



**The Hashemite University**  
**Faculty of Engineering**  
**Electrical and Computer Engineering Department**

**(Electrical Circuit Lab)**  
**Lab manual**

**Laboratory (0404214)**

**Experiments:**

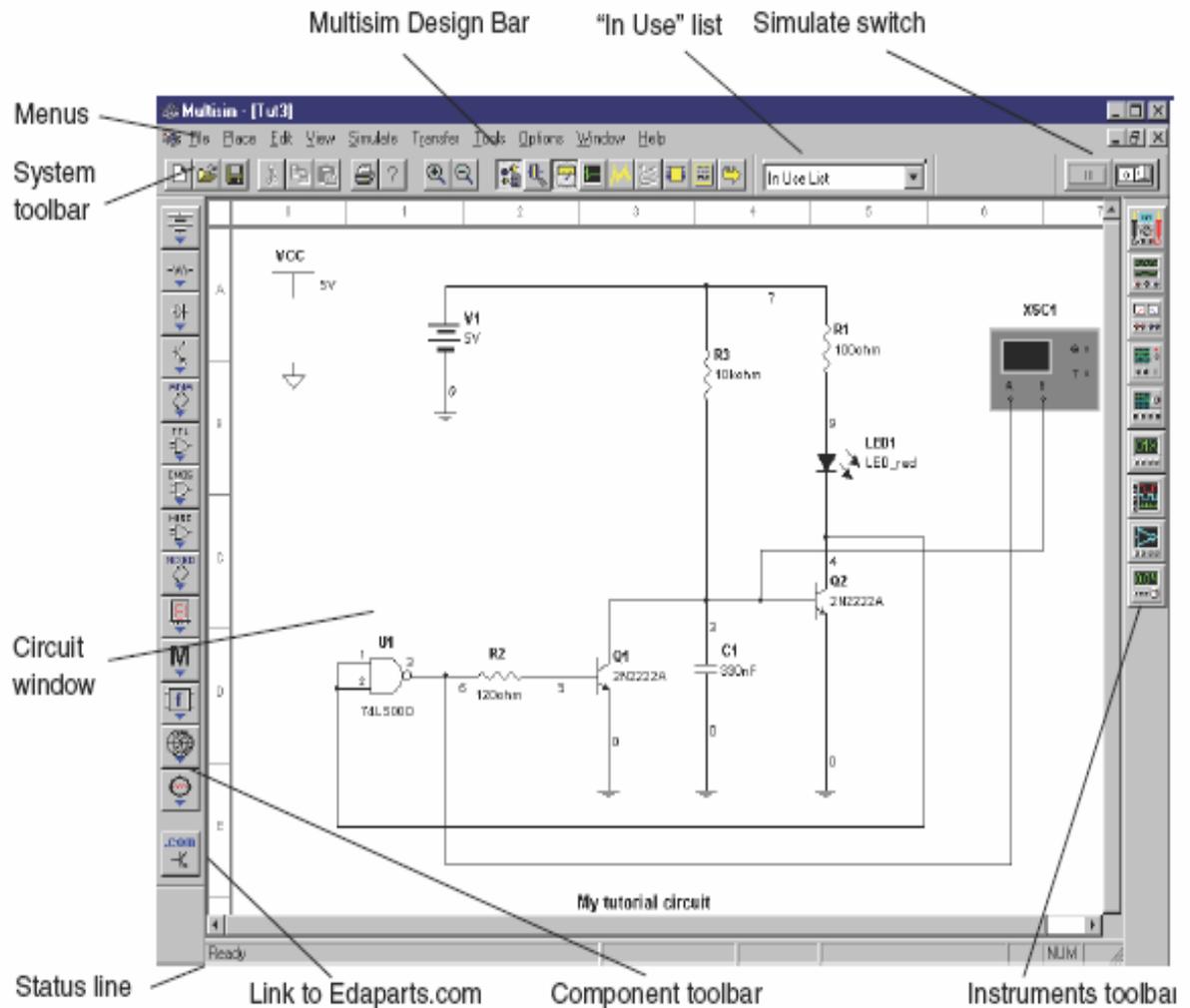
1. Lab. Equipment Familiarization
2. Measurement on DC Circuit
3. Techniques of Circuit Analysis (1)
4. Techniques of Circuit Analysis (2)
5. The Function Generator & Oscilloscope.
6. Basic Laws on AC circuits.
7. Step Response of AC Circuits.

# APPENDIX

## Introduction to the Multisim 2001 Interface

### Basic Elements

Multisim's user interface consists of the following basic elements:



**Note** Your circuit window may, by default, have a black background; however, for the purposes of this document, we show a white background. To change the background color, see “1.8.1 Controlling Circuit Display” on page 1-22.

**Menus** are, as in all Windows applications, where you find commands for all functions. The **system toolbar** contains buttons for commonly-performed functions. The **instruments toolbar** contains buttons for each instrument available with your edition of Multisim.

The link to **Edaparts.com** launches your Web browser loaded with the Electronic Workbench Edaparts.com Web site. You can then navigate the site to download parts. The **zoom toolbar** allows you to zoom in and out on the active circuit. The **Multisim Design Bar** is an integral part of Multisim, and is explained in more detail below.

The “**In Use**” list lists all the components used in the active circuit, for easy re-use. The **component toolbar** contains Parts Bin buttons that let you open Parts Bins (which, in turn, contain buttons for each family of components).

The **circuit window** is where you build your circuit designs. The **status line** displays useful information about the current operation and a description of the item the cursor is currently pointing to.

The **simulate switch** is an easy way to run, stop or pause the simulation of your circuit.

## Design Bar

The Design Bar is a central component of Multisim, allowing you easy access to the sophisticated functions offered by the program. The Design Bar guides you through the logical steps of building, simulating, analyzing and, eventually, exporting your design. Although Design Bar functions are available from conventional menus, this manual assumes you are taking advantage of the ease of use offered by the Design Bar.



The Component design button is selected by default, since the first logical activity is to place components on the circuit window.



The Component Editing button lets you modify the components in Multisim, or add components.



The Instrument design button is selected by default and lets you attach instruments to your circuit.



The Simulate button runs/pauses/stops the simulation. The green sine wave line moves while simulation is running.



The Analysis button lets you choose the analysis you want to perform on your circuit.



The Postprocessor button lets you perform further operations on the results of your simulation.



The VHDL/Verilog HDL button allows you to work with VHDL modeling (not available in all editions). For details on the functions associated with this button, see Chapter 10, “HDLs and Programmable Logic”.

## Customizing the Multisim 2001 Interface

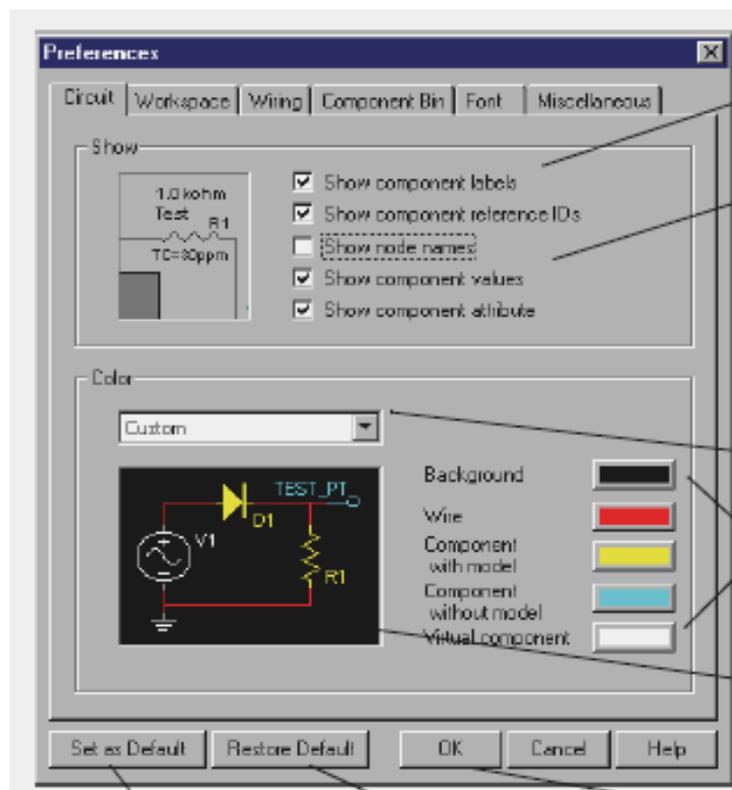
You can customize virtually any aspect of the Multisim interface, including the toolbars, colors in your circuit, page size, zoom level, autobackup interval, symbol set (ANSI or DIN) and printer setup. Your customization settings are saved individually with each circuit file you use, so you could, for example, have one color scheme for one circuit and another for a different circuit. You can also override the settings for individual instances (for example, change one particular component from red to orange) or for the entire circuit. Most customization is done through **Options/Preferences**. This command allows you to either change the settings for the current circuit file only, or create default settings that affect both the current circuit and all circuits you subsequently create. For example, if your current circuit shows component labels and you save those settings as the default, when you choose **File/New**

and create a new circuit, that circuit will be set to show component labels as well. Changing the default settings will **not** affect circuits you have already created and saved. However, if your current circuit shows component labels and you simply change it to hide such labels, the change affects only your current circuit.

## Controlling Circuit Display

You can control the way your circuit and its components appear on the screen, and the level of detail which appears.

 To set the default circuit display options for circuits, choose **Options/Preferences**. The Preferences screen appears, offering you six tabs of options, with the Circuit tab being the active tab. Use this tab to control the colors and display details for your circuit.



Shows the results of enabling the options on the right.

Enable those items you want shown. You can override your choices for a particular component, as described in "3.9.1 Displaying Identifying Information about a Placed Component" on page 3-18.

Select a built-in color scheme.

Set up a custom color scheme (only if "Custom" selected as scheme).

Shows the currently selected color scheme.

Saves the selected settings for current and subsequent circuits.

Restores the default settings to previously saved ones.

Saves the settings for the current circuit only.

For other window display options, choose the desired tab. For example, to define whether grid, page bounds and titles blocks are shown, use the Workspace tab. To set the component symbols to either ANSI or DIN, use the Component Bin tab. Experiment with these options as you wish.

 To save your options for the current circuit *only*, click **OK**.

 To save your options as the default, affecting both the current and subsequent circuits, click **Set as Default**, then click **OK**.

 To restore your previously saved settings (if you don't like the changes you made and would like to start over again), click **Restore Default** and then continue making changes.

 To cancel your changes and close the screen, click **Cancel**.

## Other Customization Options

You can also customize the interface by showing or hiding, dragging to a new location and, optionally, resizing any of the following:

- System toolbar
- Zoom toolbar
- Design Bar
- “in use” list
- Instruments toolbar
- Simulation switch

These changes apply to all circuits you are working with. Moved or resized items will return to that location and size when Multisim is next opened. Finally, you can use the **View** menu to display or hide various elements.

## Building a Circuit

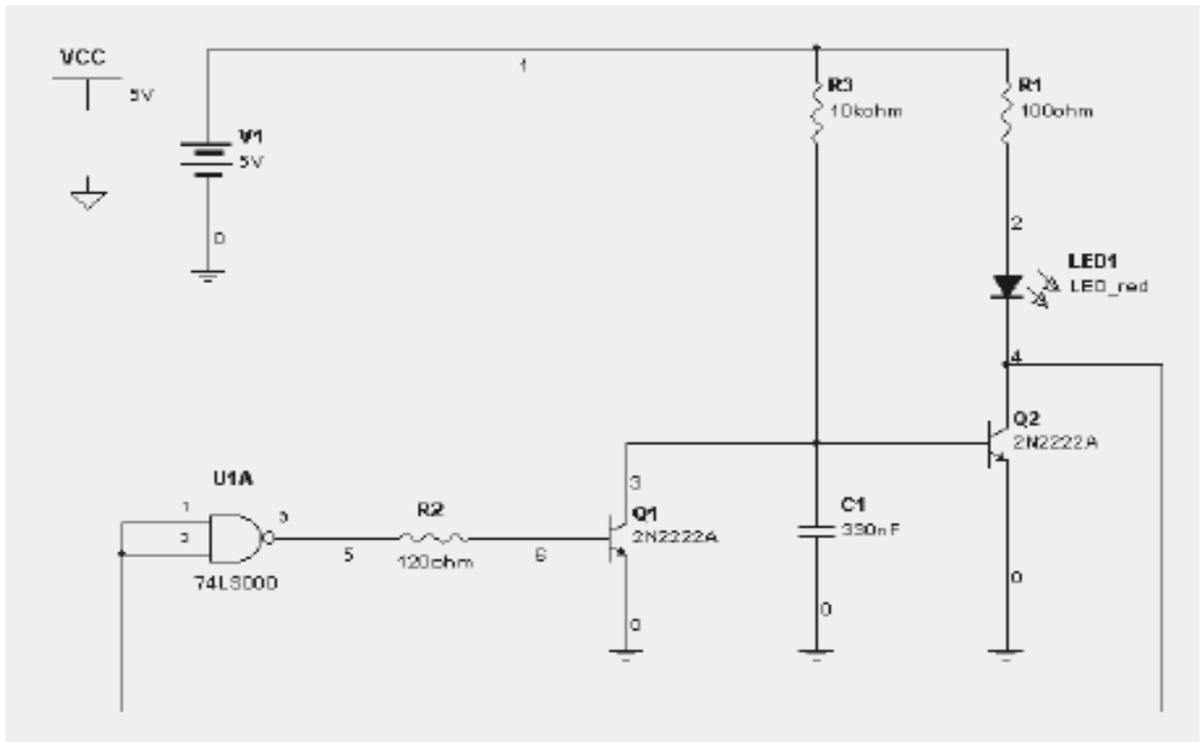
### About this Chapter

This chapter describes how to place and wire together the components of the circuit you will work with for the rest of this manual.

### Introduction

This manual takes you through the process of building and simulating a simple circuit. The first stage is to choose the components you want to use, place them on the circuit window in the desired position and orientation, wire them together, and otherwise prepare your design. The circuit you will build in this chapter is a simple circuit that flashes an LED on and off. When you have completed the steps in this chapter, you will have a circuit that looks like this:

This circuit file, in its various stages of development, is shipped with Multisim. Throughout this manual you will be given detailed instructions on how to build this circuit. However, you will also find references to the files you can open to experiment with the circuit at its different stages, if you prefer.



## Starting your Circuit File

To start building your circuit file, just run Multisim. Multisim automatically opens a blank circuit file with which you can work. The circuit window you see when you run Multisim has its color, size, and display options based on the previously set default settings. You can change these options to suit your needs by using the pop-up menu, as described on “1.8.1 Controlling Circuit Display” on page 1-22, or by consulting the *Multisim 2001 User Guide*.

## Placing Components on the Circuit Window

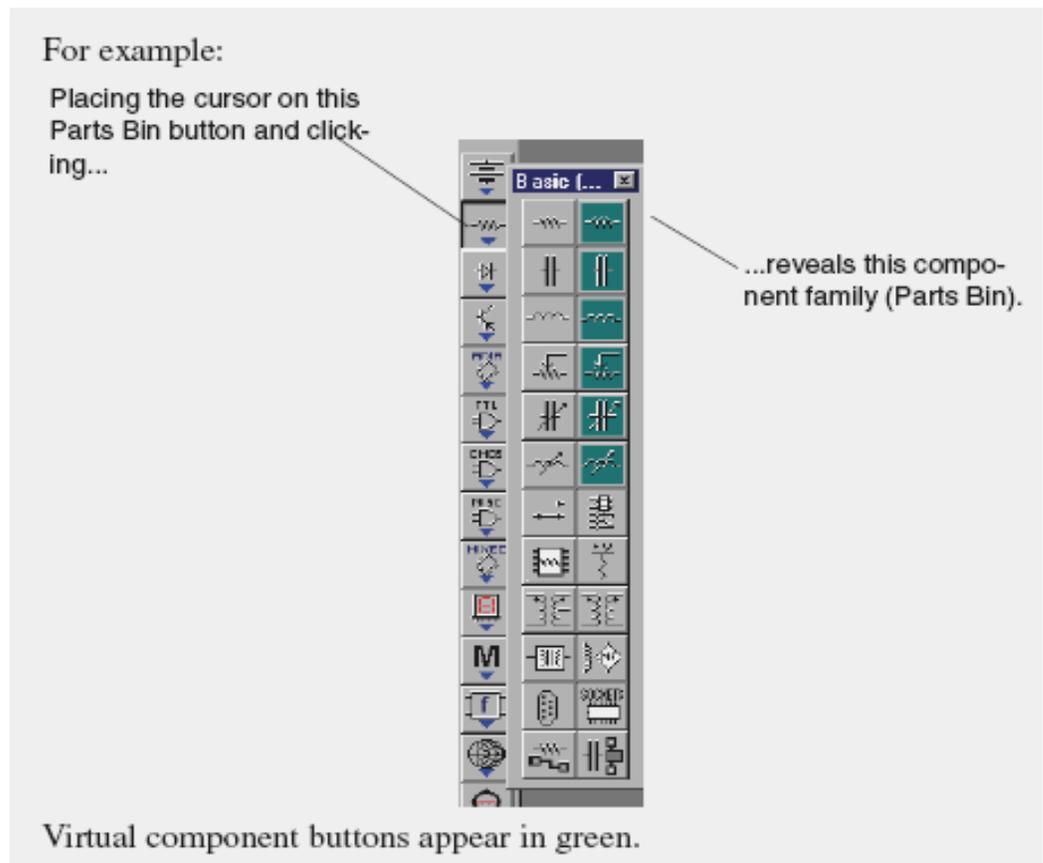
You are now ready to start placing the components for your circuit. As explained in the *Multisim 2001 User Guide*, Multisim offers up to three component databases (“Multisim Master”, “User” and, for some Multisim editions, “Corporate Library”). For the purposes of this tutorial, we are concerned only with the “Multisim Master” database, which is the component database shipped with Multisim. For more about the other database levels, see the *Multisim 2001 User Guide*.

## About the Component Toolbars

By default, the **Components** Design Bar button is selected and the component toolbar is visible. If it is not visible, click the **Components** button from the Design Bar.



The components you need to use to create circuits are grouped into logical divisions or Parts Bins. Each Parts Bin is represented by a button on the component toolbar. Placing your cursor over one of these buttons and clicking opens the corresponding Parts Bin, containing a button for each component family. The behavior of the Component toolbar and its Parts Bins is affected by your circuit’s user preferences. For more details see the *Multisim 2001 User Guide*.



**Note** The behavior of the Component toolbar is controlled by the settings in the Component Bin tab of the Preferences screen, which can be opened using **Options/Preferences**. Use the Component Bin tab to, for example, set the Parts Bin to appear when you click on a Parts Bin button and remain open even after a component is selected. For more information on these settings, see the *Multisim 2001 User Guide*.

## Placing Components

This section explains how to place components using the component toolbar and its Parts Bins. This is the typical way of placing components. As explained in the *Multisim 2001 User Guide*, you can also place components using **Place/Place Component**, which is useful if you are not sure which Parts Bin contains the component you are interested in placing.

### Placing the First Component

#### Step 1: Place a Battery

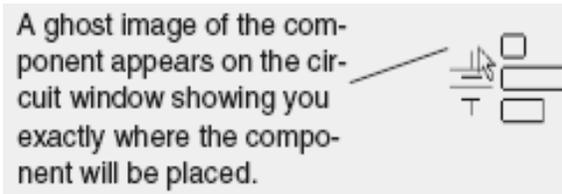
To place the first component (a 5 volt battery):

1. Place the cursor on the **Sources** Parts Bin button and click. The contents of the Sources Parts Bin appear:

**Tip** Move your cursor over any Parts Bin button or Component family button to see its name.



2.  Click the **DC Voltage Source** button and move your cursor to the circuit window. Your cursor changes to indicate a part is ready to be placed.



3. Move to the top left corner of the circuit window, where we want to place the battery. Click in this general area or, to be more precise, use the page borders as a guide and click in the intersection of row A and column 1. The battery appears on your circuit window:



The descriptive text that appears around the various components can, if you wish, be hidden. To hide it, double-click the component. The component “properties” screen appears with various tabs. On the Display tab, de-select the option “Use schematic option global setting”, then de-select the Descriptive text for the component. Show or hide descriptive text for all components by using **Options/Preferences** and changing the settings in the Show box of the Circuit tab.

## Step 2: Change the Battery’s Value

By default, the battery is a 12V battery, but our circuit calls for a 5V battery. You can easily change the battery’s value.

 To change the battery’s value:

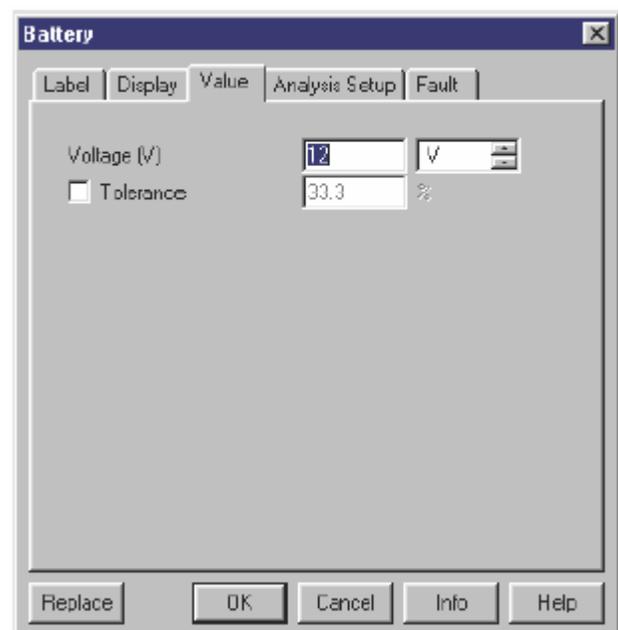
1. Double-click on the battery. The battery’s properties screen appears, with the Value tab displayed.

**Note** For more details on this screen and its tabs, see the *Multisim 2001 User Guide*.

2. Change the “12” to a “5” and click **OK**.

This ability to change the value of a part exists only for “virtual” components. Virtual components are not real; that is, you could not buy them from a supplier. Virtual parts include all sources, virtual resistor/capacitor/inductor parts, and numerous others. Virtual parts also include other “ideal” devices intended to provide theoretical equivalents of, for example, the perfect opamp.

Multisim treats virtual components slightly



differently from real parts in two ways. First, virtual components have a different default color from that of real components on your schematic. This is to remind you that, since they are not real, these components will not be exported to PCB Layout software, should you perform this step later. You will see this difference in the next step, when you place a resistor. Second, when you place such parts you do not choose from the Browser (which is shown in the next step), since you can set the value of a virtual part to anything you want.

## Placing the Next Components

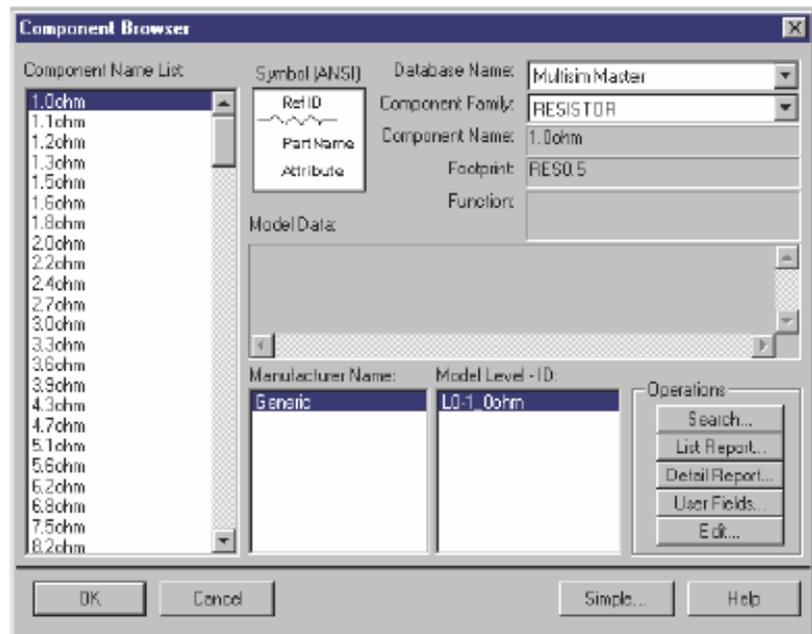
### Step 1: Place a Resistor

 To place the first resistor:

1.  Place your cursor on the **Basic** Parts Bin button and, from the toolbar that appears, click the **Resistor** button. The Component Browser screen appears:

This Browser screen appears because the Resistor family you selected contains multiple real components, that is, components you could actually purchase. It shows all the possible resistors available to you from “Multisim master” level of the database.

**Note** The Browser did not appear when you placed the battery earlier in this chapter, because the DC Voltage Source contained a virtual component which is only one item (the battery), so there was no need to have you choose from the Browser.



that  
the

2. Scroll through the **Component List** to find the 100ohm resistor we need for our circuit.

**Tip** To make your scroll through the Browser’s **Component List** faster, simply type the first few characters of the component’s name. For example, type “100” to move to that area of the list.

3. Select the 100ohm resistor and click **OK** or double click on the component value. The cursor will appear on the circuit window as a ghost image of the resistor.

4. Move your cursor to approximately A5 and click to place the component.

Notice that the resistor looks different from the battery you just placed—the resistor appears in a different color to remind you it is a real component (and will be exported to PCB Layout software, if you choose to perform that step).

### Step 2: Rotate the Resistor

To set up the resistor so it can be wired more conveniently into our circuit, we will first need to rotate it.

 To rotate the resistor:

1. Right-click on the resistor. A pop-up menu appears.
2. Choose **90 CounterCW** from the menu. The results look like this:



3. If you want, you can move the labels that accompany a component's symbol. In particular, you may want to do this after some rotations, if the labels are displayed other than as you prefer. For example, you might want to move the reference ID label. Just click and drag it to the right of the symbol, or click and use the arrow keys on the keyboard to move the label in grid increments.

### Step 3: Add Other Resistors

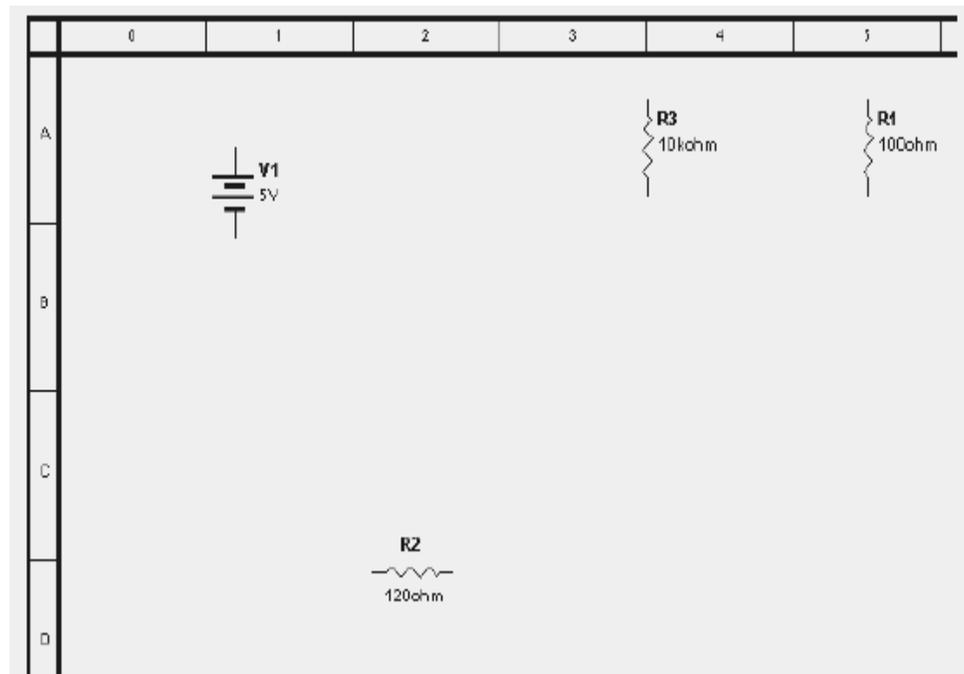
The circuit calls for two other resistors, this time a 120ohm and a 10kohm. To add the other resistors:

1. Following the instructions above, add a 120ohm resistor at roughly the intersection of row D and column 2. Notice how this second resistor is given the reference ID "R2", to indicate it is the second resistor placed.

2. Place the third resistor, a 10kohm one, at roughly the intersection of A4 and rotate it. Take a moment to look at the "In Use" list, just to the right of the Design Bar. It lists all the components you have placed so far. You can easily re-use a component from this list by clicking on it.

The results should look like this:

If necessary, you can easily move your placed components to the desired location. Simply single-click the component to select it (be sure to select the whole component, not just a label) and either drag it to the new location or use the arrow keys on your keyboard to move the component incrementally along the grid.



### Step 4: Save Your Changes

 To save your changes, choose **File/Save As**, and give a name (and location) for your circuit file.

**Note** Do not use non-English characters (for example, those with accents) when creating folder names, file names or subcircuit names.

### Placing Other Components

Following the procedures above, place the following components in the following locations as shown in the beginning of section 2.2:

- a red LED (from the Diodes family) just below R1
- a 74LS00D (from the TTL Parts Bin in the 74LS component family), roughly at D1 (Since this component consists of four separate gates, you are prompted to choose which one you will use. All are identical, so you can choose any one.)
- a 2N2222A BJT\_NPN (from the Transistor Parts Bin in the BJT-NPN component family), to the right of R2

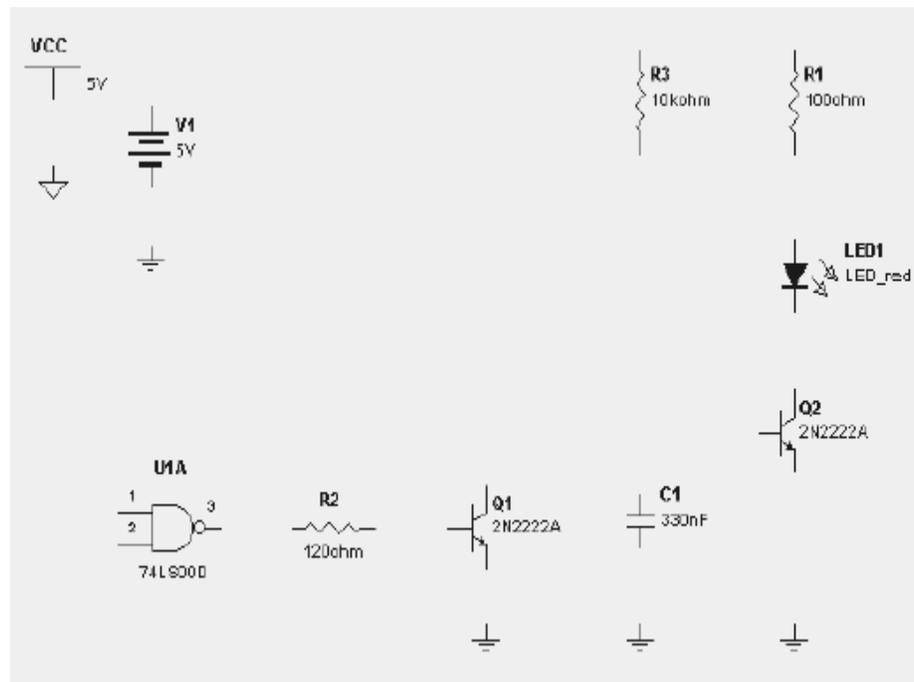
- another 2N2222A BJT\_NPN below the LED (simply copy the previous one and paste the copy at the new location)
- a 330nF capacitor (from the Basic Parts Bin in the Capacitor component family), to the right of your first BJT\_NPN, and rotate it clockwise (move the labels after rotating, if desired)
- a ground (from the Sources Parts Bin in the Ground component family), below V1, Q1, Q2 and C1 — you may use any number of grounds in your circuit, as in this tutorial, or just one with multiple parts connected to it)
- a 5 Volt VCC (from the Sources Parts Bin in the VCC component family), in the top left corner of the circuit window, and a digital ground (from the Sources group) just below it.

**Note** A circuit containing a digital component must have VCC and a digital ground placed on the circuit window. Multisim will use these two components to power the digital component. The requirements to change the voltage of VCC is the user's responsibility.

The results should look something like this:

**Tip** For a quick way to move your components into line, select them and use the arrow keys on your keyboard to move the components in grid increments. Lining them up will make wiring easier.

**Note** Don't forget to save your changes using **File/Save**.



## Changing the Label and Color of Individual Components and Nodes

Multisim assigns labels and colors to a placed component. You can modify or move component labels, and change component colors.

 To change the label of any individual component you have placed:

1. Double-click on the component. The component's properties screen appears.
2. Click the Label tab and enter or modify the label (which must be composed of letters and numbers only—no special characters or spaces).
3. To cancel your changes, click **Cancel**. To save your changes, click **OK**.

 To change the color of any individual component, right-click on it and choose **Color** from the pop-up menu. Choose the desired color from the screen that appears.

**Note** Changing the color of an individual component is different from changing the color scheme of the current circuit, or of the defaults set using **Options/Preferences**, as explained earlier in this manual.

## Wiring Components

Now that you have the components placed on the circuit window, you need to wire them together. All components have pins that you use to wire them to other components. In Multisim, you can choose to wire components either automatically or manually. Automatic wiring, a feature unique to Multisim, means Multisim automatically wires the connection for you, selecting the best path between your chosen pins. Automatic wiring avoids wiring through other components or overlapping wires. Manual wiring means you control the path of the wire on the circuit window. You can even combine the two methods in a single wire, for example, starting wiring manually and letting Multisim automatically complete the wire for you.

We need to wire together the components we placed in the circuit. Automatic wiring will take care of most of our wiring needs for this circuit. To follow along with these steps, you can either continue with the circuit you've been building in this chapter, or open the file `tut1.msm`, located in the `Tutorial` folder, to work with a circuit that has all the components appropriately placed.

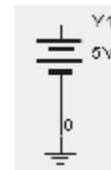
## Using Automatic Wiring

We'll begin with wiring V1 and its ground.

 To start the automatic wiring process:

1. Click the pin coming out of the bottom of V1. The cursor changes to a cross symbol indicating that you are in wiring mode.
2. Click the pin on the top of the ground. The wire is dragged from the first pin to wherever you move your cursor. The two components automatically become wired together and look like this:

**Note** By default, the wires are displayed in red. To change the default, right-click on the circuit window and choose **Color** from the pop-up menu. To change the color of just a single wire, right-click on the wire and choose **Color** from the pop-up menu.



3. Using automatic wiring, make the following connections:

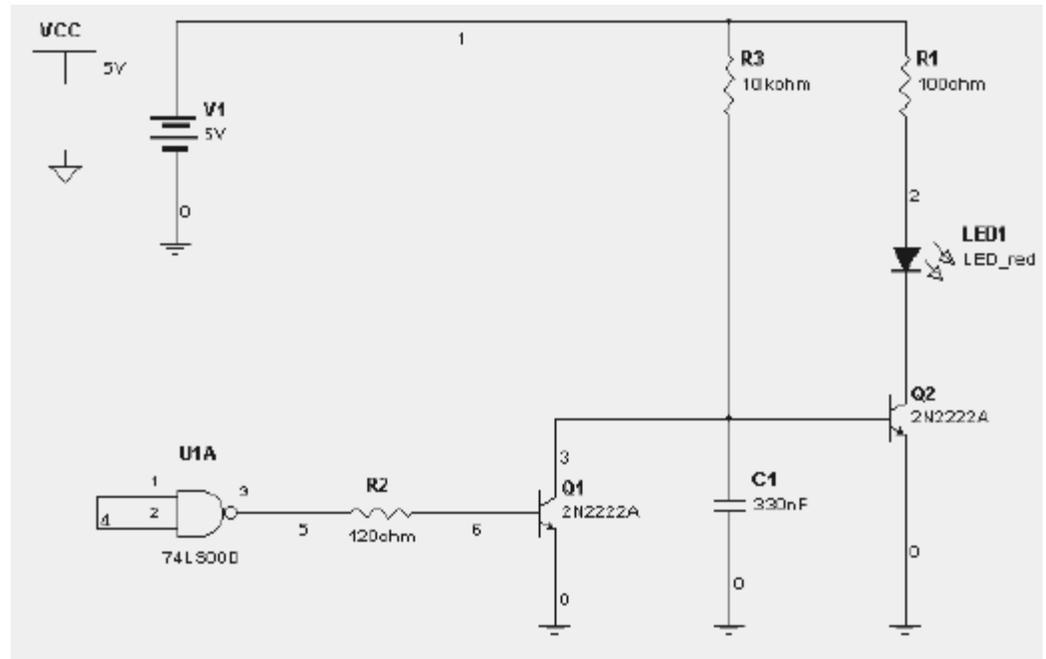
- wire V1 to R1
- wire R1 to the LED
- wire the LED to the collector of Q2
- wire the emitters of Q2 and Q1 to their grounds
- wire C1 to its ground
- wire base of Q1 to R2
- wire R2 to pin3 (output) of U1
- wire pin1 of U1 to pin 2 of U1
- wire R3 to the wire running from V1 to R1 — click on the pin of R3 and then click the wire to automatically place a junction where the wires meet
- wire base of Q2 to collector of Q1
- wire R3 to the wire running from Q2 to Q1

- wire C1 to the connection on the wire running from Q2 to Q1.

The results should look like this:

After a wire is connected between two pins the cursor returns to its normal mode and is ready for your next command.

 To delete a wire, right-click on it and choose **Delete** from the pop-up menu, or press DELETE.

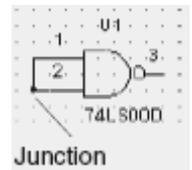


## Using Manual Wiring

We now want to connect the inputs of U1 to the wire between the LED and Q2. To have precise control over the path of the wire, we will use manual wiring. Multisim prevents you from placing two wires to a single pin. This is to prevent possible wiring errors. We therefore need to start, not from pin 1 or 2 of U1, but from the wire between them. To wire from the middle of one wire to another wire, you have to add a junction on the first wire.

 To add a junction:

1. Choose **Place/Place Junction**. The junction is shown as a ghost image at the point of the cursor.
2. Move the junction to the desired location on the wire between the inputs of U1 and click.
3. The junction appears on the wire:



We now need to use manual wiring to ensure that the wire follows the path we want. To help position the wire appropriately, display the grid (if it is not visible).

 To display the grid, right-click on the circuit window and choose **Show Grid** from the pop-up menu.

We are now ready to wire the connection manually.

 To perform a manual connection:

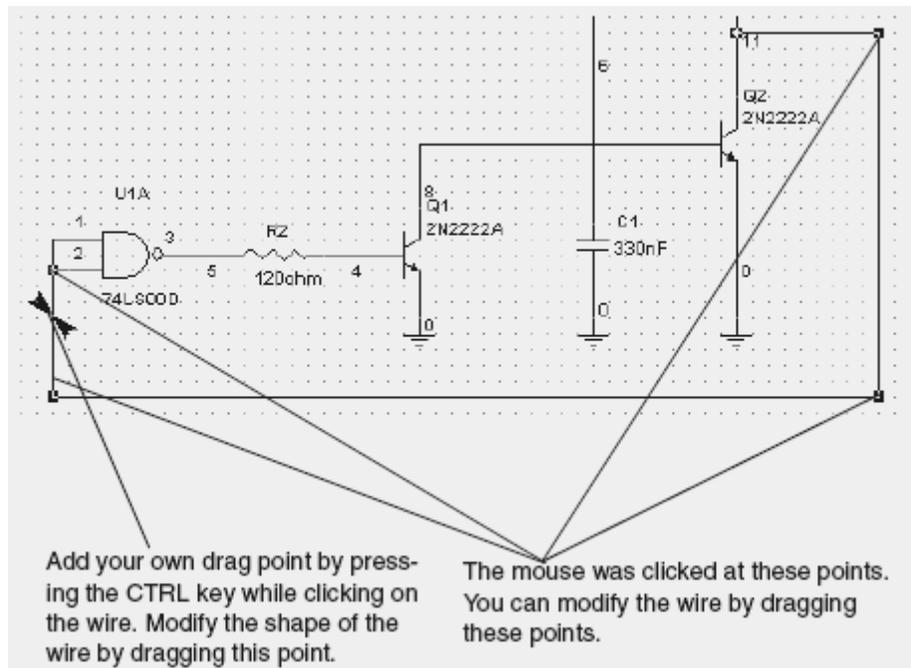
1. Click the junction you just placed at the inputs of U1. Your cursor will change to a cross symbol indicating you are in wiring mode.
2. Drag the wire down to below the other components and click. This “fixes” the wire in its place.

3. Drag the wire across the width of the circuit to a few grid spaces past the last component. Click again.
4. Drag the wire up to a point across from the wire between LED1 and Q2. Click again.
5. Drag the wire to a place on the wire between LED1 and Q2 and click again.

The results look like this:

Click on the wire you just made. The boxes displayed on the wire (“drag points”) show where you clicked as you placed the wire. You can modify the shape of the placed wire by clicking on any of these points and dragging the wire segment. If you want to, experiment with this, save your changes first.

While the wire is selected, you can add additional drag points by using the CTRL key and clicking the wire where you want to add more points. You can also move the wire by moving your cursor over the wire and then click and drag when your cursor changes to a double arrow symbol.



**Note** To remove a drag point, press the CTRL key and click on the point. When you remove a drag point, the wire will stretch to a straight line between the two points on either side of the point you removed.

## Adding Text to the Circuit

Multisim allows you to add a title block and text to annotate your circuit.

 To add a title block, choose **View/Show Title Block and Border**. The title block appears at the bottom right of your circuit window.

 To edit the title block, choose **Options/Modify Title Block**. Enter the desired text in the fields and click **OK**.

 To add text:

1. Choose **Place/Place Text**.
2. Click on the circuit window where you want the text placed. A text box appears.
3. Type the text—for example, type “My tutorial circuit”.
4. Click anywhere on the circuit window the text appears in your circuit.

 To delete text, right-click on the text box and choose **Delete** from the pop-up menu or press DELETE.

 To change the color of text, right-click on the text box, choose **Color** from the pop-up menu, and choose the desired color.

 To edit text, double-click on the text box and make your changes. Click any location out of the text box to stop editing text.

 To move text, click on the text box and drag it to a new location.

## Conclusion

You have now experimented with the different ways of locating, placing and wiring components on the circuit window. You have also seen some of the options available to you for controlling the appearance of your circuit. Before we go on to add an instrument, we are going to explore editing components in the next chapter.

## Editing a Component

### About this Chapter

This chapter gives you a brief introduction to the kinds of modifications you can make by using component editing. It shows you how to select the type of modification you wish to make and the component you wish to modify. However, because there are several different types of modification you can make, an explanation of specific tasks is beyond the scope of this manual. For details on editing components, see the Multisim User Guide.

### Introduction to Component Editing

Component editing allows you to modify any component stored in Multisim's component database. For example, an existing component might now be available in a new package (originally pin-through hole, now surface mount). You can easily copy the component information and change only the package details to create this new component.

You can also use Component editing to create your own component (and place it into the database), or delete a component from the database.

In the component database each component is identified by six types of information, each accessed through a separate tab:

- General information (such as component name, manufacturer, icon, date the component was created and the author of the component).
- Symbol (pictorial representation of the component for schematic capture).
- Model (information used to represent the actual operation/behavior of the component during simulation) — necessary only for a component that will be simulated.
- Footprint (the package information that Multisim uses when exporting a schematic diagram containing this component to a PCB Layout package such as Ultiboard, also from Electronics Workbench).
- Electrical Parameters of the components.
- User Fields (if used to further define the components. These fields can be modified by using Database Management).

### Editing a Component

1. Click the **Component Editing** button in the Design Bar.
2. Choose Edit Component. The Select Component - Edit screen appears.

3. From the **Database Name** list, choose the database containing the component you want to edit. Typically, this will be the “Multisim Master” database.

4. From the **Family Name** list, choose the component family containing the component you want to edit. The Component Name list changes to show a list of components in that family.

5. From the **Component Name** list, choose the component you want to edit.

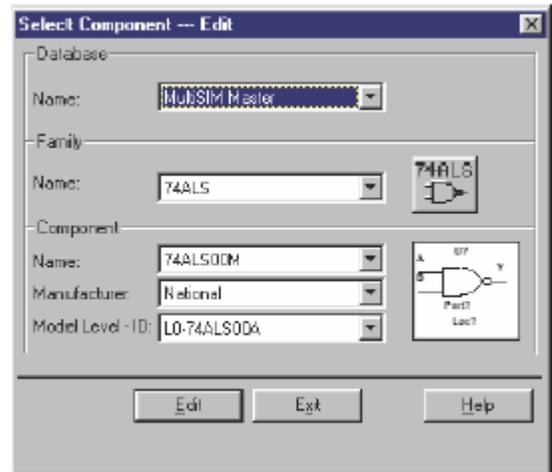
6. If necessary, choose the Manufacturer and Model of the component you want to edit (if more than one manufacturer or model exists).

The component’s symbol and icon appear on the screen.

7. To continue, click **Edit**. (To cancel, click **Exit**.)

The Component Properties screen, consisting of the six tabs mentioned earlier, appears:

These tabs correspond to the type of information you can edit for a component. To actually modify a symbol, a model, or a footprint, is somewhat involved and is therefore beyond the scope of this manual. For more about using these tabs, see the Multisim User Guide.



## Conclusion

Now that you have been introduced to Component editing, it’s time to add an instrument to your circuit, as described in the next chapter.

## Adding Instruments to Your Circuit

### About this Chapter

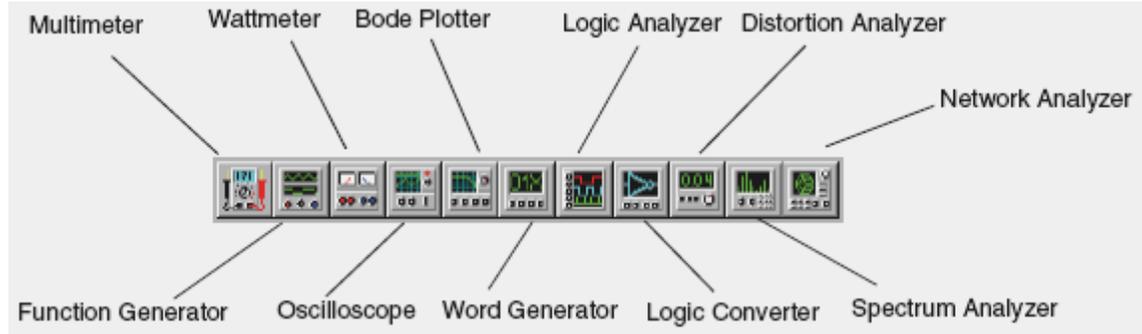
This chapter gives you a brief overview of Multisim’s instrument functions and takes you through the steps of adding an instrument to your circuit. Multisim offers a variety of virtual instruments. This chapter describes the oscilloscope—for other instruments, please see the *Multisim 2001 User Guide*.

### Introduction

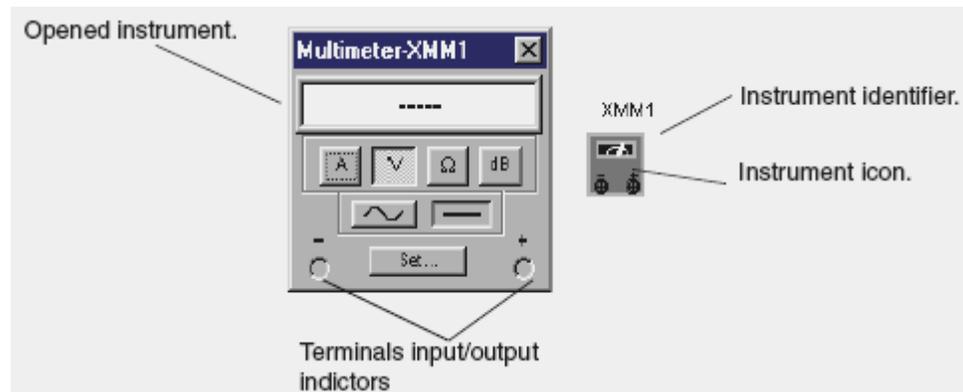
Multisim provides a number of virtual instruments. You use these instruments to measure the behavior of your circuits. These instruments are set, used and read just like their real-world equivalents. They look and feel just like the instruments you’ve seen and used in a lab. Virtual instruments are one of the best and easiest ways of examining your circuit’s behavior by displaying the results of simulation.



The instrument toolbar is displayed by default. If your instrument toolbar is not displayed, click the **Instruments** button on the Design Bar. When you click this button, the Instruments toolbar appears. It includes one button for each instrument.



Virtual instruments have two views: the instrument icon you attach to your circuit, and the opened instrument, where you set the instrument's controls and display options.



## Adding and Connecting Instruments

For the purposes of this tutorial, we want to add an oscilloscope to the circuit. Either carry on with the circuit you have been creating so far, or open the file `tut2.msm` (which has been properly wired) from the `Tutorial` folder and proceed.

### Step 1: Add an oscilloscope

 To add the oscilloscope:

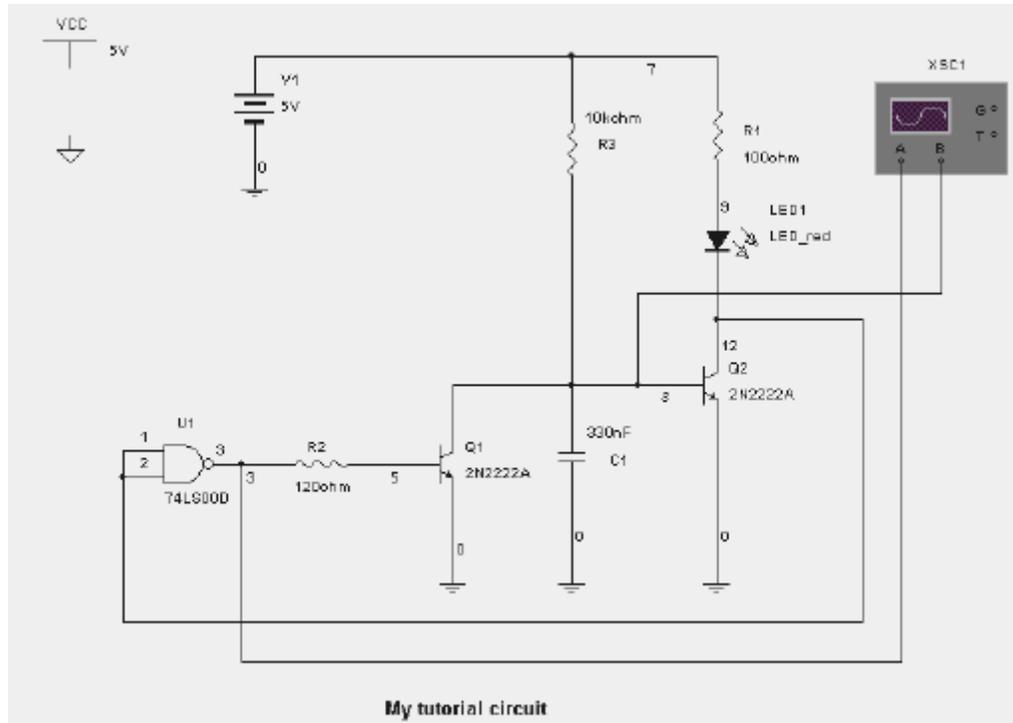
1. Click the **Instruments** button in the Design Bar. The Instruments toolbar appears.
2. Click the Oscilloscope button and move your cursor to the circuit window. Your cursor will appear as the ghost image of the instrument.
3. Move the cursor to the right side of your circuit and click.
4. The oscilloscope icon appears on the circuit window.
5. You now need to wire the oscilloscope into your circuit.

## Step 2: Wire the oscilloscope to the circuit

 To wire the oscilloscope into your circuit:

1. Click the A terminal on the oscilloscope icon and drag a wire to the junction between the output of U1 and R2.
2. Click the B terminal on the oscilloscope icon and drag a wire to between Q2 and C1.

Your circuit should look like this:

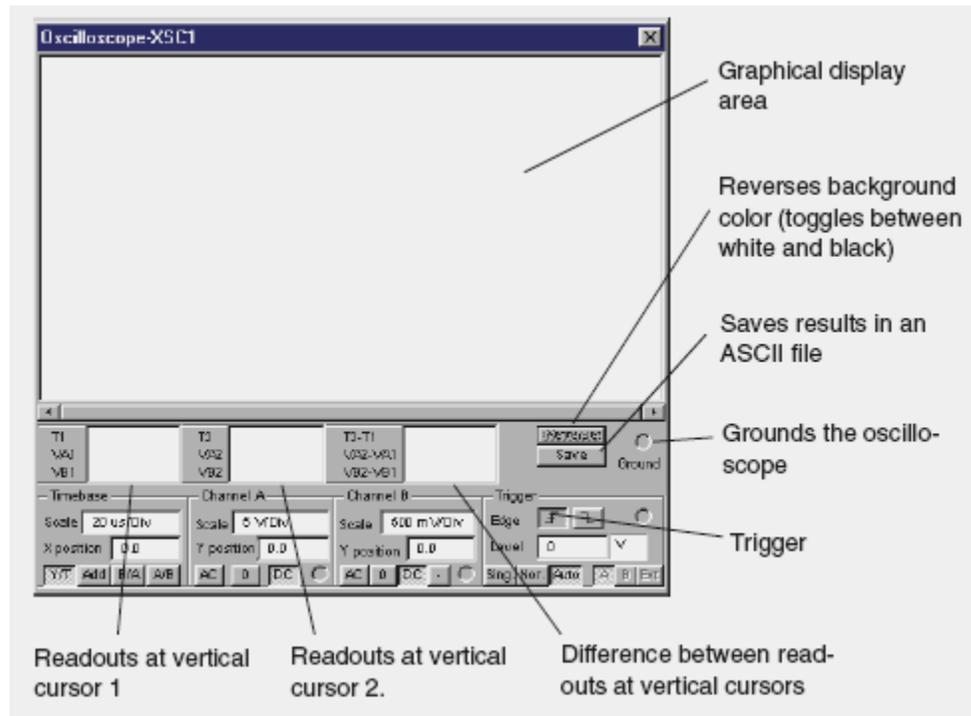


You'll see the results of your work in the next chapter when you simulate the circuit.

## Configuring Instrument Settings

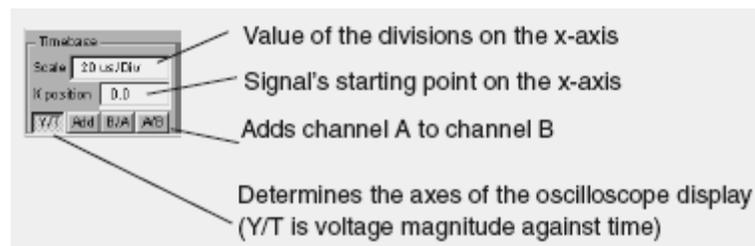
Each virtual instrument provided by Multisim includes its own series of optional settings that control its display.

 To open the oscilloscope, double-click the oscilloscope icon. The oscilloscope looks like this:



The time base section of the scope's panel controls the scale of the oscilloscope's horizontal or x-axis when comparing magnitude against time (Y/T)

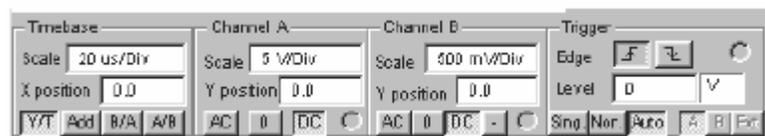
To get a more readable display, adjust the timebase in inverse proportion to the frequency—the higher the frequency, the lower (or more magnified) the time base.



 To set the time base for our circuit:

- Set the timebase scale (which should use the waveform magnitude against time (Y/T) axes) to 20  $\mu$ s/Div to best display the frequencies in our circuit.
- Set the Channel A scale to 5V/Div, and click **DC**.
- Set the Channel B scale to 500 mV/Div, and click **DC**.

The results should look like this:



## Conclusion

Now that an instrument is in place and configured correctly for our circuit, you can simulate and see the results displayed on the oscilloscope, as explained in the following chapter.

# Simulating Your Circuit

## About this Chapter

This chapter explains how to carry out a simulation on your circuit and see the results displayed on the oscilloscope.

Although Multisim 2001 offers multiple types of simulation, including SPICE, VHDL, Verilog HDL, and combinations of these, this chapter covers only SPICE. Co-simulation with VHDL is introduced in Chapter 7 of this manual, while using VHDL or Verilog HDL to write code for programmable logic design or modelling complex digital chips is addressed in the *Multisim 2001 User Guide*.

## Simulating your Circuit

You are now ready to simulate your circuit.

You can either continue with the circuit you've been building in this chapter, or open the files `tut3.msm` from the `Tutorial` folder to begin with all the components correctly placed and wired together, and with the oscilloscope wired and set up correctly.



To simulate the circuit, click the **Simulate** button in the Design Bar and, from the pop-up menu, choose **Run**.

## Viewing Simulation Results

The simulation begins, but we need some way of displaying the results. One of the best ways is to use the oscilloscope we added earlier.

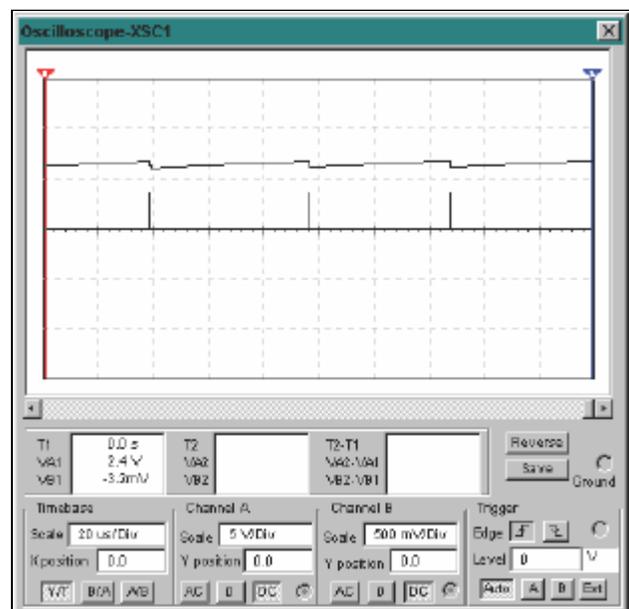
To see the results in the oscilloscope, if necessary, double-click the oscilloscope icon to “open” the instrument display, if it isn't already open.

If you changed the instrument settings as described in the previous chapter, you should soon see results looking something like this:

Also note that on the circuit window, in a capability unique to Multisim, the LED flashes on and off, displaying its behavior as it responds to circuit source stimulation during simulation.

We will now halt simulation of our circuit.

To stop the simulation, click the **Simulate** Design Bar button and, from the pop-up menu, choose **Run** again.



# Conclusion

You have learned how to begin simulation and display the simulation results on an instrument. In the next chapter you will see how to perform an analysis on your circuit and see the results of that analysis.

## Performing Analyses on Your Circuit

### About this Chapter

This chapter outlines basic information on using Multisim 2001 analyses and explains how to perform one such analysis on your circuit.

### The Analyses

Multisim provides you with many different types of analyses. Each one includes step-by-step instructions, provided on-line, to guide you through its use.

When you perform an analysis, the results are displayed on a plot in the Multisim Grapher (unless you specify otherwise) and saved for use in the Postprocessor.



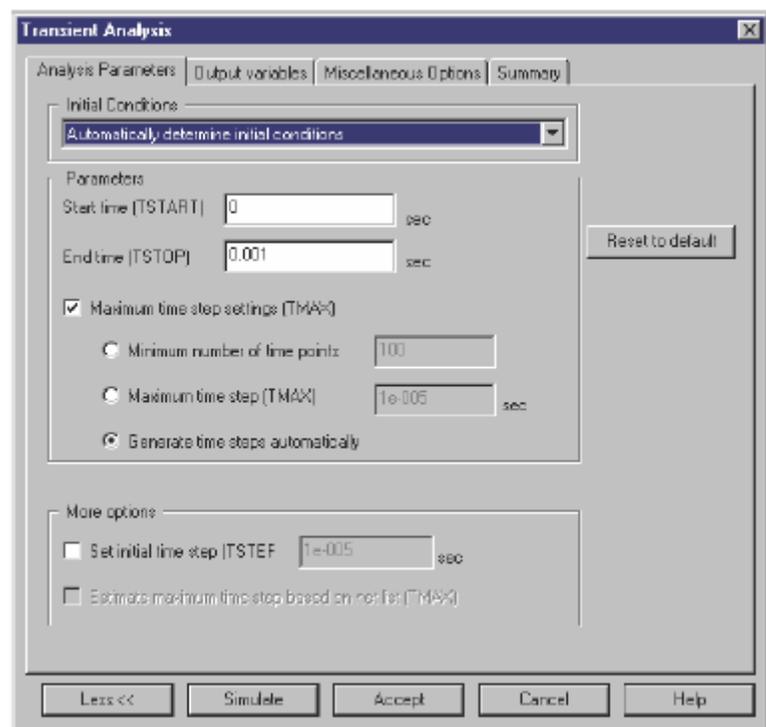
When you choose most analyses from the menu that appears after you click the **Analysis** button in the Design Bar, you see a screen with several tabs, including:

- The Analysis Parameters tab, where you set the parameters for this particular analysis
- The Output Variables tab, where you specify which nodes are to be analyzed and what is to be done with the results
- The Miscellaneous Options tab, where, if you wish, you can choose a title for the plot, as well as other options
- The Summary tab, where, if you wish, you can see a consolidated view of all the settings for the analysis.

For complete information on using the Analyses screens, refer to the *Multisim User Guide*.

### About Transient Analysis

In the *Multisim User Guide*, each analysis is described in detail, including the theory behind what Multisim does to perform the analysis. It is not necessary that you understand this information, which is provided primarily for the sake of completeness. In transient analysis, also called time-domain transient analysis, Multisim computes the circuit's response as a function of time. Each input cycle is divided into intervals, and a DC analysis is performed for each time



point in the cycle. The solution for the voltage waveform at a node is determined by the value of that voltage at each time point over one complete cycle.

## Running the Analysis



To initiate the analysis, click the **Analysis** button from the Design Bar and choose **Transient Analysis** from the pop-up menu. The simplified Transient Analysis screen appears, offering four tabs. To view the full screen with all the options and information displayed, as in the example shown below, click **More**.

The Miscellaneous tab offers options that provide you with additional flexibility, but that are not required. Use this tab to set a title for the analysis results, to check if the circuit is valid for analysis and to set custom analysis options.

The Summary tab offers a quick overview of all the various settings for your analysis. It is not required, but you can use it to view summary information about your analysis once it is complete.

To perform the analysis, you have to set values on the other two tabs.

### Step 1: Choose Output Variables

We are going to want to perform the Transient Analysis on nodes 3 (between Q1 and Q2) and 6 (from pin 3 of the gate). Choose those nodes from the available nodes on the Output Variables tab.

**Note** If you are still using your circuit, your node numbers may not match the ones above (node 3 and 6). This is likely a result of your circuit being wired correctly, but in a different sequence from the steps we suggested for the tutorial circuit. You can choose to continue to use your circuit and select the appropriate node numbers for the equivalent place in your circuit, or load `tut3.msm` from the Tutorial folder.

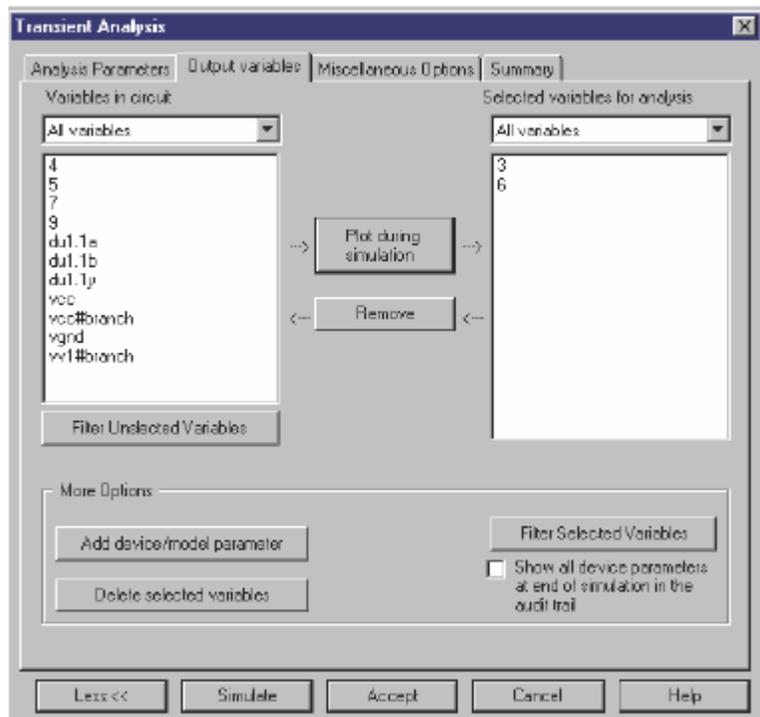


To select the nodes:

1. Select 3 from the **Filter variables displayed** and click **Plot during simulation**.

2. Select 6 from the **Filter variables displayed** and click **Plot during simulation**.

The results look like this (if your screen doesn't look like this, click **More**):

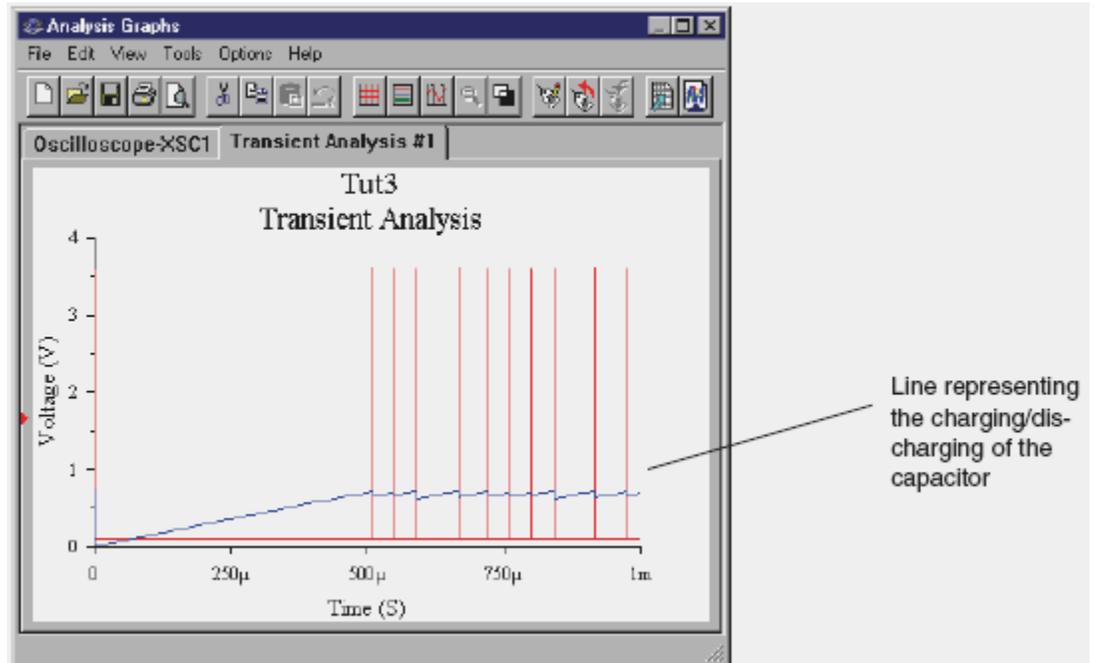


## Step 2: Set the Analysis Parameters

Analysis parameters are set on the first tab as shown earlier in the chapter, and for our circuit can remain at their default values.

## Step 3: View the Results

 To see the results, click **Simulate**. You see the following display:



The results show how the capacitor (the blue line) charges as a result of the spikes in the red lines.

 To see a key to the line color, click the legends button.

**Note** The Grapher window shows two tabs—one for the analysis you just ran, and one for the results of the simulation you ran in the previous chapter and viewed using the oscilloscope.

The Multisim Grapher offers a variety of tools for examining the results of your simulation and analysis. Take some time to experiment with the different buttons and commands. For example, in the Transient Analysis tab, by dragging the cursor, you create a “zoom” box. See the *Multisim User Guide* for full details.

## Conclusion

This chapter introduced you to the primary steps in performing an analysis in Multisim. The next Design Bar button is the Postprocessor, which is discussed in detail in the *Multisim User Guide*, but not in this manual. Instead, we will now explore using an HDL to model a component and simulate its behavior.

# GENERAL LABORATORY RULES AND SAFETY RULES

- Be **PUNCTUAL** for your laboratory session.
- Foods, Drinks and smoking are not allowed.
- Open- toed shoes are not allowed.
- The lab timetable must be strictly followed. Prior permission from the lab supervisor must be obtained if any change is to be made.
- Respect the laboratory and its other user. Noise must be kept to a minimum.
- Workspace must be kept clean and tidy at all time. Points might be taken off on student / group who fail to follow this.
- Handle all apparatus with care.
- All students are liable for any damage to equipment due to their own negligence.
- Students are strictly **PROHIBITED** from taken out any items from the laboratory without permission from the lab Supervisor.
- Do not work alone.
- Report any injuries to your instructor immediately.
- Know where the main lab power switch is located and how to turn off the power.
- In the event of a fire:
  - ✓ De-energize the circuit
  - ✓ Notify the instructor immediately (he or she may use an appropriate fire extinguisher)
  - ✓ Remove all fuels (such as paper), if possible
  - ✓ Administer first aid
  - ✓ Call Campus Public Safety Dept. if necessary. (Pull the fire alarm, evacuate the building and call from outside the building, if appropriate).
- Make connections to a de-energized circuit only. When in doubt, do not power-up without the instructor's inspection.
- Do not change the circuit with the power on. This can damage the components and create a safety hazard.
- Do not rely on fuses, relays, circuit breakers, interlock devices or other protective devices to protect you or your circuit.
- Turn off power to all equipment when finished.
- Many electrical components are hot when functioning – be careful.
- Do not bypass the protective third prong on an electrical plug. Use a properly grounded adapter when plugging into a two-prong outlet (check the outlet for proper grounding).
- Report defective equipment or components to your instructor immediately. Please do not return defective components to the Technical Support Center. (Your instructor will tag defective equipment with the nature of the problem along with the date).
- Pay particular attention to the polarity of polarized capacitors.
- If a component burns, avoid breathing the fumes. They could be toxic.
- Use only one hand at a time in any high-voltage circuit – prevent putting your body in the circuit.

- Avoid wearing loose or floppy clothing around moving machinery. Wear safety glasses.
- Do not attempt to pull someone from a “live” circuit. Turn the power off if possible. Otherwise, remove the person with a nonconducting object.

## **GRADING POLICY**

**The total mark for this lab is distributed as follows:**

Lab Report + Prelab + Quizzes + attendance	40%
Mid-term Exam	20%
Final	40%

## Laboratory Report Guidelines

Effective technical communication and documentation are essential skills for an engineer. Technical documentation can take on different forms depending on the needs of the audience. However, some common principles can be applied to all forms of technical documentation. Consideration of these principles is essential to ensure effective communication of the information that the audience needs. A few guidelines will be considered here to write effective lab reports.

### Formal Reports

- Title Page
- Objectives
- Procedure
- Methodology/Theory
- Calculations
- Analysis/Results
- Conclusion

A brief description of each section of the reports is provided below. Use the descriptions as an outline of what should be included in the report.

Title Page: The title page should include, in the following order, the course number, the title of the experiment, your name, your lab partners name, the TA or professors leading the lab section, and the date the report was submitted.

Objective: The objective should discuss the purpose of the lab experiment.

Procedure: The procedure should include a brief summary of the processes used in performing the lab experiment. Related history of the subject under investigation in the lab may also be included when appropriate. The procedure should be relatively short.

Methodology/Theory: The section on methodology or theory should discuss the experiment performed in lab. Theoretical development associated with design procedures should be discussed, particularly in the cases where it influences the choices you make in completing the lab experiment. The methodology presented should be sufficiently complete for someone reading the report to reproduce the experiment and verify the results.

Calculations: Report should include complete calculations; all circuits in the experiment should be solved, and prepared in proper manner to be compared with the related measured values.

Analysis/Results: The final data collected or measured should be placed in the results section. This would apply specifically to data or measurements associated with the purpose and design specifications of the experiment. Diagrams, numerical data organized in tables, etc. should be placed in this section, and any pertinent analysis (but not concluding remarks) should be included. Any intermediate results that were used to obtain the final results of the experiment do not need to be included in the report. Also measured values are compared with theoretical ones in this part. All results should be tabulated.

Conclusion: The conclusion should respond to the goals and objectives discussed in the introduction of the report. A general discussion of how well the experiment did (or did not) fulfill its objectives should be presented without getting personal. Applications using the results from the experiment, or a forecast of future events that relate to the work done in the experiment may also be included.

## **Using Proper Spelling, Grammar, and Technique**

Lab reports should be proofread for proper spelling and grammar. Use the spell check features of the word processor, and proofread. You may also want to have someone else proofread your work.

Proper technique is essential for documenting what was accomplished in the lab and for establishing the tone of the report. Several examples of poor technique, which should be avoided, are listed below.

- The lab report is not a personal experience. Do not discuss how you felt about the experiment. Your grade will not be based on how interesting you found the experiment, how satisfied you were personally with the results, or how frustrated you became when things did not work properly or in a timely manner.
- The lab report should not document the procedure or specific steps followed in completing the experiment. For example, it is not necessary to explain how a circuit was wired and how components were placed on a breadboard, or how the equipment was plugged in and measurements were made, etc. The only exception would be when a specific procedure required to obtain the desired result is not intuitively obvious.
- Avoid using anthropomorphisms (applying human feelings or actions to inanimate objects).
- Do not talk about yourself in the third person. Document what was done without using “I,” “we,” “the student,” etc.

Use good judgement when adding detail to your reports. Do not include unnecessary or obvious information. If there is more than one way to do something (e.g. collecting

data), then add detail that describes your choice (e.g. what piece of equipment, or what measurement function, etc. was used in collecting the data).

Additional guidelines that should be followed are listed below.

- All reports must be typed, using easy to read fonts. Clip-art images, color backgrounds and borders are inappropriate for a technical report
- Figures, schematics, plots, and equations should be labeled and numbered so that they can be referred to in the report. Inline equations are not acceptable. Equations should be separated from the text, and centered in the page. Plot axes should have appropriate labels that clearly identify what is being represented. When possible, equations, schematics, figures, etc. should be placed in the body of the report near the text that first refers to them. Only include items in the appendix if they are supplementary items or if they cannot be easily included in the body of the report.
- Data that is not presented in a graph format should be organized in tables and must be identified by appropriate title, row/column heading, and units.
- Each student must submit an individual report based on individual effort. Even though partners work together in the laboratory, each student must submit their own report.

## References

1. Blicq, R., Moretto, L., *Technically-Write*, 5<sup>th</sup> ed, Upper Saddle River, New Jersey: Prentice Hall, 1999
2. Sides, C., *How to Write & Present Technical Information*, 3<sup>rd</sup> ed, Phoenix, Arizona: The Oryx Press, 1999.

# Experiment 1

## Lab. Equipment Familiarization

### 1.1 Objective

- To introduce the Multimeter, the breadboard, the power supply, resistors and their color code.
- To learn to properly use the lab instruments and the correct method of measuring electrical quantities with each instrument.

### 1.2 Basic Information

#### The Digital Multimeter [DMM]

This device is used to measure values of electrical quantities; such as voltage, current, resistance, etc. The DMM is easy to use, and necessary for all electronics labs.

#### Voltage Measurement.

Turn on the Multimeter. Using the rotary selector switch, select the voltage function [V<sub>DC</sub>]. Select the *AUTO* range mode by making a long press on the *range* button. Insert the positive (+) lead (normally red) in the voltage socket and the negative (-) lead (normally black) in the common socket. Place the red probe on the higher voltage point and the black probe on the lower voltage point. The DMM will display the voltage drop between the probe tips.

**Voltage measurement** between any two points is made in **parallel** with the components between those two points. If the probes are reversed the reading will be negative of the original value. A Voltmeter has very large internal resistance, which is considered as open circuit (O.C.) during calculations.

#### Current measurement.

Turn on the DMM and select the current function. Place the positive (+) probe in the current socket and the negative (-) one in the common socket. Select *AUTO* range mode. Connect the tips of the probes in series with the component through which the current is being measured. A positive reading will indicate current direction from the positive (+) to the negative (-) probes.

**Current measurement** through a component is made in **series** with that component. An Ammeter has very small internal resistance, which is treated as short circuit (S.C.) during calculations. Figure 1.1 shows the connection of the DMM as a Voltmeter and an Ammeter.

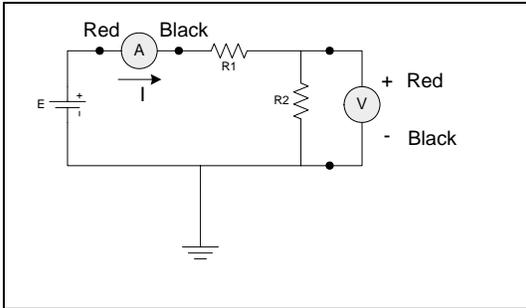


Fig.1.1: DMM Connection

**Important Note:** Always disconnect the probes of the DMM from the circuit before changing the selector switch from current to voltage or vice versa. Failing to do so will damage the DMM. Switching off the meter without disconnecting the probes is insufficient for protecting the DMM. Connecting the Multimeter in an incorrect way, or choosing the wrong selection of switches, may result in personal injury, damage to the Multimeter and/or the lab equipment. Observe and obey safety rules and instructions at all times. **If in doubt ask your instructor.**

### Resistance measurement

The ohmmeter part of a Multimeter is basically a Voltmeter and Ammeter. A built-in voltage source is connected across the resistor to be measured and an Ammeter measures the current flow. The resistance value is the ratio of voltage to current flow. Resistance should **never** be measured while it is connected in a circuit. To measure the resistance of a component: switch off the power from the circuit. Disconnect the component from the circuit. Switch the Multimeter to measure resistance, and select the *AUTO* range mode. Touch the probe tips to the end of the ends of the component, and read the value displayed.

### **The Breadboard**

The breadboard is a tool for effecting connections between electronic components without the need for soldering. It consists of groups of rows and columns of socket (called busses) connected together in a systematic way. It enables the attachment of components such as resistors, capacitors, transistors and wiring in a versatile manner. The breadboard consists of two main parts. The thinner one, which has two long rows of sockets that are connected together. The larger unit consists of columns of 5 sockets that are connected together. The figure below shows the two parts, and how the elements are connected. The leads of any component or wire are inserted into the sockets to effect the interconnection desired. The long busses of the thin part are used for power distribution to the various parts of the circuit under test.

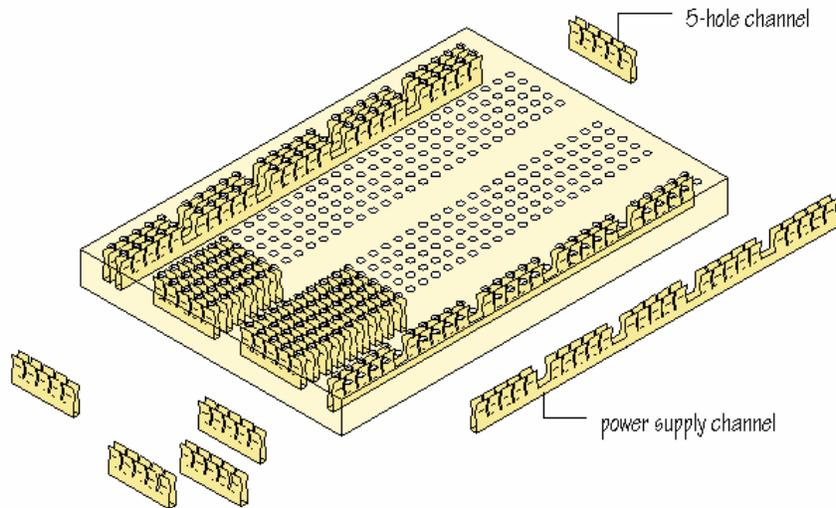


Fig.1.2: The Breadboard

## The Power Supply

The DC power supply is used to generate a constant voltage (CV) or a constant current (CI). That is, it may be used as a DC voltage source or as a DC current to drive the circuit under test. The unit in the lab has two supplies: output 1 and output 2. Both outputs are variable voltage/current supplies. The variable supplies can be used independently to achieve positive or negative output.

### ▪ Obtaining a certain voltage from the variable power supply

1. Switch the power on and make sure the output switch is made off.
2. Reduce the current and voltage knobs to minimum.
3. Switch on the output switch. [OUTPUT ON lamp turns on]
4. Increase the current knob until the CV (constant voltage) lamp turns on.
5. Increase the voltage knob up to the desired value using the coarse and fine voltage control.
6. If the CV lamp turns off after connecting the circuit under test, make sure there are no shorts in the circuit. Then increase the current knob until CV turns on again.

### ▪ Obtaining a certain current from the variable power supply

1. Switch the power on and make sure the output switch is made off.
2. Reduce the current and voltage knobs to minimum.
3. Switch on the output switch. [OUTPUT ON lamp turns on]
4. Short the output leads of the supply. (Notice CV lamp turns off and CC lamp turns on).
5. Increase the current knob up to the desired value.

6. If the CC lamp turns off after connecting the circuit under test, increase the voltage knob until CC turns on again. If it does not turn on this means the circuit is open.

**Resistors.**

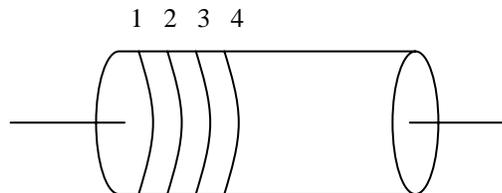
The flow of charge through any material encounters an opposing force due to collisions between the electrons and atoms in the material. This converts electrical energy into heat, and the force encountered is called resistance. Resistors are manufactured to a specific amount of resistance. They are used in circuits for many purposes, such as providing voltage drops and limiting current flow. The relative size of a resistor varies with its wattage (power) rating, since a larger size sustains higher current and heat dissipation. To identify the value of a resistor, the color code has been devised. Four or five color bands are printed on the end of the resistor (see figure1.3).The bands are always read from the end that has the band closest to it. The first and second bands represent the first and second digits. The third band indicates the power of 10 multiplied by the first two digits (i.e. the number of zeros that follow the second digit). The fourth band is the tolerance, with which the resistor was manufactured.

To reduce the cost of manufacturing resistors, only certain *Preferred Values* are available commercially. Resistors are manufactured as multiples of 10 of these preferred values. They are:

- 1.0 2.0 3.3 5.6 8.2
- 1.2 2.2 3.9 6.2 9.1
- 1.5 2.7 4.7 6.8
- 1.8 3.0 5.1 7.5

All in ohm

Resistors are manufactured in fixed wattage (power rating), mainly: 1/8, 1/4, 1/2, 1, 2 Watts, etc. This is identified by size, or printed on the component. Note that the size of a resistor is related to its current carrying capacity. Since large currents mean higher temperatures, a larger surface is needed for the heat to be dissipated, hence a large resistor size resistor indicates a high power rating.



Color	Bands 1 & 2	Band 3	Band 4
Black	0	1	-
Brown	1	10	1%
Red	2	100	2%
Orange	3	1k	-

Yellow	4	10k	-
Green	5	100k	0.5%
Blue	6	1M	0.25%
Violet	7	-	0.1%
Gray	8	-	-
White	9	-	-
Gold	-	0.1	5%
Silver	-	0.01	10%
No color	-	-	20%

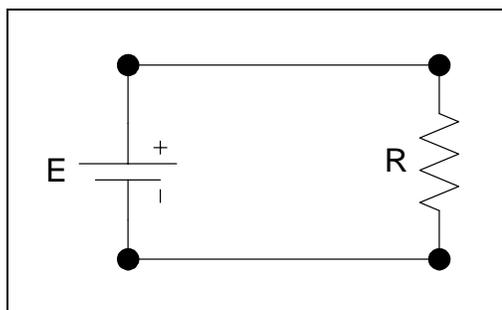
Fig.1.3: Color Code for Resistance

## 1.3 Equipment

Digital Multimeter (DMM), Power Supply (PS)

## 1.4 Procedure

1. Use the DMM to measure the resistance of the five resistors provided.
2. Read the color code of these resistors. Tabulate your results.
3. Compare your measurements with the actual values. Do the actual values lie within tolerance? Show your calculations.
4. Holding one probe between the thumb and forefinger of each hand, measure and record the value of your body resistance between your hands.
5. Setup your DC PS to 3.0 volts. Measure this with your DMM.
6. Repeat step 5 for 8.0 and 6.0 volts.
7. Are the values on the display equal to the DMM reading? Why?
8. Place the resistor  $R = 1.5 \text{ K}\Omega$  on the breadboard. Setup the PS to 5.0 volts and connect it to the resistor as shown below.
9. Measure the voltage across, and the current through the resistor. Do these values match with what you expect theoretically? Explain.
10. How much power is this resistor dissipating?



## Experiment 2 Measurements on DC Circuits

### 2.1 Objectives

The objective of this experiment is to analyze simple resistive circuits in DC. The circuits considered here are: resistors in series, resistors in parallel, series- parallel combination, voltage divider, current divider, and delta combination. This experiment will allow the experimental verification of the theoretical analysis.

### 2.2 Equipment and Instruments

Power Supply ( PS), Digital Multimeter ( DMM) and Breadboard

### 2.3 Basic Information

The theoretical analysis of the circuits under study is based on Ohm's and Kirchhoff's laws. The main equations relating the electrical parameters of each circuit are presented next.

**1. Ohm's law:** The voltage  $V$  ( in volts, V) across a resistor is directly proportional to the current  $I$  (in amperes, A) flowing through it. The constant of proportionality is the resistance  $R$  (in ohms,  $\Omega$ ).

$$V = RI \quad (2-1)$$

**2. Resistors in series:** (See Figure 2-1)

- The current through  $N$  elements in series is the same for all of them.

$$I_s = I_1 = I_2 = \dots\dots\dots I_N \quad (2-2)$$

- The voltage across the  $i$ th element is  $R_i I_i$ . The sum of the voltages across each element is equal to the Voltage applied to the entire Series combination.

$$V_s = V_1 + V_2 + \dots\dots\dots + V_N = \sum_{i=1}^N V_i \quad (2-3)$$

Equation 2-3 is formulated from Kirchhoff's voltage law.

- The equivalent resistance of the series combination is the sum of the individual resistances.

$$R_{eq} = R_1 + R_2 + R_3 + \dots + R_N = \sum_{i=1}^N R_i \quad (2-4)$$

**3. Resistors in parallel:** (See Figure 2-2).

- The voltage across  $N$  elements is the same for all of them.

$$V_s = V_1 = V_2 = \dots = V_N \quad (2-5)$$

- The current through the  $i$ th element is  $V_i/R_i$ . The sum of the currents through each element is equal to the current provided to the entire parallel combination.

$$I_s = I_1 + I_2 + \dots + I_N = \sum_{i=1}^N I_i \quad (2-6)$$

Equation 2-6 is formulated from Kirchhoff's current law.

- The reciprocal of the equivalent resistance of the parallel combination is the sum of the reciprocal of the individual resistances.

$$\frac{1}{R_{eq}} = \frac{1}{R_1} + \frac{1}{R_2} + \dots + \frac{1}{R_N} = \sum_{i=1}^N \frac{1}{R_i} \quad (2-7)$$

**4. Series- Parallel combination:** An example of a series- parallel combination circuit is shown in Figure 2-4. The analysis of this type of circuit is accomplished by substituting the series (or parallel) combinations by their equivalent resistances, such that the circuit is transformed into a pure parallel (or series) circuit. Once the electrical parameters (voltages and / or currents) have been determined for the equivalent resistances, the voltages and/or currents for the individual resistors in series or parallel combinations can be obtained by using these parameters as  $V_s$  and  $I_s$  for the corresponding combination.

**5. Voltage divider:** From Equation 2-3. A series circuit with two resistors will divide the applied voltage  $V_s$  into two voltages  $V_1$  and  $V_o$  across each resistor. Notice that  $V_o$  is the output of the voltage divider (See Figure 2-4), as it is referenced to ground. The proportion in which the input is divided is given by

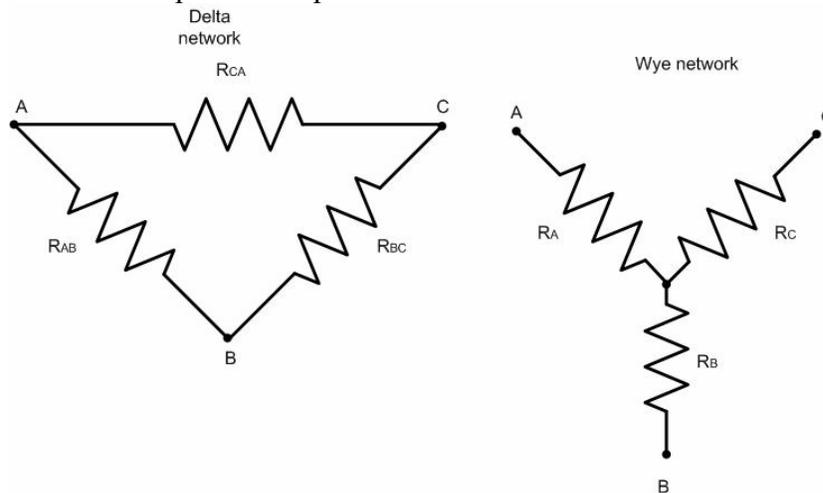
$$V_{out} = \frac{R_2 \cdot V_s}{R_1 + R_2} \quad (2-8)$$

In order for this circuit to operate as a voltage divider, the output current  $I_o$  must be zero or very small Compared with the current through  $R_2$ .

**6. Current divider:** (See Figure 2-5). From Equation 2-6, a parallel circuit with two resistors will divide the applied current  $I_s$  into two currents  $I_1$  and  $I_2$  through each resistor. The proportion in which the input current  $I_s$  is divided is given by:

$$I_3 = \frac{R_2 \cdot I_s}{R_2 + R_3} \quad (2-9)$$

**7. Delta combination circuit:** (See Figure 2-5). An example of a delta combination circuit is shown in Figure 2-6. The analysis of this type of circuit is accomplished either by mesh analysis or by using  $\Delta$  to  $Y$  Transformation formulas listed below. This technique will simplify the circuit to a simple series-parallel combination circuit that can be solved easily.



**Figure(2- 1)**

To convert a Delta ( $\Delta$ ) to Wye (Y)

To convert a Wye (Y) to Delta ( $\Delta$ )

$$R_A = \frac{R_{AB}R_{AC}}{R_{AB} + R_{AC} + R_{BC}} \quad (2-10)$$

$$R_{AB} = \frac{R_A R_B + R_A R_C + R_B R_C}{R_C} \quad (2-13)$$

$$R_B = \frac{R_{AB}R_{BC}}{R_{AB} + R_{AC} + R_{BC}} \quad (2-11)$$

$$R_{BC} = \frac{R_A R_B + R_A R_C + R_B R_C}{R_A} \quad (2-14)$$

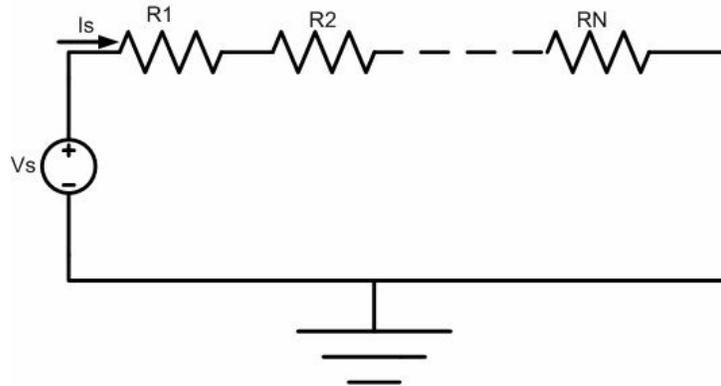
$$R_C = \frac{R_{AC}R_{BC}}{R_{AB} + R_{AC} + R_{BC}} \quad (2-12)$$

$$R_{AB} = \frac{R_A R_B + R_A R_C + R_B R_C}{R_B} \quad (2-15)$$

## 2.4 Procedure

This part of experiment requires assembling the resistive circuits presented in figures from 1 to 5 and measuring data from all of them.

- Resistors in series:** Assemble the circuit in Figure 2-1 with  $N=3$  and the components values shown in table 2-1. Take measurements to complete the entries corresponding to the experimental values.

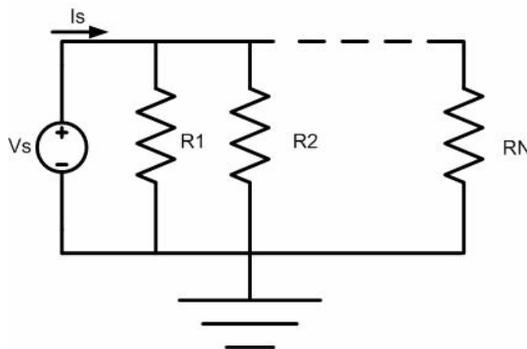


**Fig (2-2): Resistors in series.**

Parameter	$R_1$	$R_2$	$R_3$	$R_{eq}$	$V_s$	$V_{R1}$	$V_{R2}$	$V_{R3}$	$I_s$
Units	K $\Omega$				V				mA
Theoretical	1.0	2.2	5.6		10				
Experimental									
%Error									

Table 2-1: Resistors in series

- Resistors in parallel:** Assemble the circuit in Fig (2-3) with  $N=3$  and the component values shown in table 2-2. Take measurements to complete the entries corresponding to the experimental values.



**Fig (2-3): Resistors in parallel.**

Parameter	$R_1$	$R_2$	$R_3$	$R_{eq}$	$V_s$	$I_{R1}$	$I_{R2}$	$I_{R3}$
Units	K $\Omega$				V	mA		
Theoretical	1.0	2.2	5.6		10			
Experimental								
%Error								

Table 2-2: Resistors in parallel

3. **Series-parallel combination:** Assemble the circuit in figure 2-3 with the component values shown in table 2-3. Use  $V_s=10V$ . Take measurements to complete the entries corresponding to the experimental values. Notice that the resistor experimental values can be taken from the previous measurements in table 2-1 and table 2-2. Measure the voltage across each resistor and use Ohm's law (Equation 2-1) and the resistor experimental values to determine the experimental values of  $I_{Ri}$ ,  $i=1,2,\dots,4$ .

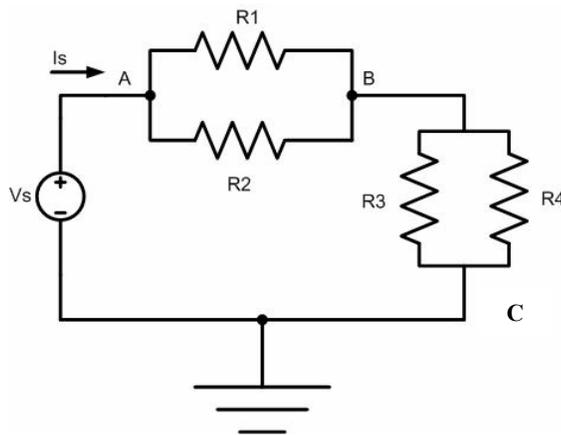
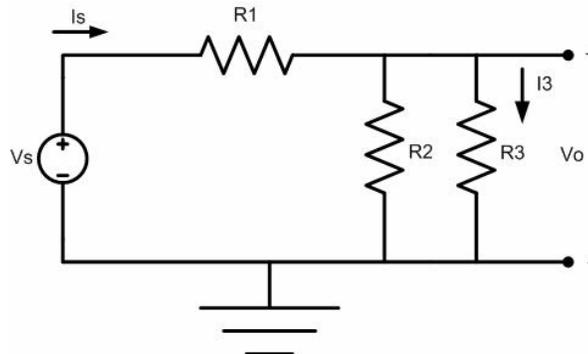


Fig 2-4: Series-parallel combination.

Parameter	Unit	Theor	Exper	%Error	Param	Unit	Theor	Exper	%Error
$R_1$	K $\Omega$	1.0			$I_{R1}$	mA			
$R_2$		2.2			$I_{R2}$				
$R_3$		5.6			$I_{R3}$				
$R_4$		8.2			$I_{R4}$				
$R_{a-c}$					$I_s$				
$V_{ab}$	V				$V_{bc}$	V			

Table 2-3: Resistors in series-parallel combination

4. **Voltage and Current divider:** Assemble the circuit in Fig 2-4 with the component values shown in table 2-4. Take measurements to complete the entries corresponding to the experimental values.



Fig(2-5): Voltage and Current divider

Parameter	$R_1$	$R_2$	$R_3$	$V_s$	$V_o$	$I_s$	$I_1$	$I_2$
Units	K $\Omega$			V		mA		
Theoretical	2.2	1.5	0.75	10				
Experimental								
%Error								

Table 2-4: Voltage and current divider

5. **Delta combination:** Assemble the circuit in Figure 2-5 with the component values Shown in table 2-5. Use  $V_s=10V$  Take measurements to complete the entries corresponding to the experimental values.

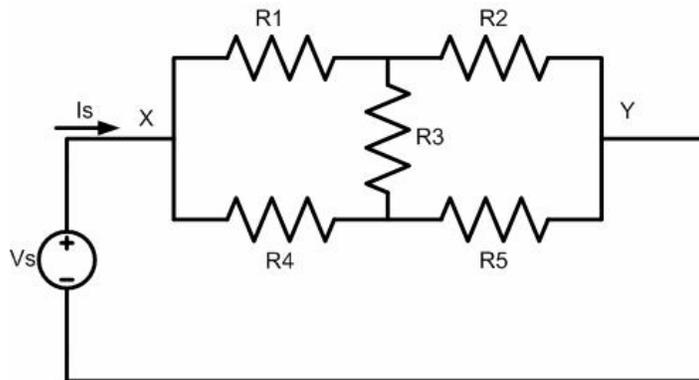


Fig 2-6: Delta combination

Parameter	$V_s$	$R_1$	$R_2$	$R_3$	$R_4$	$R_5$	$R_{x-y}$	$I_s$
Units	V	K $\Omega$						mA
Theoretical		8.2	2.2	6.2	1.0	5.6		
Experimental								
%Error								

Table 2-5 delta combination

## 2.5 Analysis

This section is intended for the analysis and comparison of the experimental and theoretical Results. Answer all the questions.

- a.** Calculate the percentage error between the measured and the theoretical data and complete all the corresponding entries in Tables 2-1 through 2-5. The percentage error is given by:

$$error\% = \frac{|d_{th} - d_m|}{d_{th}} \times 100\%$$

Where  $d_{th}$  and  $d_m$  are the theoretical and measured data respectively.

- b.** From the above results, discuss the possible causes of error.
- c.** From the results of the series- parallel combination circuit select the true statements: (a)  $I_s = I_{R1} + I_{R2} + I_{R3} + I_{R4}$ , (b)  $V_{ab} = V_s - V_{bc}$   
(c)  $I_s = I_{R1} + I_{R2} = I_{R3} + I_{R4}$
- d.** In the last circuit, Figure 2-5, the resistors are connected in what we call  $\Delta$  connection. Using mesh analysis can you solve this circuit?. but we can solve this circuit in much easier way by using another technique called delta to star transformation, by using this method find  $I_s$ ?

## **Experiment 3**

### **Techniques of circuit analysis (1)** **(Nodal, Mesh, Superposition)**

#### **3.1 Objectives**

The objective of this experiment is to analyze resistive circuits in DC, employing the node-voltage method, the mesh-current method, superposition method. Experimental results will allow the verification of the theoretical analysis.

#### **3.2 Basic Information**

In Experiment 2 the analysis of simple circuits was carried out by Kirchhoff's laws and Ohm's laws. This approach can be used for all circuits. However, for circuits with more elements and structurally more complicated a systematic approach is preferable. The techniques of circuit analysis studied here provide an aid in the analysis of more complex circuits.

#### **3.3 Techniques of circuit analysis**

The purpose of circuit analysis is to determine the current in each branch can be defined as a single path in the network, composed of one simple element and the node at each end of that element - where the current is unknown. The formulation of  $b$  independent equations with these currents as variables is achieved by applying Kirchhoff's laws. In a circuit with  $n$  nodes,  $n - 1$  equations are formulated by applying Kirchhoff's current law (KCL) to any set of  $n - 1$  nodes and the remainder  $b - (n - 1)$  equations can be written by applying Kirchhoff's voltage law (KVL) to that number of meshes in the circuit. In general,  $n_e \leq n$  and  $b_e \leq b$ , where  $n_e$  and  $b_e$  are the number of essential nodes and essential branches. Therefore, the formulation of the independent equations is achieved in terms of essential nodes and branches. However, by introducing new variables (node voltages and mesh current), the circuit can be described with just  $n_e - 1$  equations or just  $b_e - (n_e - 1)$  equations. A brief description of each method is presented next.

##### **3.3.1 Node-Voltage Method**

The node-voltage method permits the description of a circuit in terms of  $n_e - 1$  equations. Figure 3-1 shows a circuit suitable for the analysis with the node-voltage method. In this circuit three essential nodes can be identified ( $n_e = 3$ ), therefore two ( $n_e - 1$ ) node-voltage equations describe the circuit. To select the set of  $n_e - 1$  nodes to perform the analysis, one of the essential nodes is selected as a

reference node. If there is a ground node, it is usually most convenient to select it as the reference node, otherwise the node with the most branches is chosen. The equations are then written by applying KCL to each non reference node expressing the branch currents in terms of the node voltages. Once the equation system is solved and the node voltages are known, all the branch currents can be calculated.

### 3.3.2 Mesh-Current Method

The mesh-current method permits the description of a circuit in terms of  $b - (n_e - 1)$  equations. Figure 3-2 shows a circuit suitable for analysis with the mesh-current method. In this circuit five essential nodes ( $n_e = 5$ ) and eight essential branches ( $b_e = 8$ ) can be identified, therefore  $b - (n_e - 1)$  four mesh-current equations describe the circuit. The equations are formulated by applying KVL to each mesh, expressing the voltages across the elements on the mesh currents. Once the equation system is solved and the mesh currents are known, all the branch currents (and any other parameter of interest) can be calculated.

### 3.3.3 Techniques of linear circuit analysis

The previous two techniques hold for all types of circuits in general, but there are some techniques that can only be applied to only specific circuits not for all circuits, linear circuits are one of this type of circuits. Let us define a *linear* element as a passive element that has a linear voltage-current relationship. By a “*Linear voltage-current relationship*” we simply mean that multiplication of current through the element by a constant  $K$  results in the multiplication of the voltage across the element by the same constant  $K$ . We must also define a *linear dependent source* as a dependent current or voltage source whose output current or source is proportional only to the first power of a specified current or voltage variable in the circuit or to the sum of such quantities. We must define also a *Linear circuit* as a circuit composed entirely of independent sources, linear dependent sources, and linear elements.

### The Superposition Principle

Some circuits have more than one current or voltage source. Superposition theorem defines a method to determine the currents and voltages in such a circuit. This is done by considering each source at a time, while all other sources are replaced by their internal resistances. The superposition theorem states:

Current (or voltage) in any given branch of a multiple-source linear circuit can be found by determining the currents (or voltages) in a particular branch, produced by each source acting alone, with all other sources replaced by their internal resistances. The total current (or voltage) in the branch is the algebraic sum of the individual source currents (or voltages) in that branch.

The steps in applying the theorem are as follows:

1. Choose one source at a time, and replace all other voltage sources with a short circuit ( $R=0$ ), and all other current sources with an open circuit ( $R=\infty$ ).
2. Determine the currents and voltages required.
3. Repeat steps 1 & 2 for each source in the circuit.
4. The total current or voltage in a branch is the algebraic sum of the individual source currents or voltages in that branch.

### 3.4 Procedure

This part of experiment requires assembling the resistive circuits presented in the previous section and measuring data from all of them.

#### 1. Node-voltage method.

- a. Assemble the circuit in Figure 3-1 with the component values shown in Table 3-1.

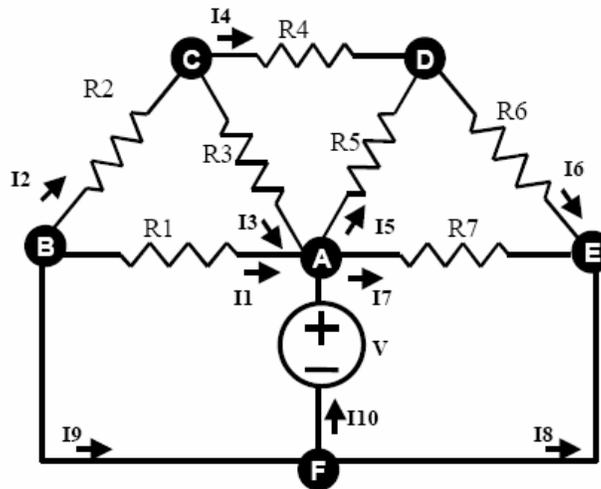


Figure 3-1

Parameter	Unit	Theoretical	Experimental	%Error
R <sub>1</sub>	KΩ	4.7		
R <sub>2</sub>		1.5		
R <sub>3</sub>		1.8		
R <sub>4</sub>		3.3		
R <sub>5</sub>		1.0		
R <sub>6</sub>		2.2		
R <sub>7</sub>		6.8		
V	V	10		

Table 3-1

b. Measure the voltage and current across and through each resistance respectively

Parameter	Unit	Theoretical	Experimental	%Error	
$V_{AF}$	V				
$V_{AB}$					
$V_{AC}$					
$V_{AD}$					
$V_{AE}$					
$V_{BC}$					
$V_{CD}$					
$V_{DE}$					
$I_1$		mA			
$I_2$					
$I_3$					
$I_4$					
$I_5$					
$I_6$					
$I_7$					
$I_8$					
$I_9$					

Table 3-2

**2. Mesh-current method.**

a. Assemble the circuit in Figure 3-2 with the component values shown in Table 3-3.

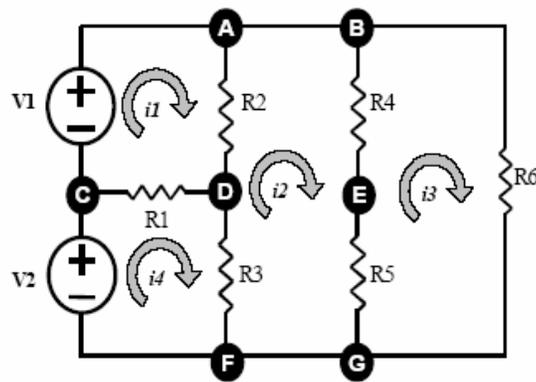


Figure 3-2

Parameter	Unit	Theoretical	Experimental	%Error
R <sub>1</sub>	KΩ	2.2		
R <sub>2</sub>		3.3		
R <sub>3</sub>		4.7		
R <sub>4</sub>		6.8		
R <sub>5</sub>		5.6		
R <sub>6</sub>		8.2		
V <sub>1</sub>	V	10		
V <sub>2</sub>		5		

Table 3-3

b. Measure the voltage and current across and through each resistance respectively.

Parameter	Unit	Theoretical	Experimental	%Error
V <sub>AD</sub>	V			
V <sub>DF</sub>				
V <sub>BE</sub>				
V <sub>EG</sub>				
V <sub>CD</sub>				
V <sub>BG</sub>				
V <sub>FA</sub>		mA		
I <sub>1</sub>				
I <sub>2</sub>				
I <sub>3</sub>				
I <sub>4</sub>				
I <sub>DC</sub>				
I <sub>AD</sub>				
I <sub>DF</sub>				

Table 3- 4

### 3. Superposition.

Determine the experimental values in circuit Figure 3-2 by using superposition

- Replace V<sub>1</sub> with a short circuit, keep V<sub>2</sub> ON, Take measurements to complete the entries corresponding to the experimental values in table 3-5. **Be careful to follow the passive sign convention.**
- Replace V<sub>2</sub> with a short circuit, keep V<sub>1</sub> ON, Take measurements to complete the entries corresponding to the experimental values in table 3-6.

Parameter	Unit	Theoretical	Experimental	%Error
$V'_{AD}$	V			
$V'_{DF}$				
$V'_{BE}$				
$V'_{EG}$				
$V'_{CD}$				
$V'_{BG}$				
$V'_{FA}$		mA		
$I'_1$				
$I'_2$				
$I'_3$				
$I'_4$				
$I'_{DC}$				
$I'_{AD}$				
$I'_{DF}$				

Table 3- 5

Parameter	Unit	Theoretical	Experimental	%Error
$V''_{AD}$	V			
$V''_{DF}$				
$V''_{BE}$				
$V''_{EG}$				
$V''_{CD}$				
$V''_{BG}$				
$V''_{FA}$		mA		
$I''_1$				
$I''_2$				
$I''_3$				
$I''_4$				
$I''_{DC}$				
$I''_{AD}$				
$I''_{DF}$				

Table 3- 6

#### **4. Analysis**

1. From your results in table 3-2, prove that KCL holds true. Write a nodal equation at each node, do the summation. For your equations, choose Current going out of the node. Each node should sum to 0.
2. From your results in table 3-4, prove that KVL holds true. Write a loop (mesh) equation for each loop. Do summation. Each mesh should sum up to 0.
3. From your results in tables 3-4, 3-5 and 3-6, prove that superposition holds true. The summation of each voltage and each current in table 3-5, with its corresponding in table 3-6, should be equal to the corresponding values in table 3-4.
4. The Ammeter readings that you observe in the experiment, was a mesh current of a branch current? Justify your answer?

**Experiment 4:**  
**Techniques of circuit analysis (2)**  
**(Thevenin, Norton, Max. power transfer).**

#### 4.1 Objectives

The objective of this experiment is to simplify resistive circuits in DC employing the Thevenin and Norton equivalents. Experimental results will allow the verification of the theoretical analysis. Also to verify the maximum power transfer theorem, this will lead us to learn about some types of variable resistors such as decade resistance box and potentiometers.

#### 4.2 Basic Information

In experiment 2 was demonstrated the use of series-parallel reductions of resistive circuits in order to simplify the circuit analysis. In this experiment, Thevenin and Norton equivalent circuits are discussed as additional methods to simplify the circuit analysis.

##### 4.2.1 Thevenin's Theorem

Any linear network may, with respect to a pair of terminals, be replaced by an equivalent voltage source  $V_{Th}$  (equal to the open circuit voltage) in series with a resistance  $R_{Th}$  seen between these terminals. The theorem presents a very useful method for simplifying complex circuits. We must realize that the equivalency is with respect to a selected pair of terminals. We replace the original circuit lying on one side of a pair of terminals by its equivalent Thevenin voltage source and resistance. To evaluate  $V_{Th}$  and  $R_{Th}$  follow the steps:

1. Open-circuit the terminals to which the Thevenin equivalent is desired. In other words disconnect all of the circuitry that will not be replaced by the equivalent circuit.
2.  $R_{Th}$  is the total resistance at the open-circuit terminals, when all voltage sources are replaced by short circuits, and open circuits replace all current sources.
3.  $V_{Th}$  is the voltage across the open-circuited terminals.
4. Replace the original circuitry by its Thevenin equivalent circuit, with the Thevenin terminals occupying the same position as the original terminals. The external circuit removed in step 1 may now be reconnected.

#### 4.2.2 Norton's Theorem

Any linear network may, with respect to a pair of terminals, be replaced by a current source  $I_N$  (equal to the short-circuit current) in parallel with the resistance  $R_{Th}$  seen between the two terminals. Similar to Thevenin's Theorem, we are presented with a simple method of reducing a more complex circuit to a simpler one. The difference lies in converting the reduced circuit to a current source instead of a voltage source, with the equivalent resistance in parallel rather than in series. To evaluate  $I_N$  and  $R_N$  follow the steps:

1. Follow the procedure in step 1 of Thevenin's equivalents.
2.  $R_N = R_{Th}$  calculated in step 2 above.
3.  $I_N$  is the short circuit current passing through the terminals.
4. Replace the original circuitry by its Norton equivalent circuit, with the Norton terminals occupying the same position as the original terminals. The external circuit removed in step 1 may now be reconnected.

#### 4.2.3 Maximum Power Transfer Theory

An independent voltage source in series with a resistance  $R_s$ , or an independent current source in parallel with a resistance  $R_s$ , delivers maximum power to a load resistance  $R_L$  when  $R_s = R_L$ .

The maximum power delivered, in this case, is

$$P_{\max} = \frac{V_s^2}{4R_s} \quad (4-1)$$

#### 4.2.4 Variable resistors

In addition to fixed-value resistors – which we dealt with in the previous three experiments- variable resistors, are used extensively in electrical and electronic circuits. The two types of variable resistors are rheostat and the potentiometer. A rheostat is essentially a two terminal device. Its circuit symbol is shown in Figure 4-1. Points A and B connect in to the circuit. A rheostat has a maximum resistance value, specified by the manufacturer, and a minimum value, usually  $0 \Omega$ . The arrowhead in the figure indicates a mechanical means of adjusting the rheostat so that the resistance, measured between points A and B, can be adjusted to any intermediate value within the range of variation.

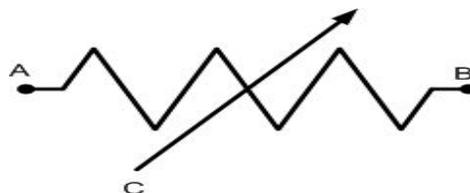


Fig. 4-1 A Rheostat is a variable resistor with two terminals

The circuit symbol for a potentiometer Figure 4-2, shows that this is a three-terminal device. The resistance between points A and B is fixed. Point C is the variable arm of the potentiometer. The arm is a metal contactor that slides along the un-insulated surface of the resistance element. The amount of resistance material between the point of contact and one of the end terminals determines the resistance between those two points. Thus the longer the surface between points A and C, the greater is the resistance between these two points. In other words, the resistance between points A and C varies as the length of the element included between points A and C. The same is true for points B and C. The resistance  $R_{AC}$  from A to C plus the resistance  $R_{CB}$  from C to B make up the fixed resistance  $R_{AB}$  of the potentiometer. The action of the arm is to increase the resistance between C and one of the end terminals and at the same time to decrease the resistance between C and the other terminal while the sum of the two resistances  $R_{AC}$  and  $R_{CB}$  remains constant.

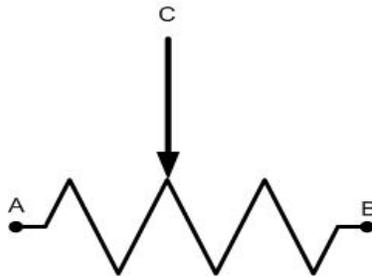


Fig. 4-2 A potentiometer

A potentiometer may be used as a rheostat if the center arm and one of the end terminals are connected into the circuit and the other end terminal is left disconnected. Another method of converting a potentiometer in to a rheostat is to connect a piece of wire between the arm and one of the end terminals; for example, C can be connected to A. The points B and C now serve as the terminals of a rheostat. One of the most important uses of potentiometers is a variable voltage divider, the figure below illustrates this.

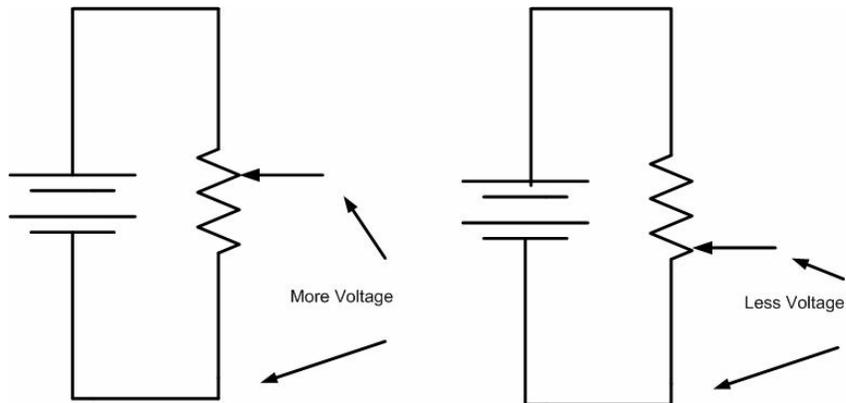


Figure 4-3 using a potentiometer as a variable voltage divider.

Just like the fixed voltage divider, the **potentiometer**'s voltage *division ratio* is strictly a function of resistance and not of the magnitude of applied voltage. In other words, if the **potentiometer** knob or lever is moved to the 50 percent (exact center) position, the voltage dropped between wiper and either outside terminal would be exactly 1/2 of the applied voltage, no matter what that voltage happens to be, or what the end-to-end resistance of the **potentiometer** is. In other words, a **potentiometer** functions as a variable voltage divider where the voltage division ratio is set by wiper position. This application of the **potentiometer** is a very useful means of obtaining a variable voltage from a fixed-voltage source such as a battery. If a circuit you're building requires a certain amount of voltage that is less than the value of an available battery's voltage, you may connect the outer terminals of a **potentiometer** across that battery and "dial up" whatever voltage you need between the **potentiometer** wiper and one of the outer terminals for use in your circuit:

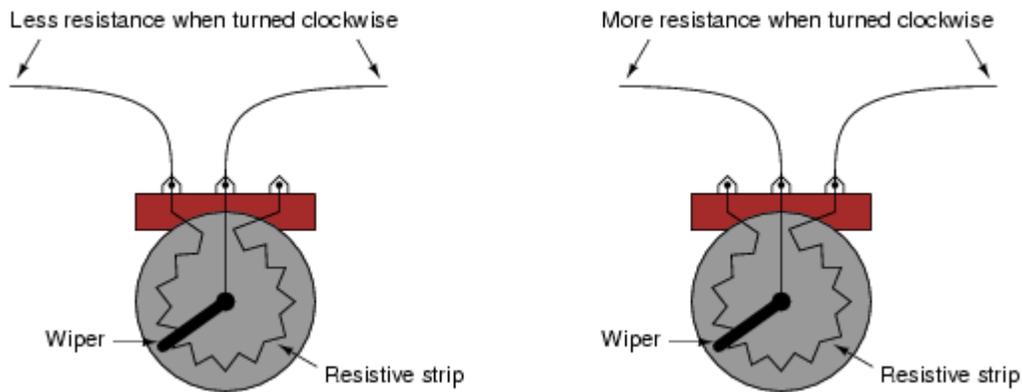
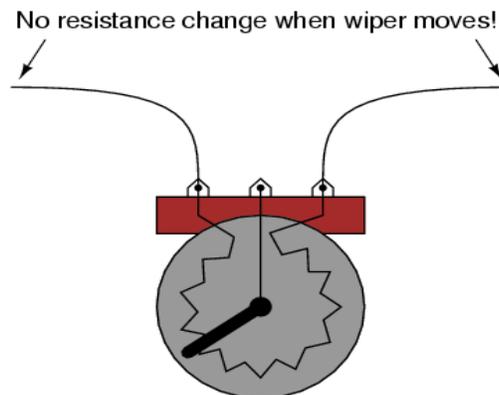


Figure 4-4 (a) Movement of potentiometer.

Be careful, though, that you don't use the two outer terminals, as this will result in *no change in resistance* as the potentiometer shaft is turned. In other words, it will no longer function as a *variable* resistance:



(b)

Figure 4-4 Construction of potentiometer

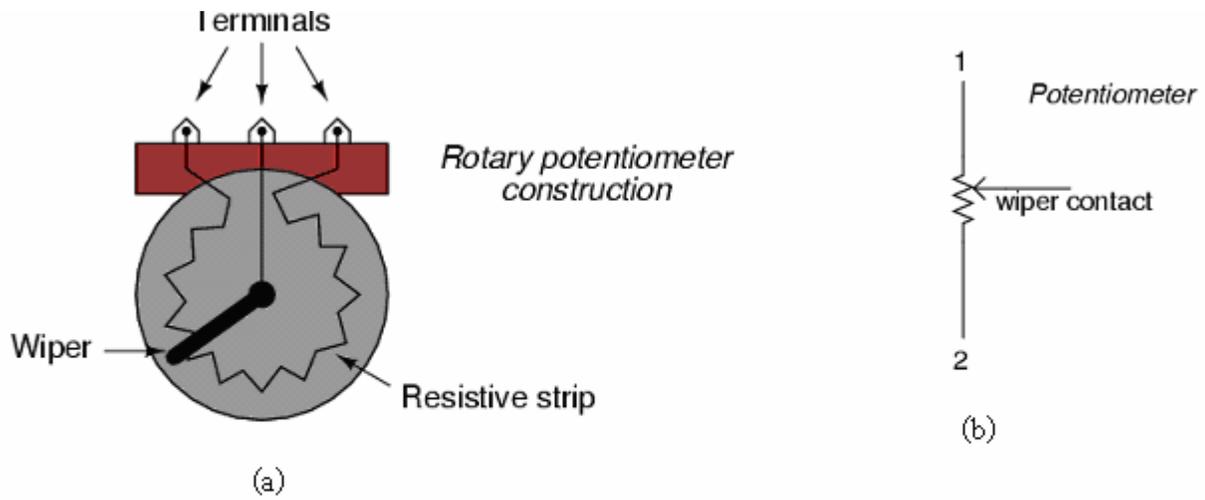


Figure 4-7 Construction of Rotary Potentiometer

### 4.3 Procedure

#### 1. Thevenin and Norton Equivalents:

- a. Assemble the circuit in Figure 4-6 with the component values shown in Table 4-1. ( $R_L$  is 10  $K\Omega$  potentiometer (Pot), **in this section there is no need to connect this Pot. in the circuit.**)

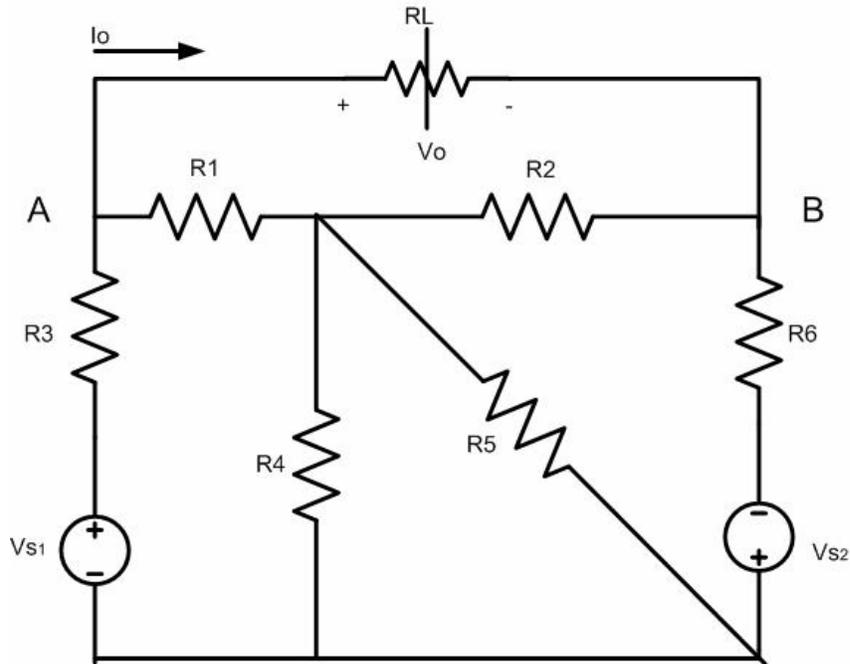


Fig. 4-6

Parameter	Unit	Theoretical	Experimental	%Error
$R_1$	K $\Omega$	1.0		
$R_2$		1.5		
$R_3$		1.0		
$R_4$		3.3		
$R_5$		2.2		
$R_6$		1.0		
$R_{th}$				
$V_{s1}$	V	12		
$V_{s2}$		5		
$V_{th}$				
$I_{sc}$	mA			

Table 4-1

b.  $V_{th}$  measurement:

to measure  $V_{OC}$ , disconnect  $R_L$ , and measure  $V_{AB}$ . Where  $V_{th} = V_{OC} = V_{AB}$ .

c. Isc measurement:

to measure Isc place, A-B terminals should be Shorted by the Ammeter. (Connect the Ammeter across A- B Terminals).

d. Rth Measurement:

to measure Rth, short sources  $V_{S1}$ ,  $V_{S2}$  and Connect the Ohmmeter across terminals A-B.

2. Maximum Power Transfer:

a. Set the pot. To 1K $\Omega$ , reconnect it in the circuit, Measure the voltage across the pot.

Calculate the power dissipated by the pot.

b. Increase the pot. Resistance in steps as illustrated in Table 4-2

c. Take measurements corresponding to the entries in the table.

d.

$R_L$ (K $\Omega$ )			$V_L$ (V)			$P = \frac{V_L^2}{R_L}$ mW		
Theor.	Exper.	%Error	Theor.	Exper.	%Error	Theor.	Exper.	%Error
0.22								
0.33								
2.2								
1								
3.3								
4.7								
47								
100								

Table 4-2

3. Potentiometer Application Circuit, Loaded Voltage Divider fig4-7:

In the circuit below, we want to analyze the effect of load on the voltage divider circuit, to verify this effect experimentally, its necessary to have a voltage source a cross the divider that will remain constant with or with out load, in the circuit below the divider network consists of  $R_1, R_2, R_3$  connected to a voltage source, a potentiometer is connected to the divider circuit as a variable load. By adjusting the potentiometer the value of the load will change ( $R_L$ ), this will affect both  $I_1$  and  $I_L$ , in order to analyze the loading effect we want to detect the changes in values of both currents, to do that follow the following steps:

- a. Assemble the circuit in Figure 4-7 with the component value shown in table 4-3.

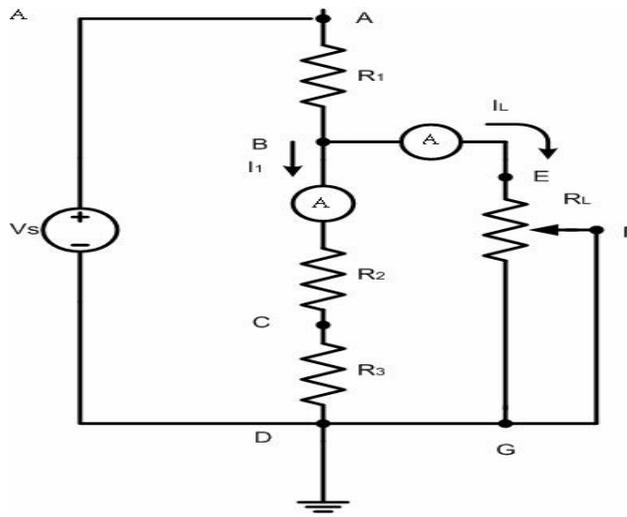


Fig. 4-7

Parameter	Unit	Theoretical	Experimental	%Error
$R_1$	K $\Omega$	1.8		
$R_2$		1.2		
$R_3$		1.5		
$R_L$ (POT)		10		
$V_s$	V	12		

Table 4-3

- b. Adjusting the potentiometer until  $I_L = 2\text{mA}$ , record the measurements required in table 4-4, repeat this step for various values of  $I_L$  as illustrated in the table. For each value of  $I_L$  record the measurements required.

I <sub>L</sub> (load current) mA	I <sub>1</sub> mA		V <sub>BD</sub> V		V <sub>CD</sub> V		R <sub>L</sub> Ω	
	Theor.	Exper.	Theor.	Exper.	Theor.	Exper.	Theor.	Exper.
2								
4								
6								

Table 4-4

#### 4. Analysis

1. Plot on graph paper, a graph of the power dissipated in the pot. vs. its resistance. Indicate the point of maximum power on the graph.
2. Calculate the Thevenin's equivalent circuit that can be seen from Points A,B. Calculate  $P_{MAX}$ .
3. Show whether the maximum power transfer theorem holds for this circuit or not.
4. Explain, in your words, the effect of load on the voltage relationships in a voltage divider circuit.
5. Refer to your data in table 4-4. What effect does changing the load resistance have on load current? Give specific examples from your data. Explain the reason for this effect?
6. Refer to your data in table 4-4. How is current I<sub>1</sub>, affected by changes in the load current, I<sub>L</sub>? Give specific examples from your data. Explain the reason for this effect?
7. Refer to your data in table 4-4 and figure 4-2. What effect do changes in the load current I<sub>L</sub>, have on the divider tap voltages, V<sub>CD</sub> and V<sub>BD</sub>? Give specific examples from your data. Explain the reason for this effect?

## Experiment 5 The Function Generator and Oscilloscope

### 5.1 Objectives:

To introduce time varying and periodic signals.  
To introduce the Function Generator as lab tool  
To introduce the Cathode Ray Oscilloscope as a lab tool.

### 5.2 Basic Information:

#### 5.2.1 Time varying and periodic signals:

Time varying signals are signals whose values change with time  $f = f(t)$  [Fig 5.1].

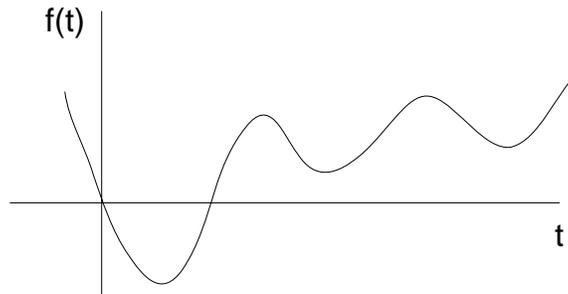


Fig 5.1

Examples of such signals include:

- The ramp signal  $f(t) = t$ .
- The sinusoidal signal  $f(t) = A \sin(\omega t)$ .
- The exponential signal  $f(t) = e^t$ .

An important subset of time varying signals is the set of periodic signals [Fig. 5.2], defined by the equation:

$$f(t) = f(t+T)$$

Where  $T$  is the period of the signal.

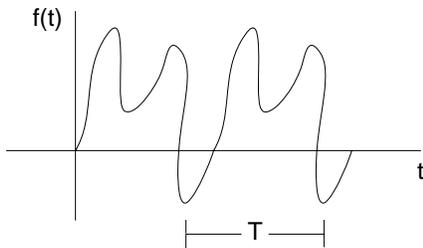


Fig. 5.2

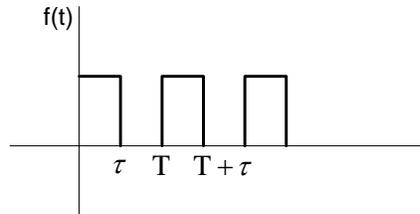


Fig. 5.3

A common periodic signal is the square wave [Fig 5.3], defined by the equation:

$$f(t) = \sum_{n=-\infty}^{n=+\infty} A[u(t - nT) - u(t - n(T - \tau))] \quad (5-1)$$

Where  $\tau$  is the width, if the pulse T is the period, and  $u(t)$  is the unit step function.

### 5.2.2 The Sinusoidal Signals:

An important periodic signal is the sinusoidal signal.

$$f(t) = A \sin(\omega t + \theta) \quad (5-2)$$

In studying this signal several important parameters can be identified. Given the signal

$$V(t) = A \sin(\omega t + \theta) = A \sin(2\pi f t + \theta) \quad (5-3)$$

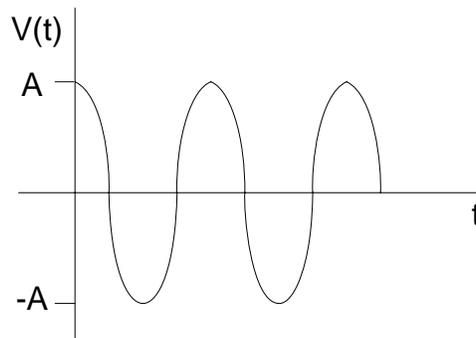


Fig. 5.4

Where:

- $A$  : The amplitude of the signal, also known as the peak voltage  $V_p$ .
- $\omega$  : the radian frequency measured in rad / sec.
- $f$  : The frequency measured in Hz.
- $V_{p-p}$  :  $2A$ , the peak to peak voltage.
- $T$  : The period, equal to the reciprocal of the frequency.
- $\theta$  : The phase angle of the signal.

Another important parameter of the sinusoidal signal is the RMS value, which stands for the RMS. It is calculated by the formula:

$$V_{RMS} = \frac{A}{\sqrt{2}}$$

The RMS value is defined as the effective value of the sinusoidal signal, equal to the DC signal would deliver the same power if applied to the same resistor.

For RMS value for v is given in equation (5-4)

$$v = V_m \cos(\omega t + \phi)$$

$$V_{rms} = \sqrt{\frac{1}{T} \int_{t_0}^{t_0+T} V_m^2 \cos^2(\omega t + \phi) dt}$$

The RMS values for the triangle and the square wave are:

$$V_{RMS} = \frac{A}{\sqrt{3}}, \text{ for triangle wave.}$$

$$V_{RMS} = A, \text{ for square wave.}$$

Where A is the peak voltage.

### 5.2.3 The Function Generator (FG):

The function generator is an instrument that delivers a choice of different waveforms, with adjustable frequency over a wide range. The most common waveforms are: sine, triangle and square.

The value of the current is controlled by the frequency control circuit, the constant current is fed to an integration circuit, the output of which is a triangular signal. A comparator uses the triangular wave to supply a sine wave. While the sine shaping circuit converts the triangular wave into a sinusoidal signal. Through a selector the amplifier provides the output signal.

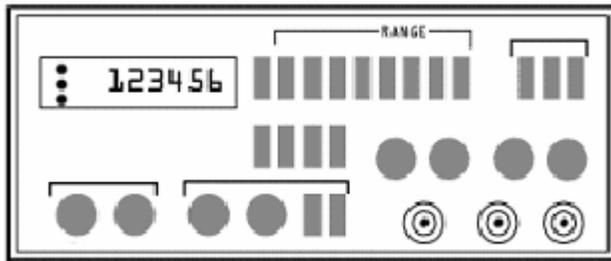


Fig. 5.5 Front view of a FG

The following table describes the controls and indicators of the FG used in lab.

<b>COUNTER DISPLAY</b>	Display the frequency of the internally generated waveform
<b>RANGE switch</b>	Select output frequency range. Seven ranges from 1Hz to 10 MHz, for example if the 100 KHz range is selected, the output frequency can be adjusted from 10 KHz to 100 KHz.

<b>FREQUENCY CONTROL</b>	Sets the desired frequency within the range selected. This is done by setting both the CAORSE and FINE controls.
<b>FUNCTION switch</b>	Selects Sine, Square or triangle waveform as OUTPUT jack
<b>OUTPUT LEVEL control</b>	Controls the Peak-to-Peak amplitude of the signal at the output jack.
<b>-20 dB switch</b>	For amplitude attenuation at -20 dB level.
<b>OUTPUT jack</b>	The output signal.
<b>TTL/CMOS jack</b>	A square (TTL/CMOS) wave and it depends on the position of the CMOS LEVEL switch.
<b>DC OFFSET control</b>	This control is enabled by the DC OFFSET switch.
<b>DUTY CYCLE control</b>	Enabled by the DUTY CYCLE switch and by rotating this dial the duty cycle of the main signal is adjusted
<b>GATE LED</b>	Indicates when the frequency counter display is updated. When selecting any of the RANGE switches. The LED will flash number of times depending on the selected range. As the LED turns off , the display is updated.

Table 5.1: Controls and Indicators

### 5.2.4 The Cathode Ray Oscilloscope (CRO):

The CRO displays the amplitude of the electrical signals, on screen, as a function of time. The horizontal axis of the CRO is deflected at a constant time rate. While the vertical axis is deflected in response to the amplitude of an input signal. The heart of the oscilloscope [ Fig. 5.6] is the cathode ray tube (CRT). The CRT generates an electron beam, accelerates the beam to high velocity, and deflects it to create the image. The oscilloscope has a time base generator which supplies the correct voltage to the CRT to deflect the beam ( a spot on the screen) horizontally, at a constant time rate. The signal under study is fed to the vertical amplifier, to increase its amplitude, and then to the CRT to deflect the beam vertically. To synchronize the horizontal and vertical deflections, a triggering circuit is used.

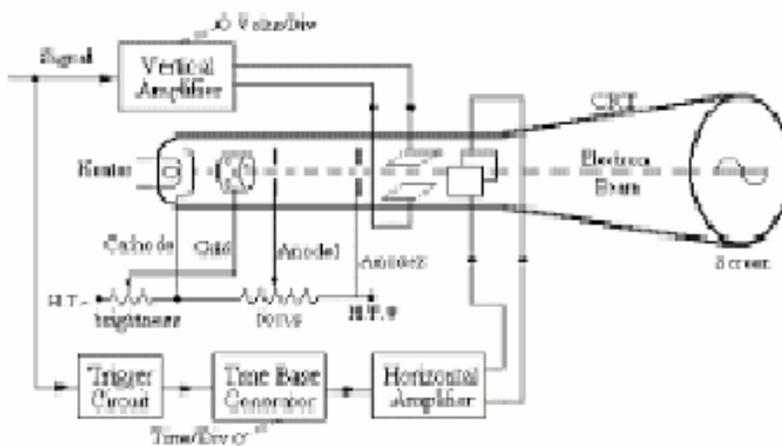


Fig. 5.6 Block diagram of the oscilloscope

The front panel of the CRO used in the lab can be divided into these main sections:

- 1- CRT controls.
- 2- Horizontal controls
- 3- Vertical controls

### 5.2.5 DC offset:

A DC offset is the value of the DC voltage added to a time varying signal.

$$V(t) = d + asim(\omega t + \theta)$$

$$DC\ offset = \frac{B + C}{2} \tag{5-5}$$

Where d is the DC offset.

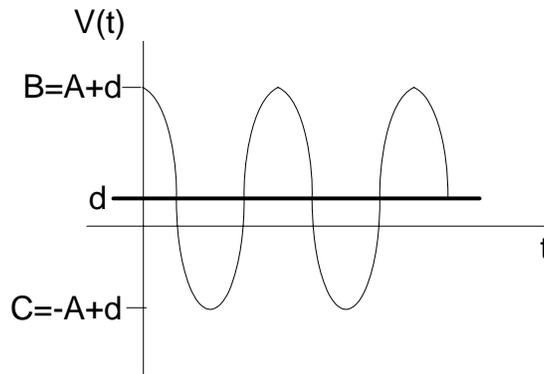


Fig. 5.7

### 5.2.6 Duty Cycle:

The duty cycle is defined as the ratio of the pulse width t (positive part) to the period T, usually expressed as percentage.

$$Duty\ Cycle = \frac{t}{T} \times 100 \tag{5-6}$$

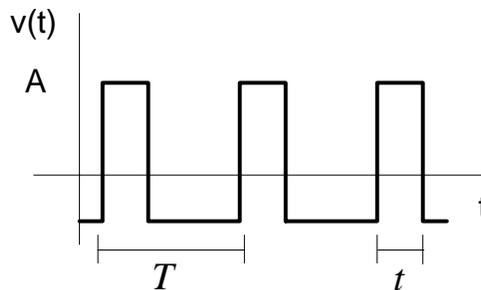


Fig. 5.8 A pulse wave

### 5.2.7 Rise and Fall time:

When investigating pulse or square waves, the critical feature is the rise time of the voltage step. The rise time of a square wave is defined, as the time required the signal to increase from 10% to 90% of its amplitude. Similarly, the fall time of a square wave is defined, as the time required the signal to decrease from 90% to 10% of its amplitude.

If the rise time of the signal is less than 100ns, that it has very fast rise time, the rise time of the oscilloscope and probe will affect the measurements.

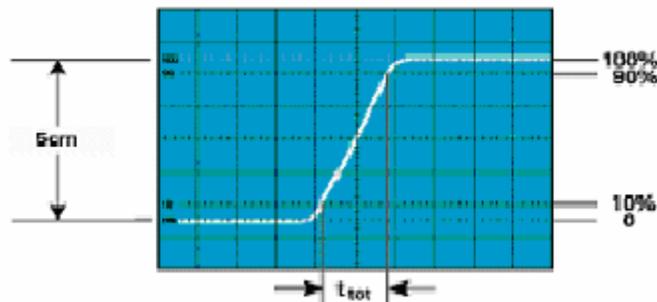


Fig. 5.9

The rise time of the signal can be calculated using the following formula.

$$t_r = \sqrt{t_{tot}^2 - t_{osc}^2 - t_p^2}$$

Where:

$t_r$  : The true rise time of the signal.

$t_{tot}$  : The total measured rise time.  $t_{90\%} - t_{10\%}$ .

$t_{osc}$  : The rise time of the oscilloscope amplifier. In our case is approx. 8.75ns.

$t_p$  : The rise time of the probe. In our case is approx. 2ns.

Same method is used for fall time.

### 5.3 Procedure:

1. switch on the CRO and FG and allow some time to warm up.
2. With the DMM connected to the CRO probe, measure the input resistance for the DC and AC input couplings. Comment on this results.
3. Connect the OUTPUT jack of the FG to the CRO.
4. Adjust the FG to provide a triangle signal with  $V_{pk-pk} = 4$  V and  $f = 60$  KHz.
5. Measure the signal values from the CRO display, and plot it on a graph paper.

6. Obtain a square wave  $V_{pk} = 4V$  ,  $f = 1MHz$ . Measure the rise time of this signal.
7. Obtain a square wave  $V_{pk} = 4V$  ,  $f = 100 kHz$ .
8. Use dual mode to compare OUTPUT and TTL/CMOS outputs in amplitude, frequency, waveform , phase. Plot both signals in one graph paper using the same time scale.
9. Use the ADD function of the oscilloscope to get OUTPUT +TTL/CMOS. Set both channels to the same volts/div scale. Plot the resulting waveform.
10. Invert CH2 and get OUTPUT -TTL/CMOS, Plot the resulting waveform.
11. Obtain a sine wave with  $V_{pk-pk} = 8 V$ ,  $f = 15KHz$ . Press the DC OFFSET button, increase the dc offset until the sine wave starts to be distorted.
12. Measure from the CRO the maximum value of DC offset at which an undistorted output is possible. Set the DMM to measure a DC voltage and measure the OUTPUT jack. Compare with the CRO results.
13. Obtain a square wave with  $V_{pk-pk} = 4 V$ ,  $f = 10KHz$ .. Press the DUTY CYCLE button, adjust until the wave has a 20% duty cycle. Plot the signal on a graph paper.
14. Connect the circuit shown on Fig. 5.10. Set the input coupling to DC. Use the circuit to calculate the unknown resistor  $R_x$ . Compare this with the color code.

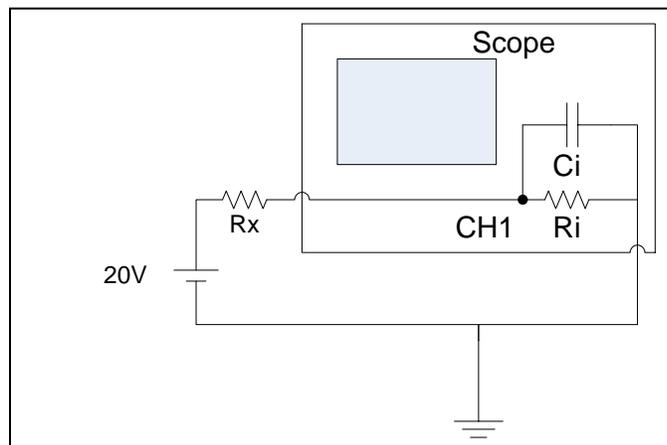


Fig. 5.10 An unknown resistance connected to an oscilloscope

## Experiment 6

### Basic laws on AC circuits (RMS, Average value, KVL, KCL).

#### Objective

1. To introduce time varying and periodic signals.
2. To implement Ohm's law, KVL and KCL for AC circuits.

#### Basic Information

The basic laws of circuit theory are applicable to AC signals the same way they are for DC signals:

a- Ohm's Law :  $V = Ri(t)$

b- KVL: the algebraic sum of the voltages around any closed path in a circuit is zero.  $\sum_{i=0}^N V_i = 0$ . Where  $N$  is the number of drop voltages around any closed loop.

c- KCL: the algebraic sum of the currents around entering any node is zero.  $\sum_{i=0}^N I_i = 0$ . Where  $N$  is the number of entering currents through any node.

#### Equipment

Function Generator (FG), Cathode Ray Oscilloscope (CRO), Digital Multimeter (DMM), and various components.

**Procedure**  
*Part A*

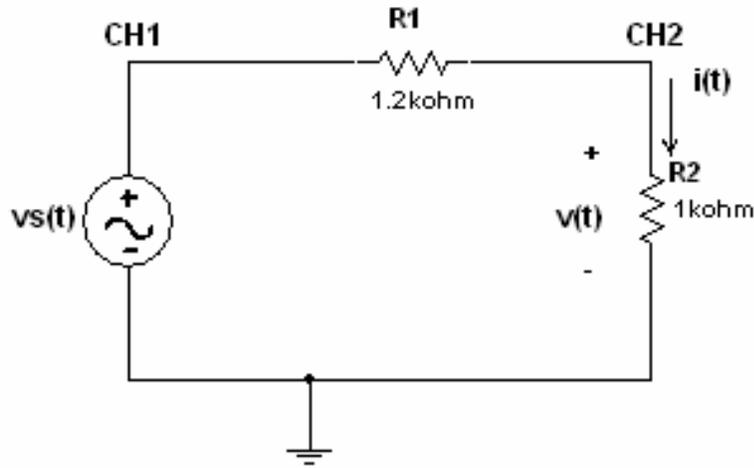


Fig.6.1 Basic AC Measurements

- Connect the circuit shown in figure1
- Complete the table below *and always check the source signal after connection*

Source	$v_s(t)$ (Volt)	Frequency (KH)	$v(t)$ (Volt)	$i(t)$ (mA)	$v(t)_{RMS}$ (Volt)	$i(t)_{RMS}$ (mA)	<i><b>Plot of <math>v(t)</math> and <math>i(t)</math> on the same time scale</b></i>
Sinwav	5p-p	1.5					
Sinwav	5p-p	15					
Sinwav	10p-p	15					
Sawtoot	10p-p	2					
Sawtoot	10p-p	15					

Table 6.1

- Write down the formulas for  $v(t)$  and  $i(t)$  as a function of time (just for the sine-wave).
- Calculate the RMS values of the table above and compare your results with the measured one.

## Part B

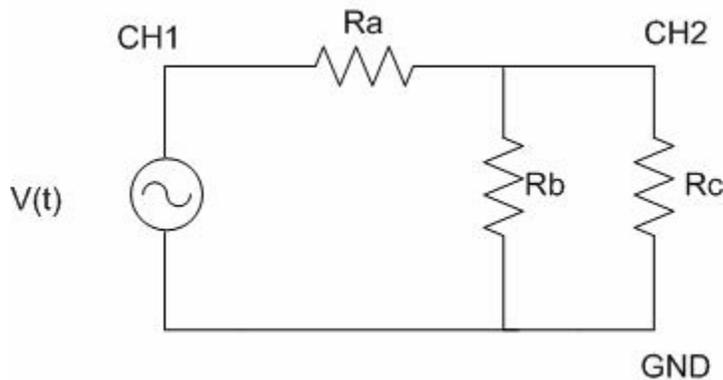


Fig.6.2 Basic AC circuit.

1. Generate  $v(t)$ , a sinusoidal signal with  $V_{PK} = 2.0$  volts.  $\omega = 15.7\text{Krad/s}$ .
2. Assemble the circuit in figure 2. Where  $R_a = 1.8\text{K}\Omega$ ,  $R_b = 1\text{K}\Omega$ ,  $R_c = 750\Omega$ . Check and adjust, if necessary, the source signal after connection.
3. Use the CRO to display the signals  $V_a(t)$ ,  $V_b(t)$ ,  $V_c(t)$ , and draw them Underneath each other on one sheet of graph paper using the same scale.

**Important note:**  $V_a(t)$  cannot be measured by placing the CRO probes directly across  $R_a$ , as this will ground the resistors  $R_b$  and  $R_c$ . To avoid this, place CH1 & CH2 probes across both  $R_a$  and  $R_b$ , invert CH2 display and add CH1 to CH2 to get  $CH1-CH2 = V_a(t)$ .

4. Show that KVL applies to the time functions of voltages in this circuit.
5. Calculate the current in each one of the resistors. Does KCL apply? Show that from your calculations.
6. Measure the RMS values of all currents and voltages in the circuit.
7. Calculate the above RMS values and compare with the measurements. Show that KVL and KCL apply to The RMS values.

## Questions

1. From the experiment you have done, Do you think the amplitude or the or the frequency of the source will affect the RMS value or not, Why? Justify your answer.
2. Does the phase shift in the source voltage affect the RMS value? Why?
3. Does the DC offset in the source voltage affect the RMS value or not, Why?

## Experiment7 Step Response of AC Circuits.

### 1 Objectives

In this experiment the natural and step responses of RL circuits are examined. The use of computer controlled equipment is also introduced here.

### 2 Introduction and Test Circuits

Inductors and capacitors have the ability to store energy. It is important to determine the voltages and currents that arise in circuits composed by resistors, and either inductors or capacitors, when energy is *released* or *acquired* by the inductor or capacitor as a consequence of an abrupt change in the DC voltage or current in the circuit. The description of the voltages and currents in this type of circuits is done in terms of differential equations of *first order*.

#### 2.1 Natural Response

The currents and voltages that arise when the energy stored in an inductor or capacitor and suddenly released to the resistors in the circuit are referred to as the *natural response* of the circuit. The behavior of these currents and voltages depends only on the nature of the circuit, and not on external sources of excitation.

##### 2.1.1 Natural Response of an RL Circuit

In an RL circuit, the natural response is described in terms of the voltage and current at the terminals of the resistor when the external source of power stops delivering energy to the circuit. The expressions for the current and voltage across the resistor are:

$$i(t) = I_0 e^{-t/\tau}, \quad t \geq 0 \quad (7-1)$$

$$v(t) = I_0 R e^{-t/\tau}, \quad t \geq 0 \quad (7-2)$$

Where  $I_0$  is the initial current through the inductor before the power source goes off and the inductor starts releasing energy to the circuit.

The symbol  $\tau$  represents the *time constant* of the circuit,

$$\tau = L / R \quad (7-3)$$

An RL circuit is shown in Fig. 7-5. Here,  $V_s$  provides a square signal with a DC offset voltage such that the bottom part of the waveform is aligned with the zero volts level.  $R_S$  is the internal resistance of the voltage source, hence when the square wave takes the value of

zero volts (and can be viewed as a short circuit) the energy of the inductor  $L$  is released through the combination of  $R_1$  and  $R_S$ . In order to observe the natural response of the circuit, the period  $T$  of the square wave must be long enough to allow the complete charge and discharge of the inductor. Usually  $T = 20\tau$  is appropriate for this purpose.

Summarizing, the natural response of an RL circuit is calculated by (1) finding the initial current  $I_o$  through the inductor, (2) finding the time constant of the circuit (Eq. 7-3), and (3) using Eq. 7-1 to generate  $i(t)$ .

### 2.1.2 Natural Response of an RC Circuit

The natural response of an RC circuit is analogous to that of an RL circuit. The expressions for the current and voltage across the resistor are:

$$V(t) = V_o e^{-t/\tau}, \quad t \geq 0 \quad (7-4)$$

$$i(t) = \frac{V_o}{R} e^{-t/\tau}, \quad t \geq 0 \quad (7-5)$$

$$\tau = RC \quad (7-6)$$

Where  $V_o$  is the initial voltage across the (fully charged) capacitor before the power source is switched off. And the capacitor starts releasing energy to the circuit. An RC circuit is shown in Fig. 7-3. Summarizing, the natural response of an RC circuit is calculated by:

- (1) Finding the initial voltage  $V_o$  across the capacitor,
- (2) Finding the time constant of the circuit (Eq. 7-6), and
- (3) Using Eq. 7-4 to generate  $v(t)$ .

### 2.1.3 Parallel RLC circuit:

The current in the circuit consisting of a resistor, a capacitor and an inductor connected in series is given by:

$$i(t) = A_1 e^{s_1 t} + A_2 e^{s_2 t}, \quad s_{1,2} = -\alpha \pm \sqrt{\alpha^2 - \omega_0^2} \quad \text{for } \alpha > \omega_0 \quad \text{Over damped}$$

$$i(t) = (A_1 t + A_2) e^{-\alpha t}, \quad \text{for } \alpha = \omega_0 \quad \text{Critical damped}$$

$$i(t) = (A_1 \cos \omega_d t + A_2 \sin \omega_d t) e^{-\alpha t}, \quad \text{for } \alpha < \omega_0 \quad \text{Under damped}$$

, and  $\omega_0 = 1/\sqrt{LC}$ , the resonant frequency  $\alpha = 1/2RC$  Where the damping coefficient are  $A_2$  and  $A_1$ , and the constants  $\omega d = \sqrt{\omega_0^2 - \alpha^2}$  the natural resonant frequency determined by the initial conditions.

### 3 Procedure

#### 3.1 Step response of RL Circuits:

a. DC RL Circuit:

1. Assemble the circuit in Figure 7.1 with the component values shown in Table 7-1. By using Ohmmeter measure the internal resistance of the Inductor.

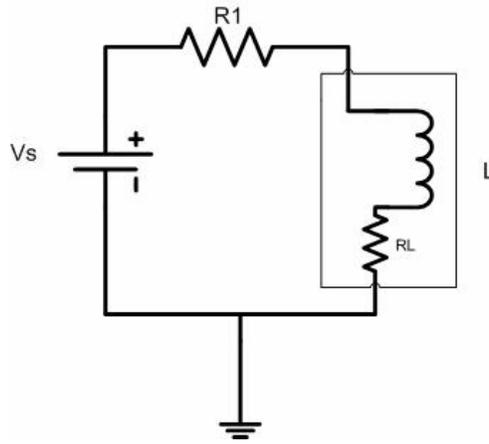


Fig.7.1 DC RL circuit

2. Take measurements to complete the entries corresponding to the Experimental values. In the same table.

Parameter	Unit	Theor.	Exp.	%Error
$R_1$	K $\Omega$	1.2		
$R_L$ (internal resistance of inductor)	$\Omega$			
L	mH	47		
$V_s$	V	10		
$V_{R1}$				
$V_L$				
$I_L$	mA			

Table 7-1

b. Transient RL circuit:

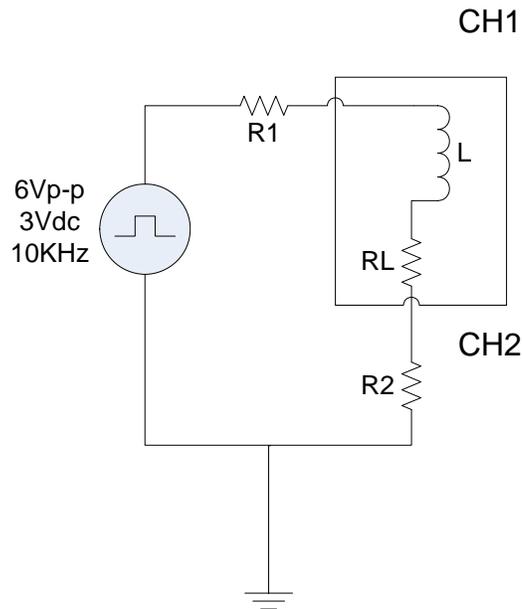


Fig.7.2 Transient RL circuit

1. Assemble the circuit in Figure 7.2 with the component values shown in Table 7-2. Measure the internal resistance of the inductor.
2. Set FG to supply a square wave with 6  $V_{p-p}$  amplitude and 10 KHz frequency. Add 3V DC offset. Check the amplitude of the signal after the connection to the circuit.
3. Connect CH1 and CH2 as illustrated in the graph, notice that there is a small resistance  $R_2$  connected in series with the inductor. This is because the oscilloscope can measure voltage only, therefore, its necessary to measure scaled version of  $i_C(t)$ . Divide the voltage by  $R$  to obtain the current in the circuit,. So **CH2 represent  $i_L(t)$** .
4. Observe the waveforms of the inductor voltage and current, and **plot** them on a graph paper.
5. Calculate the time constant for this circuit and find the voltage and current at  $t=\tau$ , and  $t=3\tau$ . **Do not forget to include the inductor internal resistance in your calculations**

Parameter	Unit	Theor.	Exp.	%Error
$R_1$	$K\Omega$	1.0		
$R_2$	$\Omega$	10		
$R_{int}$				
$L$	mH	10		
$\tau$	s			

Table 7-2 Results of the transient RL circuit shown in Fig. 7.2

### 3.2 RC circuit:

a. DC RC circuit:

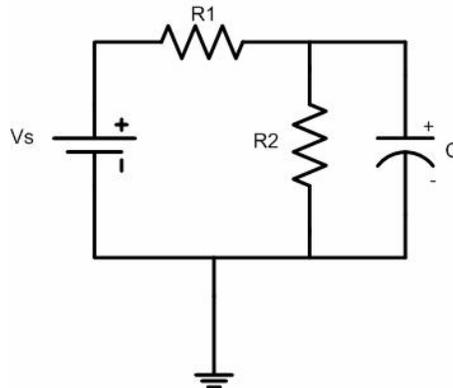


Fig 7-3

1. Assemble the circuit in Figure 7-3 with the component values shown in Table 7-3.
2. Take measurements to complete the entries corresponding to the Experimental values. in the same table.
3. After taking measurements, discharge the capacitor by disconnecting The power supply, observe that by connecting the voltmeter a cross the capacitor.

Parameter	Unit	Theor.	Exp.	%Error
R1	KΩ	1.2		
R2		1.0		
C	μF	100		
Vs	V	10		
VR1				
VR2				
VC				

Table 7.3

b. Transient RC circuit

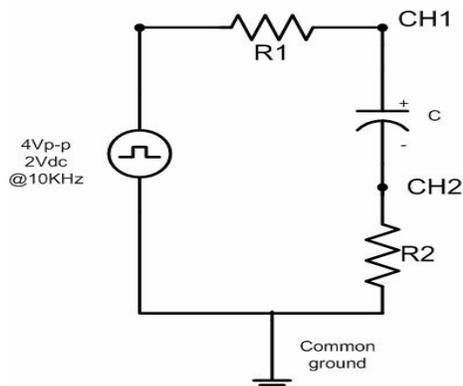


Fig.7-4

1. Assemble the circuit in Figure 7-4 with the component values shown in Table 7-4.
2. Set FG to supply a square wave with 4 V<sub>PK-PK</sub> amplitude and 10KHz frequency. Add 2V DC offset. Check the amplitude of the signal after the connection to the circuit.
3. Connect CH1 and CH2 as illustrated on the graph, notice that there is a small resistance R2 connected in series with the capacitor. This is because the oscilloscope can measure voltage only, therefore, its necessary to measure scaled version of  $i_c(t)$ . Divide the voltage by R to obtain the current in the circuit,. So **CH2 represent  $i_c(t)$** .
4. Observe the waveforms of voltage and current, and **plot** them on a graph paper.
5. Calculate the time constant for this circuit and compare with the voltage and current at  $t=\tau$ , and  $t=3 \tau$ .

Parameter	Unit	Theor.	Exp.	%Error
R1	K $\Omega$	1.0		
R2	$\Omega$	10		
C	nF	10		
$\tau$	s			

Table 7.4

### 3.3 Parallel RLC circuit:

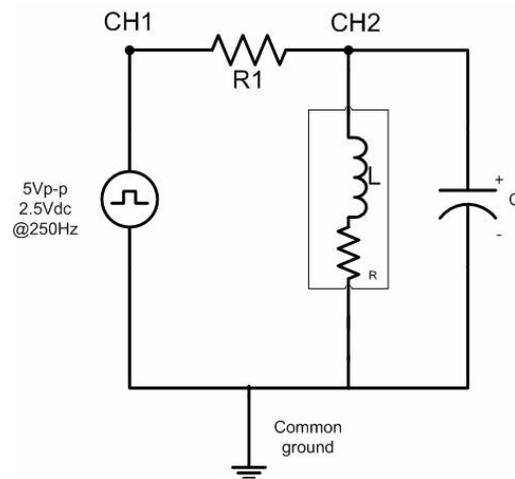


Fig.7-5

1. Assemble the circuit in Figure 7-5 with the component values shown in Table 7-5. Measure the internal resistance of the inductor.

2. Set FG to supply a square wave with  $5V_{PK-PK}$  amplitude and 250Hz frequency. Add 2.5V DC offset. Check the amplitude of the signal after the connection to the circuit.
3. Connect CH1 and CH2 as illustrated on the graph, CH2 **represent V(t)**.
4. Observe the waveform of voltage, and **plot** it on a graph paper.
5. Determine  $\alpha$ ,  $\omega_0$ , the type of damping of this circuit and find the expression for the voltage V(t). **Do not forget to include the inductor internal resistance in your calculations.**

Parameter	Unit	Theor.	Exp.	%Error
R1	K $\Omega$	1.0		
Rint	$\Omega$			
L	mH	1.0		
C	$\mu F$	0.1		

Table 7-5