
  2. For ($t > 0$) calculate the time constant.

3. For ($t > 0$) write on the coil voltage and current expressions.

- For the RC circuit,

  1. For ($t=0^-$) find the values of current and voltage on the capacitor.

  2. For ($t > 0$) calculate the time constant.

  3. For ($t > 0$) write on the capacitor voltage and current expressions.

EXPERIMENT STUDY:

RL Circuit

1. Set the circuit diagram shown in Figure 1. Draw the circuit diagram which is set on MultiSim.
2. Observe changing in the voltage on the coil by using an oscilloscope and current changing by using an ammeter.
3. Draw voltage-time graph on a graph paper which you saw on the oscilloscope.
4. Draw the current-time graph on the graphpaper with using current rates that you get.
5. Find the time constant from voltage graph and compare with theoretical value.

RC Circuit

1. Set the circuit diagram shown in Figure 2. Draw the circuit diagram which is set on MultiSim.

2. Observe changing in the voltage on the capacitor using an oscilloscope and current changing using with an ammeter.

3. Draw voltage-time graph on a graph paper which you saw on the oscilloscope.
4. Draw the current-time graph on the graphpaper with using current rates that you get.
5. Find the time constant from voltage graph and compare with theoretical value.

CONCLUSION AND COMMENT:

1. Report all processes and measurements made in the experiment. Add your comment.
2. Calculate theoretical calculations of circuit and compare them with measuraments.

EXPERIMENT VII. : Step Responses of RL and RC Circuits

INTRODUCTION: Step response of RL and RC circuits (suddenly connected to a DC source) :Step response of RL and RC circuits, DC voltage or current source is obtained from the reaction to the sudden granting of the circuit.

PURPOSE: The purpose of this experiment is to investigate the behavior of RL and RC circuits under DC voltage. In this experiment, practice-oriented techniques for the detection of unknown circuit parameters will be introduced.

PRELIMINARY STUDY:

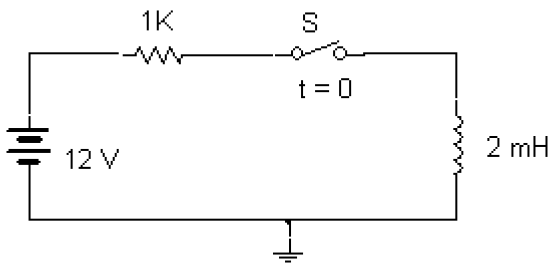


Figure 1 – RL Application Circuit

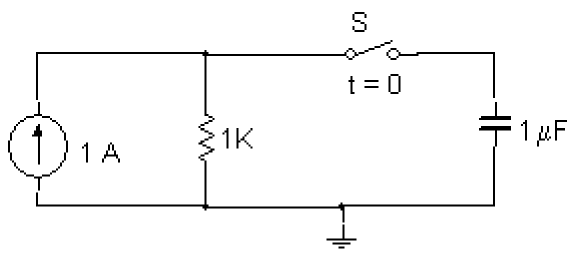


Figure 2 – RC Application Circuit

At Figure 1 and Figure 2 S switches are shown in the circuits, on closed position for a long time, and at the time of (t = 0) assumed that switches are opening.

- For the RL circuit,
  1. For ($t = 0^-$) find the values of the voltage and current to the coil.

2. For (t > 0) calculate the time constant.

3. For (t > 0) write on the coil voltage and current expressions.

- For the RC circuit,

1. For ($t=0^-$) find the values of current and voltage on the capacitor.

2. For (t > 0) calculate the time constant.

3. For (t > 0) write on the capacitor voltage and current expressions.

EXPERIMENT STUDY:

RL Circuit

1. Set the circuit diagram shown in Figure 1. Draw the circuit diagram which is set on Pspice.

2. Observe changing in the voltage on the coil using an oscilloscope and current changing using with an ammeter.

3. Draw voltage-time graph on a graph paper which you see on the oscilloscope.

4. Draw the current-time graph on the graph paper with using current rates that you get.

5. Find the time constant from voltage graph and compare with theoretical value.

RC Circuit

1. Set the circuit diagram shown in Figure 2. Draw the circuit diagram which is set on Pspice.

2. Observe changing in the voltage on the capacitor using an oscilloscope and current changing using an ammeter.

3. Draw voltage-time graph on a graph paper which you see on the oscilloscope.

4. Draw the current-time graph on the graph paper with using current rates that you get.

5. Find the time constant from voltage graph and compare with theoretical value.

CONCLUSION AND COMMENT:

1. Report all processes and measurements obtained in the experiment. Add your comments.

2. Calculate theoretical calculations of circuit and compare them with measurements.

EXPERIMENT VIII: Natural Response of Parallel R-L-C Circuit

PURPOSE: The objective of this experiment is to examine the natural responses of RLC circuits when DC voltage is applied.

PRELIMINARY STUDY:

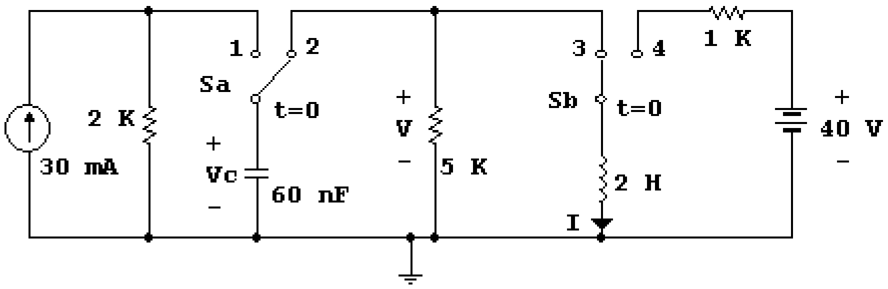


Figure 1– RLC Application Circuit

It is assumed that position of Sa switch is “1” and position of Sb switch is “4” for a long time in the figure above. And also (t = 0) time positions of Sa and Sb switches are changed as “2” and “3” respectively.

3. Determine the initial values of V_C and I. ($t = 0^-$)
4. Determine the roots of characteristic function for $t > 0$.
5. Find the response type of the circuit .
6. Find the v(t) function. ($t >= 0$)

EXPERIMENT STUDY:

1. Realize the steps 1 and 4 of the prelab.
2. Draw the Pspice circuit diagrams.

CONCLUSION AND COMMENT:

1. Report all measurements and operations done in the experiment section. Add your comments.
2. Compare the values you obtained from theorical calculations with measurements.

EXPERIMENT IX. : Unit Step Response of Parallel R-L-C Circuit

PURPOSE: The purpose of this experiment, to examine the unit step responses of RLC circuits under DC voltage. In this experiment, it will be used to implement techniques determining unknown parameters of the circuit

PRELIMINARY STUDY:

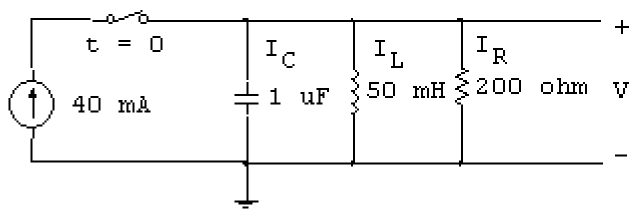


Figure 1– RLC Application Circuit

At Figure 1, switch is shown in the circuits, on closed position for a long time, and at the time of (t = 0) assumed that switch is opening.

1. Find the initial values of I_C, I_L and I_R currents. ($t = 0^+$)
2. Find the roots of the characteristic function.
3. Find the functions V, I_C, I_L ve I_R and draw graphs. ($t >= 0$)
4. Find the type of response of the circuit.

EXPERIMENT STUDY:

1. Carry out steps 1 and 3 of the preliminary study.
2. Draw a Pspice circuit diagrams have installed.

EXPLANATION:

All graphics will be drawn on graph paper.

CONCLUSION AND COMMENT:

1. Report all processes and measurements made in the experiment. Add your comment.
2. Calculate theoretical calculations of circuit and compare them with measuraments.

EXPERIMENT X: Natural And Unit Step Responses Of Serial R-L-C Circuit

PURPOSE: The purpose of this experiment, to examine the natural and unit step responses of RLC circuits under DC voltage. In this experiment, it will be used to implement techniques determining unknown parameters of the circuit.

PRELIMINARY STUDY:

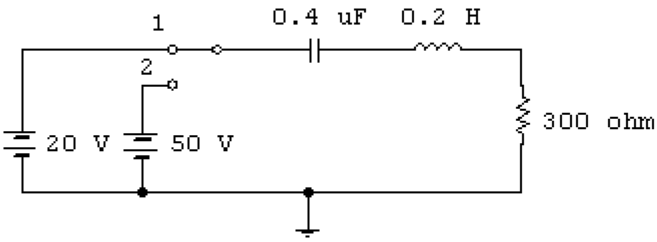


Figure 1– RLC Application Circuit

Assume that switch in the figure is on position "1" for a long time and then it is taken to position "2" at time (t = 0).

1. Find the initial values V_C, V_L and V_R voltages. ($t = 0^+$)
2. Determine the roots of characteristic function.
3. Find the I_C, V_C, V_L and V_R functions and draw graphics. ($t >= 0$)

EXPERIMENT STUDY:

1. Implement the step 1 and 3 in the prelab.
2. Draw the Pspice circuit diagrams which you set up.

CONCLUSION AND COMMENT

1. Report all measurements and operations done in the experiment section. Add your comments.
2. Compare the values you obtained from theorical calculations with measurements.

EXPERIMENT XI : Design Experiment

The problem is to design a simple circuit using a thermistor to measure the temperature of a tank used to store volatile liquids. The measurement is to be accurate over a temperature range of 0°C to 50°C. The temperature must be measurable in an instrumentation room located a safe distance form the storage tank. A thermistor is a temperature sensitive resistor that can be used in a voltage divider circuit to obtain an output voltage V_T that is functionally related to the thermistor temperature T as shown in the Figure 1.

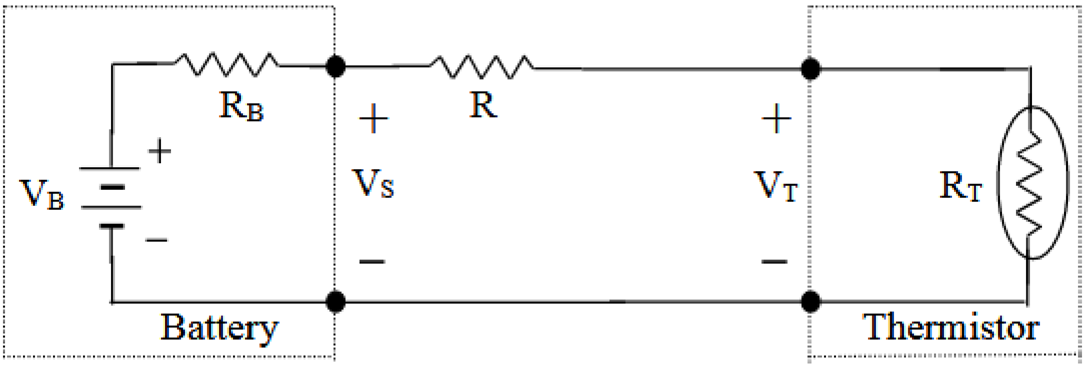


Figure 1 – Thermistor Circuit

Thermistors with a negative temperature coefficient typically have a resistance versus temperature of the form,

$$R_T = R_0 e^{-\alpha(T-T_0)} \qquad (1.1)$$

An electronic supply catalog shows thermistors with the tabularized characteristics to be available. Note that T_0 = 25°C for these thermistors, i.e., R_0 =R_T (25° C).

Resistance Ratio (RR)	$R_T(\Omega)$ @ 25° C.
9.0	2.2K
9.0	6.8K
7.2	12K
7.2	22K
9.2	33K
9.4	47K
9.9	68K
9.9	100K
10.0	150K
12.0	470K

The Resistance Ratio (RR) shown in the table is (R_T at 0°C)/(R_T at 50°C), i.e.,

$$RR = R_T(0)/R_T(50) \tag{1.2}$$

Assume that a 47 KΩ thermistor from the above table is bonded to the storage tank in order to be measure tank temperature and that this bond has high thermal conductivity so that the thermistor temperature is the same as the tank temperature. Also, assume that electrical heating of the thermistor due to current is negligible and that the wires connecting the instrumentation room components to the thermistor are large enough that their resistance is negligible.

Assume that a Six Volt lead acid cell battery is available as a source and that the battery has a nominal terminal open circuit voltage (V_B) of 6.3 volts and an internal resistance (R_B) of 3 ohms. Additional constraints are that the battery has a 1 amp-hour rating, that the circuit must operate for 1 year without battery replacement, and that the series resistor R must be selected from standard values for 5% resistors as shown in **Standard Resistor Value Multipliers**.

Assume that a portable, battery powered, voltmeter with a 1MΩ input resistance is to be used to measure V_S and V_T. In your analysis, consider using a Thevenin equivalent circuit in order to incorporate the effect of this voltmeter on the circuit. Design the circuit, i.e., select a series resistor R such that a plot of V_T versus T_T has a relatively small deviation from linearity over the temperature range of 0° to 50°C. Be sure that good sensitivity to temperature is obtained by the design, i.e., the range

of voltage over the full range of temperature is sufficient to allow it to be accurately measured. A 50% voltage swing is considered adequate.
Once the design is established, find a linear equation for use in calculating tank temperature T_T from the voltage reading V_T. Note that in order to minimize errors due to any change in battery voltage, this equation should have the form,

$$T_T = T_1 + K_V V_T / V_S \quad (1.3)$$

Standard Resistor Value Multipliers

These multiplier values in the table apply to all ± 5% tolerance resistors. Resistors with ± 10% tolerance are available only in values marked by a *. The multipliers are:

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

The multipliers in the table apply for nominal resistor values of 10nΩ. For example, using the 1.1 multiplier, you can get standard 5% resistors with values of 11Ω, 110Ω, 1.1kΩ, 11kΩ, 110kΩ, 1.1MΩ, etc. Also, while you can choose 5% resistors with nominal values of 10Ω (1.0*10Ω) or 11Ω (1.1*10Ω) or 12Ω (1.2*10Ω), you cannot choose a 10% resistor with a nominal value of 11Ω (1.1*10Ω) since this value is within the tolerance limit of both the 10Ω and the 12Ω resistors.

Document your design as follows:

1. Clearly define the value selected for the series resistor R.
2. Clearly define T_1 and K_V used in the linear equation (1.3) to calculate T_T .
3. Briefly describe the process you used to arrive at a solution to the design problem. Give any additional assumptions made in arriving at the solution.

4. Provide a plot of V_T versus T_T using the nominal component values for your final design. Also provide a plot of the deviation in V_T from linearity. Use Excel, MATLAB, or any other mathematical computer program to generate these plots.
5. Identify the maximum deviation in the voltage reading V_T over the full temperature range and convert this to a maximum error in measured temperature.