Multisim v9 Digital Primer

to accompany

Electronics
Principle and Applications
Seventh Edition
by
Charles A. Schuler

McGraw Hill **Higher Education**

Contents

Introduction

In an effort to help the reader understand the concepts presented in this text book, key examples and problems will be presented through the use of computer simulation using MultiSim. MultiSim is an interactive circuit simulation package that allows the student to view their circuit in schematic form while measuring the different parameters of the circuit. The ability to create a schematic quickly and then analyze the circuit through simulation makes MultiSim a wonderful tool to help students understand the concepts covered in the study of electronics.

This appendix will introduce the reader to the features of MultiSim that directly relate to the study of DC, AC, and semiconductor electronics. The topics covered are:

- Work Area
- Opening a File
- Running a Simulation
- Saving a File
- Components
- Sources
- Measurement Equipment
- Circuit Examples

Work Area

The power of this software lies in its simplicity. With just a few steps, a circuit can be either retrieved from disk or drawn from scratch and simulated. The main screen, as shown in Figure 1, is divided into three areas: The drop down menu, the tool bars, and the work area.
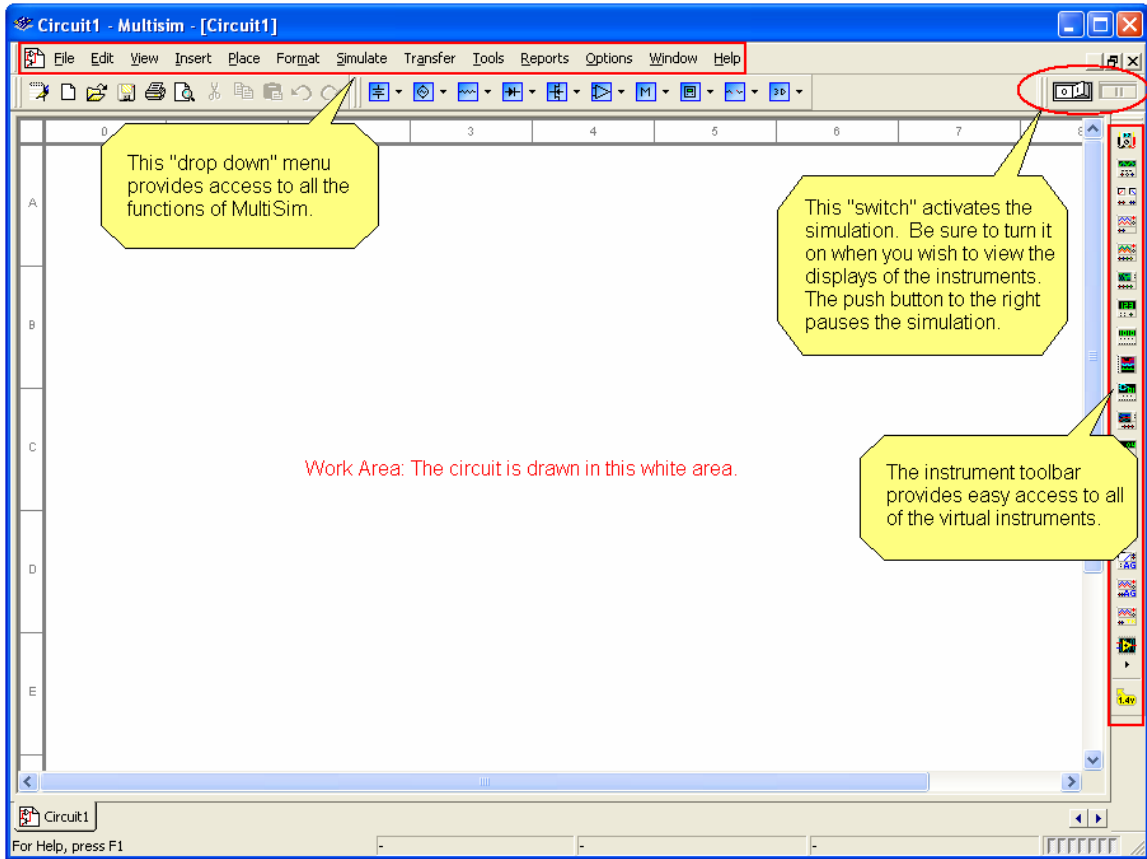

Figure 1 Main Screen

The drop down menu gives the user access to all the functions of the program. Initially, the user will only utilize a few of the different menu selections. Each of the drop down menu main topics can be accessed by either a mouse click or by pressing the <Alt> key and the underlined letter. For example, to access the File menu simply press <Alt><F> at the same time. The File menu will drop down as shown in Figure 2.
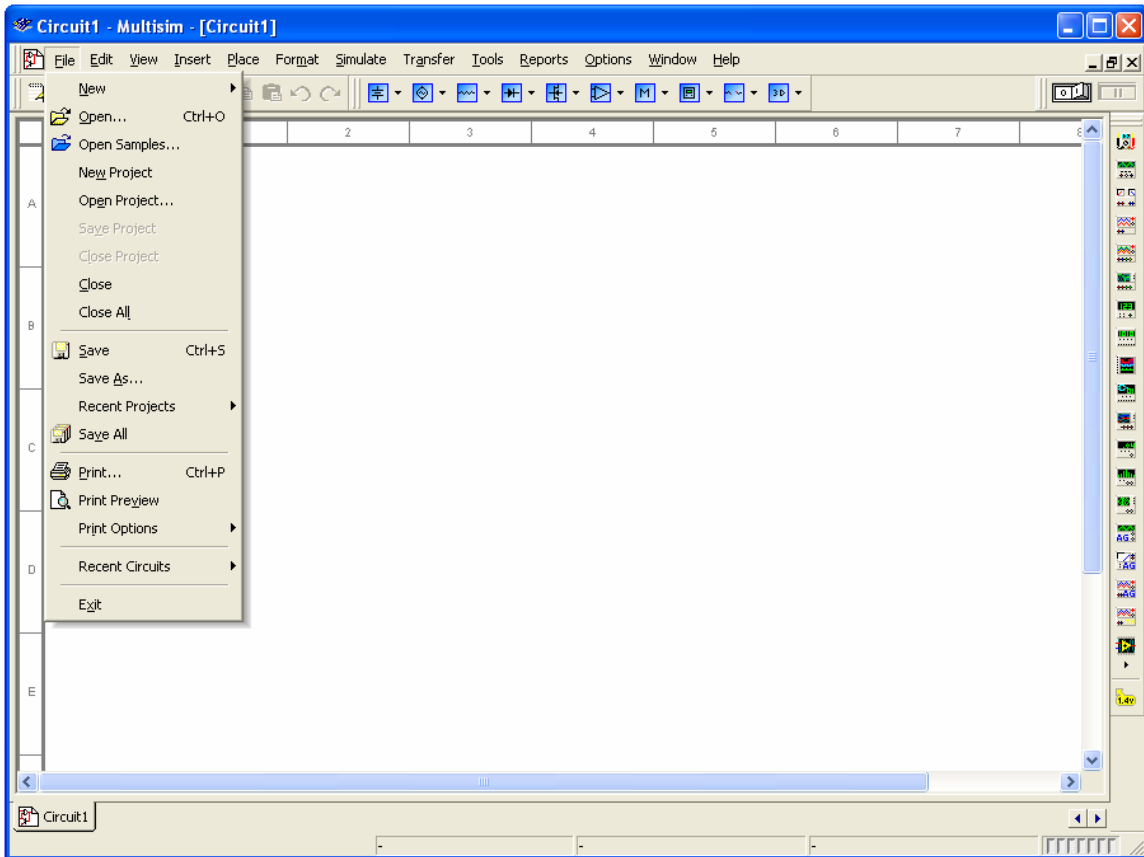

Figure 2 File Drop Down Menu

Initially, there are only two selections from the drop down menu that need be mastered: Opening a file and saving a file. The rest of the menu options can be explored as time permits.

The tool bars beneath the drop down menu provide access to all of the menu selections. Typically, a user will access them through the tool bars instead of the drop down menus. The most important icon in the assorted tool bars is the on-off switch. The on-off switch starts and stops the simulation. The push button next to the on-off switch will pause the simulation. Pressing the Pause button while the simulation is running allows the viewing of a waveform or meter reading without the display changing.

Opening a File

The circuits referenced in this textbook are included on a compact disk located in the back of this textbook. The files are divided into folders, one for each chapter. The name of the file provides a wealth of information to the user.

Example: A typical file name would be "Ch 4 Problems 4-11." This first part of the file name tells the user that the file is located in the folder labeled "Chapter 4." The second part of the file name tells the user that it is question 11 out of the Problems section at the end of Chapter 4.

To open a file, either mouse click on the word "File" located on the drop down menu bar and then mouse click on the Open command or mouse click on the open folder icon located on the tool bar as shown in Figure 3. Both methods will cause the Open File dialog box shown in Figure 4 to pop open. Navigate to the appropriate chapter folder and retrieve the file needed.
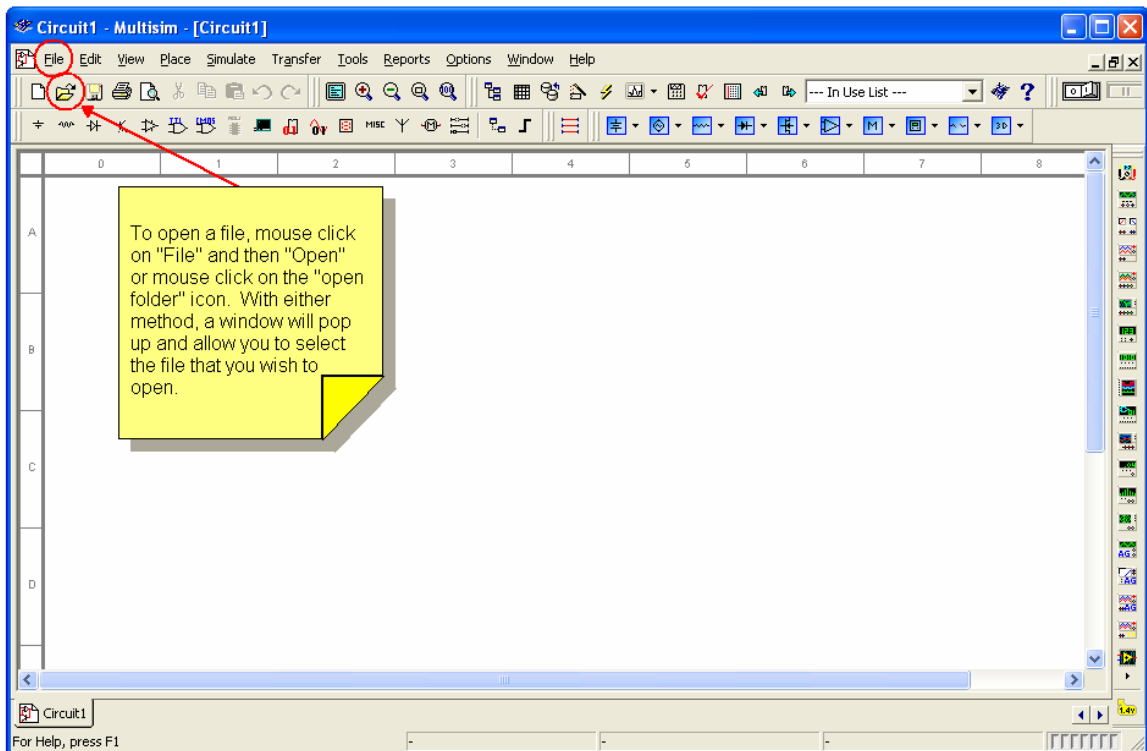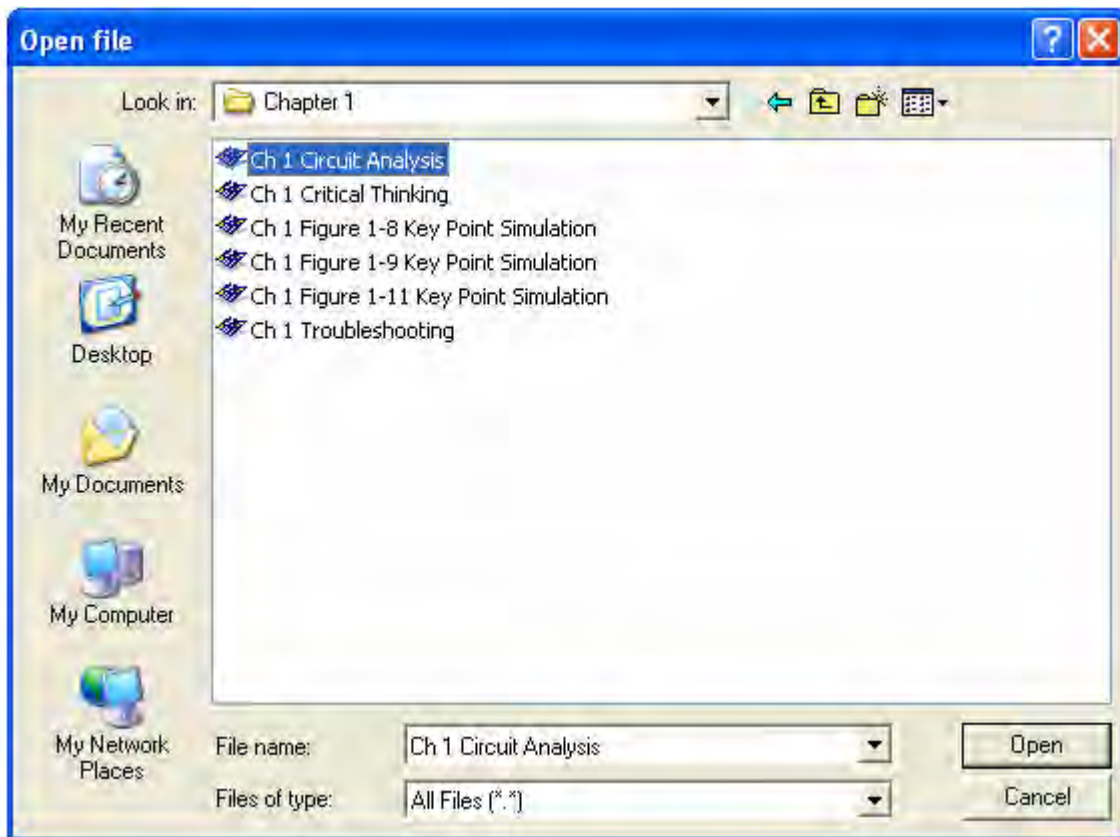

Figure 3 Opening a File

Figure 4 Open file Dialog Box

Running a Simulation

The MultiSim files developed for this textbook present the circuit in a standard format. The instrumentation is typically connected to the appropriate places within the circuit. If the instrument's display is not visible, double mouse click on the instrument icon and the display will pop up. There is a help screen referred to as the "Description Box" with helpful information relating to the circuit and the instruments within the circuit. This Description Box is opened by pressing <Control><D> while in MultiSim.

The simulation can be started three ways:
1. Selecting "Simulate" from the drop down menu and then selecting "Run."
2. Pressing the <F5> key.
3. Pressing the toggle switch with a mouse click.
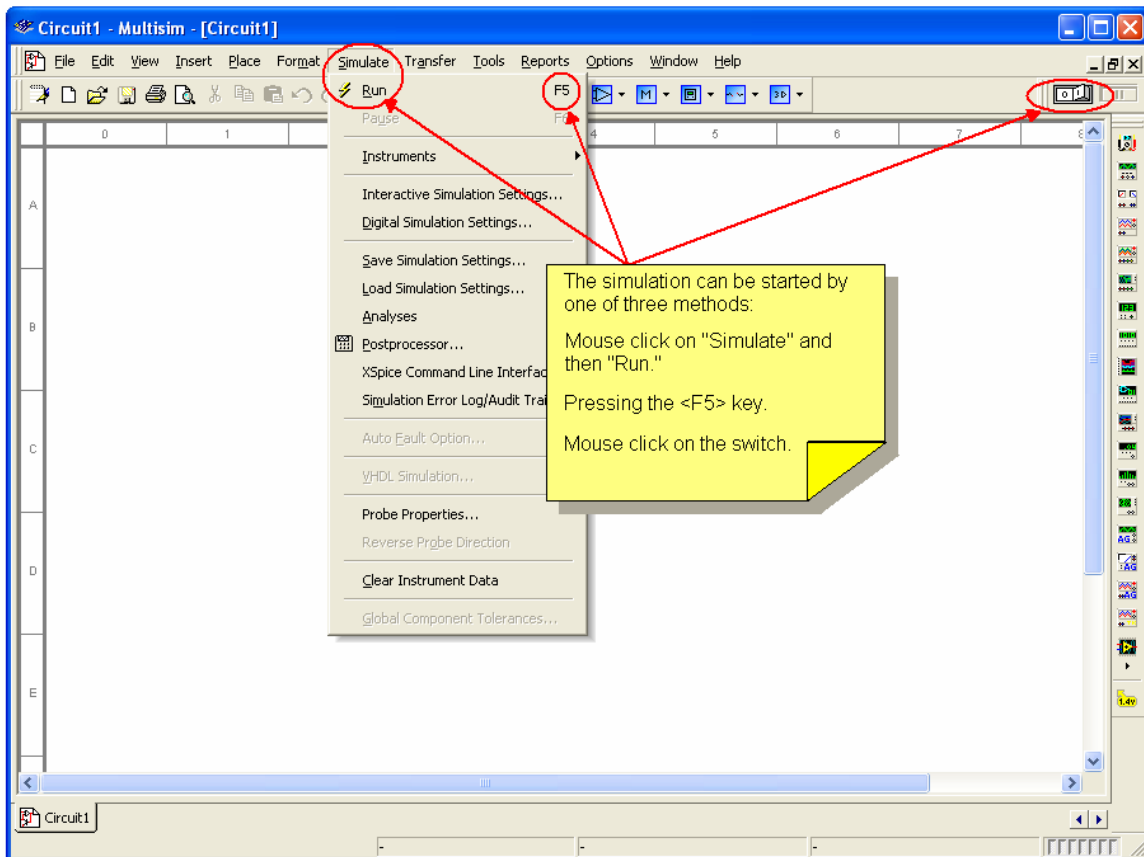
All three of these ways are illustrated in Figure 5.

Figure 5 Starting the Simulation

Saving a File

If the file has been modified, it needs to be saved under a new file name. As shown in Figure 6, select "File" with a mouse click located on the drop down menu bar. Then select "Save As" from the drop down menu. This will cause the "Save As" dialog box to open. Give the file a new name and "press" the save button with a mouse click. The process is demonstrated in Figure 7.
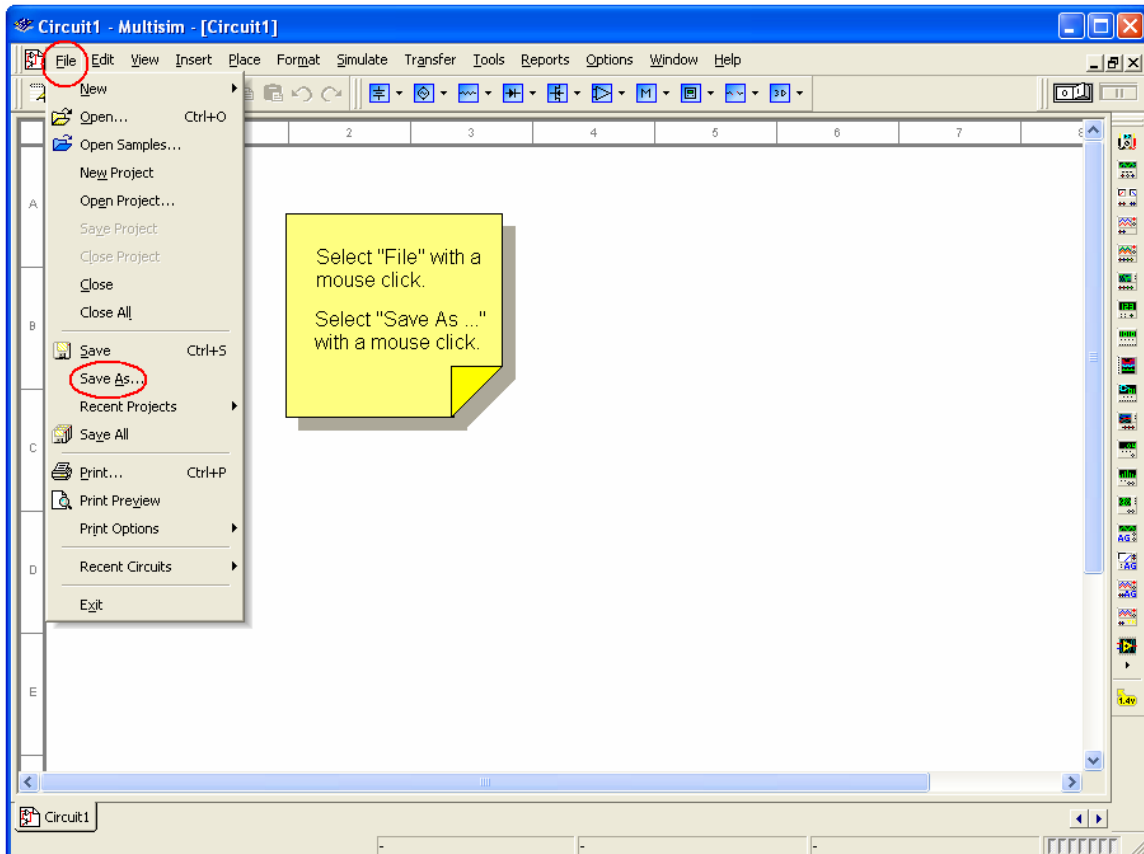

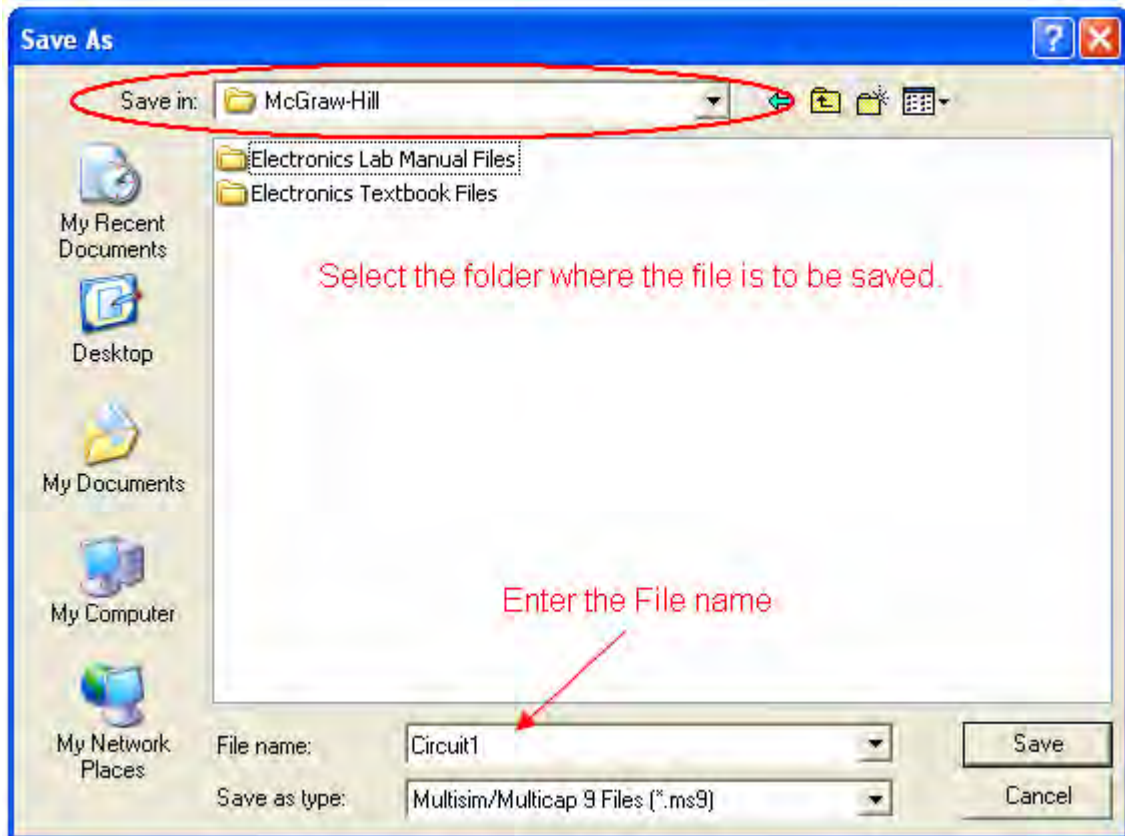**Figure 6 "Save As. . ." Screen**

Figure 7 "Save As . . . " Dialog Box

Components

There are two kinds of component models used in MultiSim: Those modeled after actual components and those modeled after "ideal" components. Those modeled after ideal components are referred to as "virtual" components. There is a broad selection of virtual components available, as shown in Figure 8.
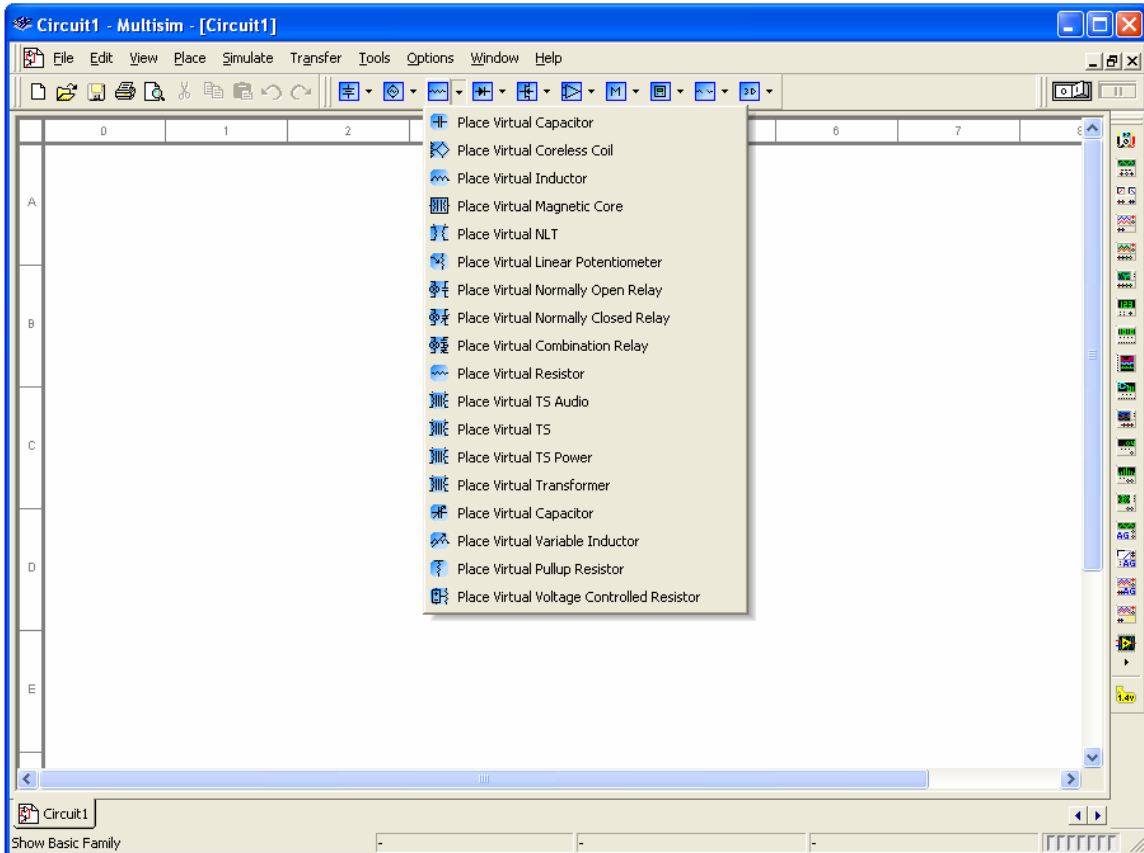

Figure 8 Virtual Component List

The difference between the two types of components resides in their rated values. The virtual components can have any of their parameters varied, whereas those modeled after actual components are limited to real world values. For example, a virtual resistor can have any value resistance and percent tolerance as shown in Figure 9.
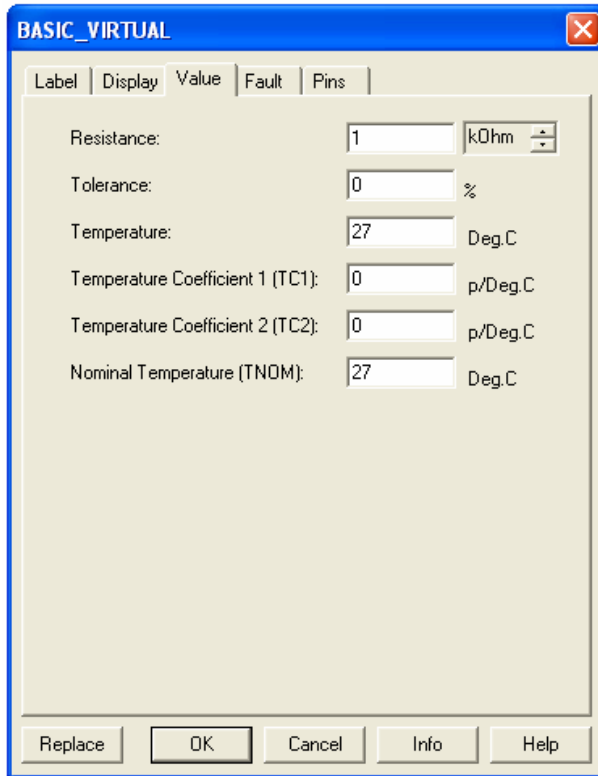
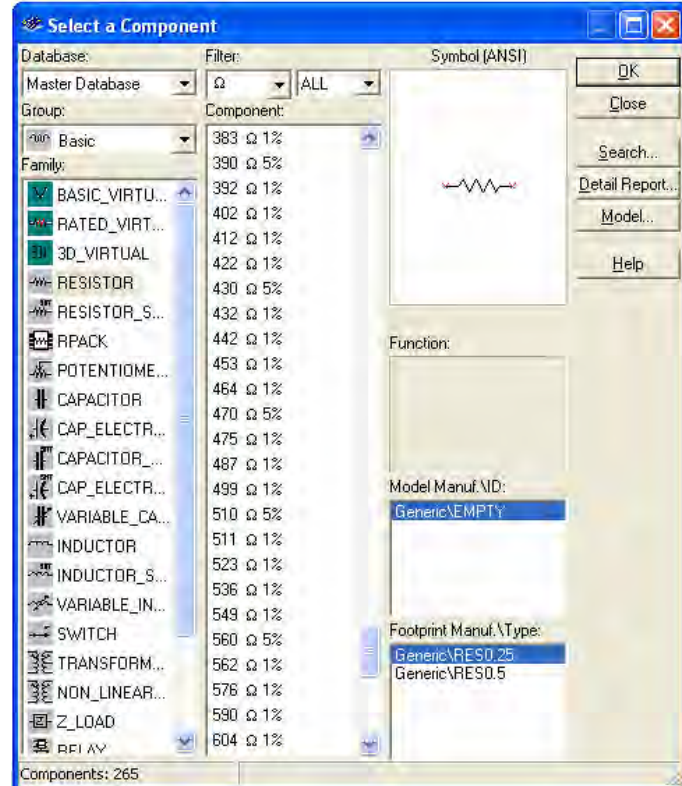**Figure 9 Configuration Screen for a Virtual Resistor**   **Figure 10 Component Listing for Resistors**

The models of the actual resistors are limited to standard values with either a 1% or 5% tolerance. The same is true for all other components modeled after real components, as shown in Figure 10. This is especially important when the semiconductor devices are used in a simulation. Each of the models of actual semiconductors will function in accordance with their data sheets. These components will be listed by their actual device number as identified by the manufacturers. For example, a common diode is the 1N4001. This diode, along with many others, can be found in the semiconductor library of actual components. The parameters of the actual component libraries can also be modified, but that requires an extensive understanding of component modeling and is beyond the scope of this appendix.

If actual components are selected for a circuit to be simulated, the measured value may differ slightly from the calculated values as the software will utilize the tolerances to vary the results. If precise results are required, the virtual components can be set to specific values with a zero percent tolerance.

Two of the components require interaction with the user. The switch is probably the most commonly used of these two devices. The movement of the switch is triggered by pressing the key associated with each switch as shown in Figure 11. The key is selected while in the switch configuration screen, Figure 12. If two switches are assigned the same key, they both will move when the key is pressed.
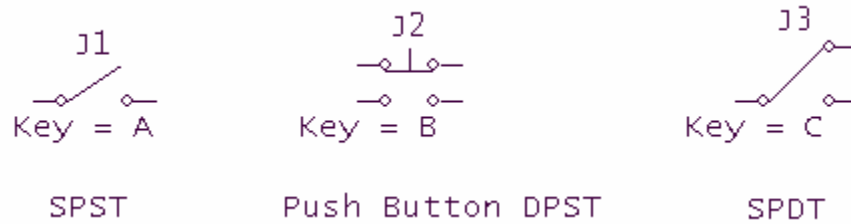
Figure 11 Switches

Figure 12 Switch Configuration Screen

The second component that requires interaction with the user is the potentiometer. The potentiometer will vary its resistance in predetermined steps with each key press. The pressing of the associated letter on the keyboard will increase the resistance and the pressing of the <Shift> key and the letter will decrease the resistance. As shown in Figure 13, the percent of the total resistance is displayed next to the potentiometer. The incremental increase or decrease of resistance is set by the user in the configuration screen. The associated key is also set in the configuration screen as shown in Figure 14.
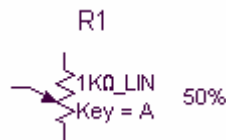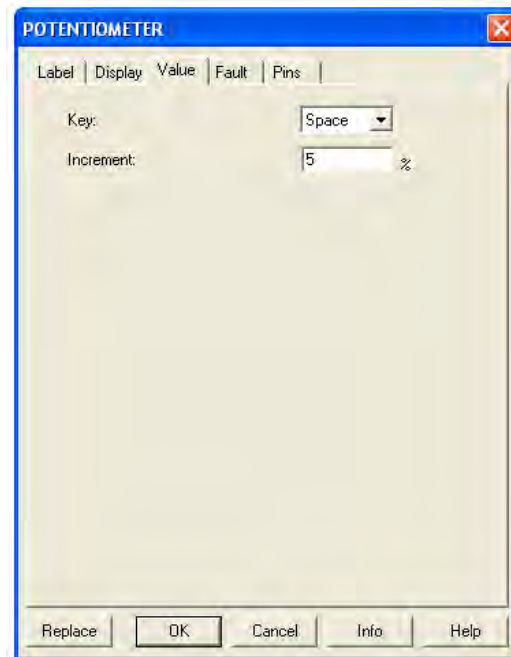

Figure 13 Potentiometer

Figure 14 Potentiometer Configuration Screen

Sources

In the study of DC and AC electronics, the majority of the circuits include either a voltage or current source. There are two main types of voltage sources: DC and AC sources. The DC source can be represented two ways: As a battery in Figure 15 and as a voltage supply.
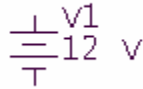

Figure 15 DC Source as a Battery

The voltage rating is fully adjustable. The default value is 12 VDC. If the component is double clicked, the configuration screen shown in Figure 16 will pop up and the voltage value can be changed.
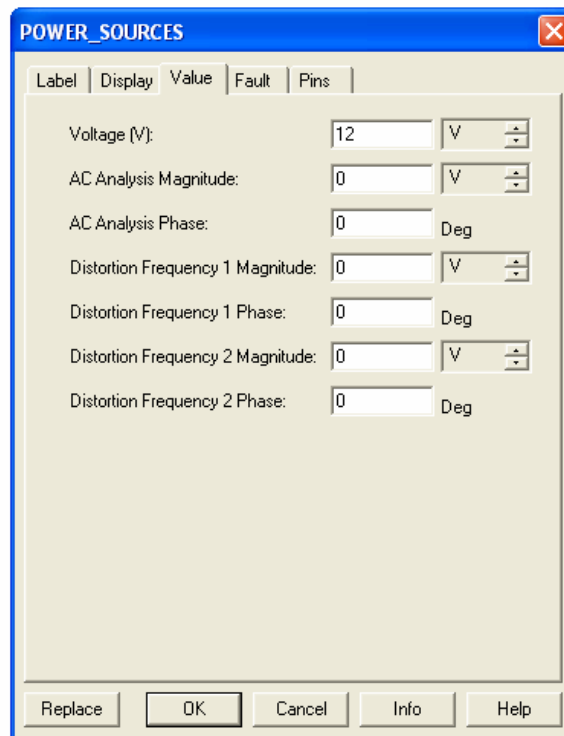

Figure 16 Configuration Screen for the DC Source

The voltage supplies are used in semiconductor circuits to represent either a positive or negative voltage supply. Figure 17 contains the V_{CC} voltage supply used in transistor circuits. FET circuits will utilize the V_{DD} voltage supply as illustrated in Figure 18.

VCC
5V

VDD
5V

Figure 17 V_{CC} Voltage Source   **Figure 18 V_{DD} Voltage Source**

Figure 19 depicts the $+V_{CC}$ and the $-V_{EE}$ voltage sources. These sources are found in operational amplifier circuits. Operational amplifiers typically have two voltage supplies: A negative (V_{EE}) and a positive (V_{CC}) voltage supply, Figure 20.

VCC
15V

+

−

−15V
VEE

VCC
5V

VEE
−5V

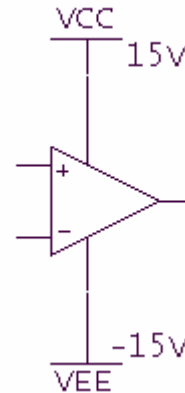**Figure 19 V_{CC} and V_{EE} Voltage Sources**   **Figure 20 V_{CC} and V_{EE} Op Amp Example**

The voltage rating is fully adjustable for all three voltage sources. The default value is +5 VDC for V_{CC} and V_{DD}. The default value for V_{EE} is -5 VDC. If the component is double clicked, the configuration screen shown in Figure 21 will pop up and the voltage value can be changed.

Figure 21 V_{CC} Configuration Screen

The AC source can be represented as either a schematic symbol or it can take the form of a function generator.   The schematic symbol as shown in Figure 22 will include information about the AC source. This information will include the device reference number, V_P value, frequency, and phase shift. These values are fully adjustable. The default values are shown in Figure 23. If the component is double clicked, the configuration screen will pop up and the value can be changed.

Figure 22 AC Source

Figure 23 Configuration Screen for an AC Source

MultiSim provides two function generators: The Generic model and the Agilent model. The Agilent model 33120A has the same functionality as the actual Agilent function generator.

The Generic function generator icon is shown in Figure 24, along with the configuration screen. The configuration screen is displayed when the function generator icon is doubled clicked. The Generic function generator can produce three types of waveforms: Sinusoidal wave, triangular wave, and a square wave. The frequency, duty cycle, amplitude, and DC offset are all fully adjustable.
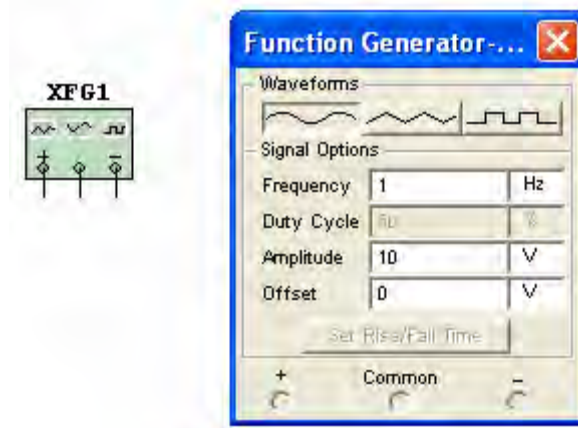
Figure 24 Generic Function Generator and Configuration Screen

The Agilent function generator is controlled via the front panel as shown in Figure 25. The buttons are "pushed" by a mouse click. The dial can be turned by dragging the mouse over it or by placing the cursor over it and spinning the wheel on the mouse. The latter is by far the preferred method.
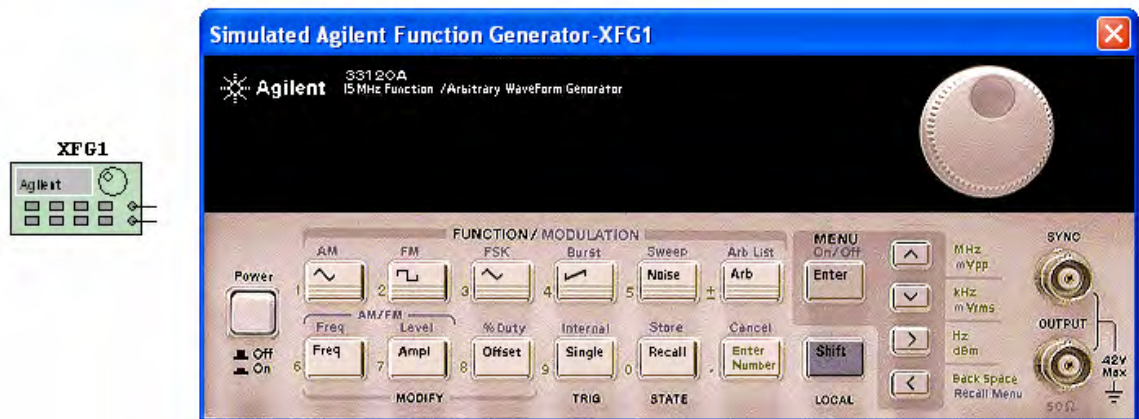
Figure 25 Agilent Function Generator

There are two types of current sources: DC and AC sources. The DC current source is represented as a circle with a downward pointing arrow in it. The arrow in Figure 26 represents the direction of current flow. The arrow can be pointed upward by rotating the symbol 180°.

Figure 26 DC Current Source

The current rating is fully adjustable. The default value is 1 A. If the component is double clicked, the configuration screen in Figure 27 will pop up and the current value can be changed.
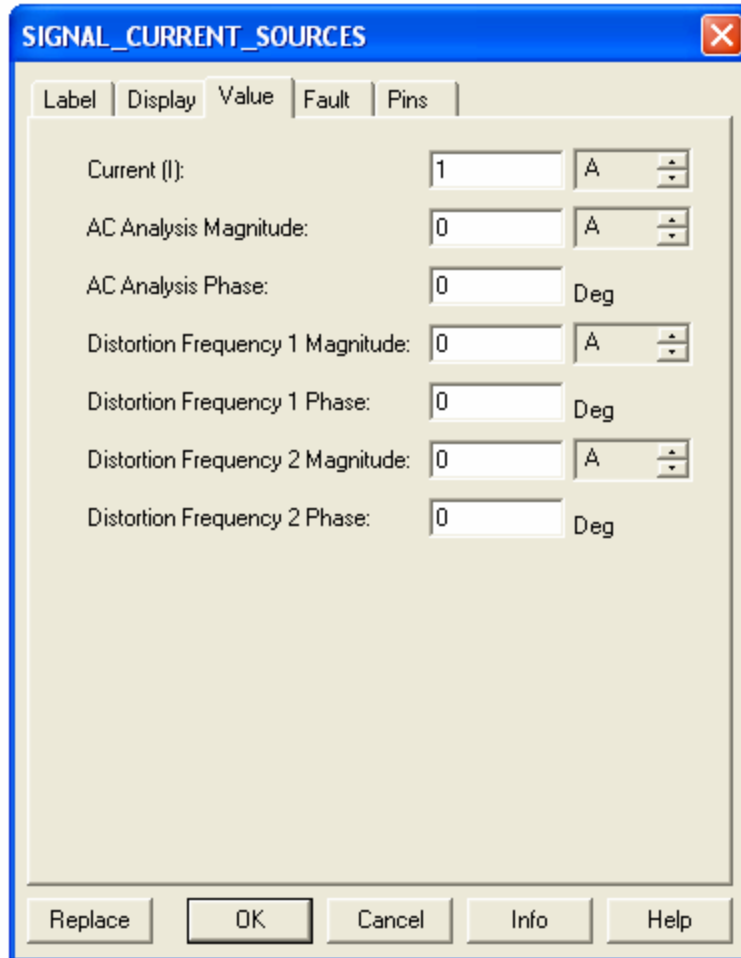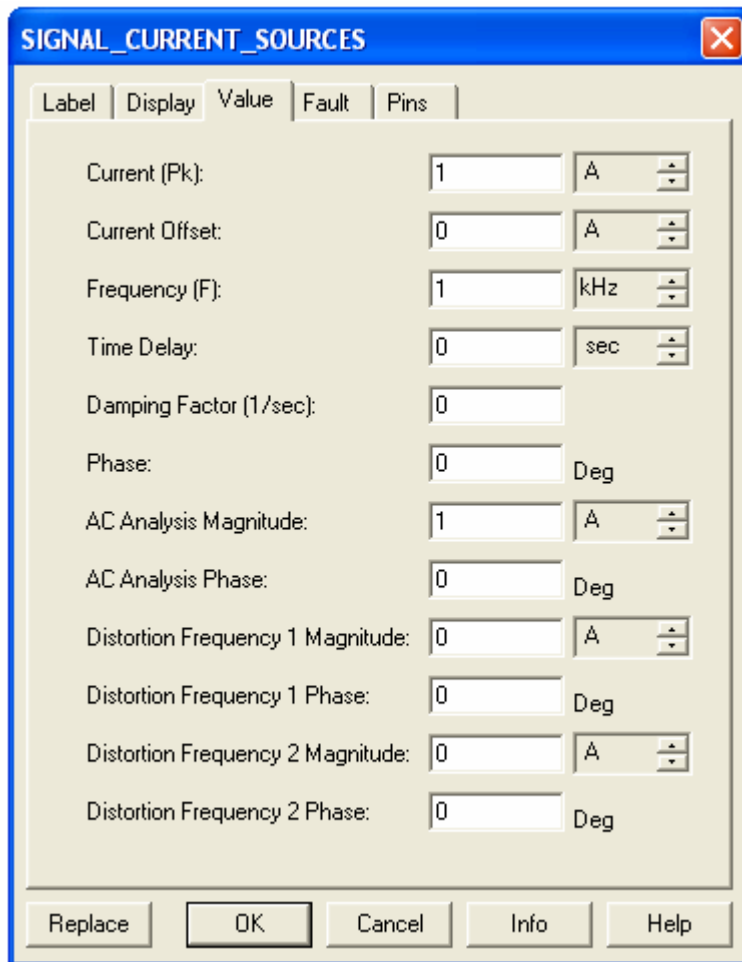

Figure 27 DC Current Source Configuration Screen

The AC current source is represented as a circle with a downward pointing arrow. There is a sine wave across the arrow. The schematic symbol in Figure 28 will include information about the AC current source. This information will include the device reference number, I_{Peak} value, frequency, and phase shift. These values are fully adjustable. The default values are shown in Figure 29. If the component is double clicked, the configuration screen will pop up and the values can be changed.

Figure 28 AC Current Source   **Figure 29 AC Current Source Configuration Screen**

MultiSim requires a ground to be present in the circuit in order for the simulation to function properly. The circuit must contain a ground and all instrumentation must have a ground connection. The schematic symbol for ground is shown in Figure 30.

Figure 30 Ground

Measurement Equipment

MultiSim provides a wide assortment of measurement equipment. In the study of DC, AC and semiconductor electronics, the three main pieces of measurement equipment are the digital Multimeter, the oscilloscope, and the Bode Plotter. The first two pieces of equipment are found in test labs across the world. The Bode Plotter is a virtual device that automates the task of plotting output voltage verses frequency. This is usually done by taking many measurements and plotting the results in a spreadsheet. The Bode Plotter performs this task for you.

Multimeters

There are two Multimeters to choose from: The Generic Multimeter and the Agilent Multimeter. The Generic Multimeter will measure current, voltage, resistance, and decibels. The meter can be used for both DC and AC measurements. The different functions of the meter are selected by mouse clicking on the icon to the left in Figure 31. The mouse click will cause the Multimeter display to pop up. The different functions on the display can be selected "pushing" the different buttons via a mouse click.

Figure 31 Generic Multimeter icon and Configuration Screen

The Agilent Multimeter icon and meter display are shown in Figure 32. The display is brought up by mouse clicking on the Agilent Multimeter icon. This Multimeter has the same functionality as the actual Agilent Multimeter. The different functions are accessed by "pushing" the buttons. This is accomplished by mouse clicking on the button. The input jacks on the right side of the meter display correspond to the five inputs on the icon. If something is connected to the icon, the associated jacks on the display will have a white "X" in them to show a connection.
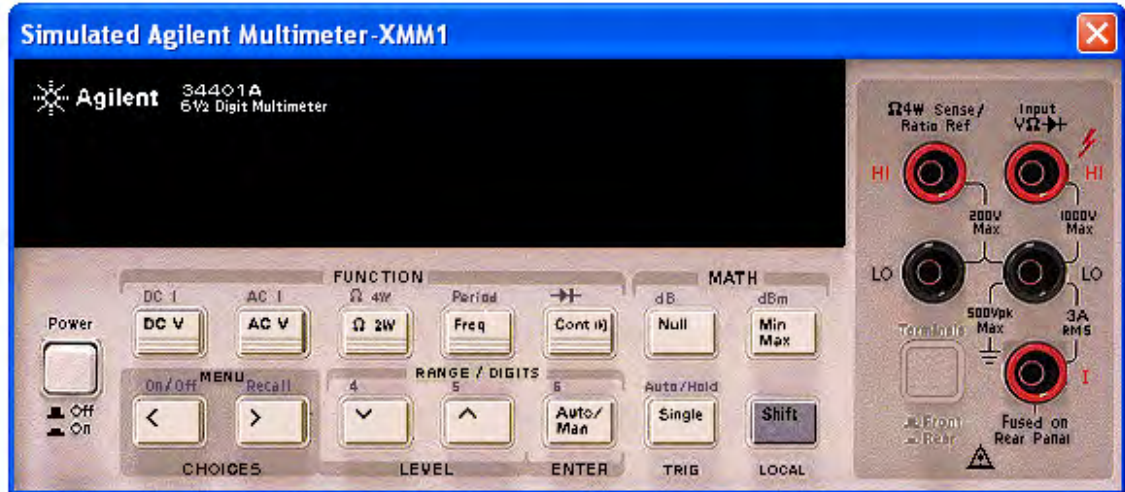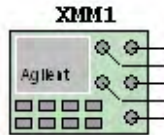
Figure 32 Agilent Multimeter Icon and Meter Display

Oscilloscopes

There are three oscilloscopes to choose from: The Generic oscilloscope, the Agilent oscilloscope, and the Tektronix oscilloscope. The Generic oscilloscope shown in Figure 33 is a dual channel oscilloscope. The oscilloscope display is brought up by mouse clicking on the oscilloscope icon. The settings can be changed by clicking in each box and bringing up the scroll arrows.
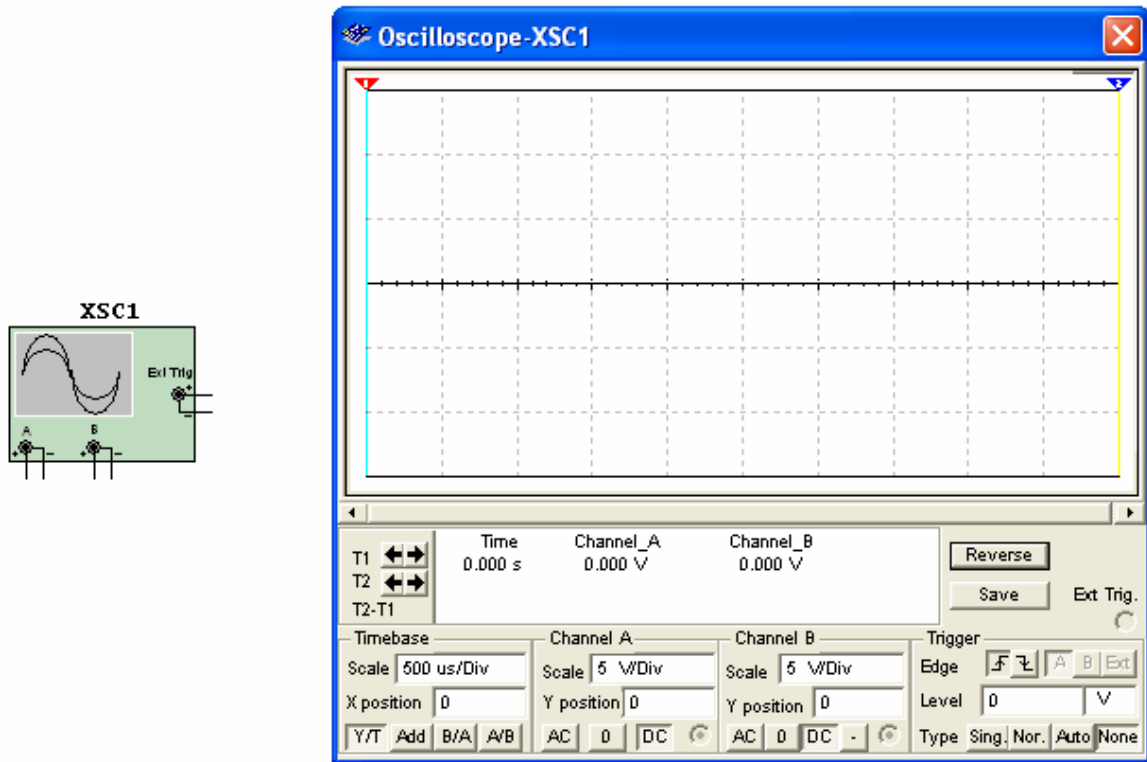

Figure 33 Generic Oscilloscope Icon and Oscilloscope Display

The Agilent oscilloscope icon in Figure 34 has all of the functionality of the Model 54622D dual channel oscilloscope. The Agilent oscilloscope is controlled via the front panel as shown in Figure 35. The buttons are "pushed" by a mouse click. The dials can be turned by dragging the mouse over it or by placing the cursor over it and spinning the wheel on the mouse. The latter is by far the preferred method.
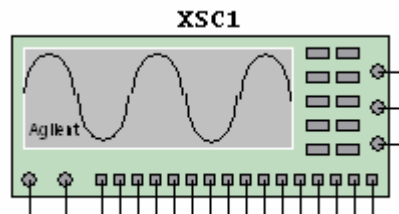
Figure 34 Agilent Oscilloscope Icon
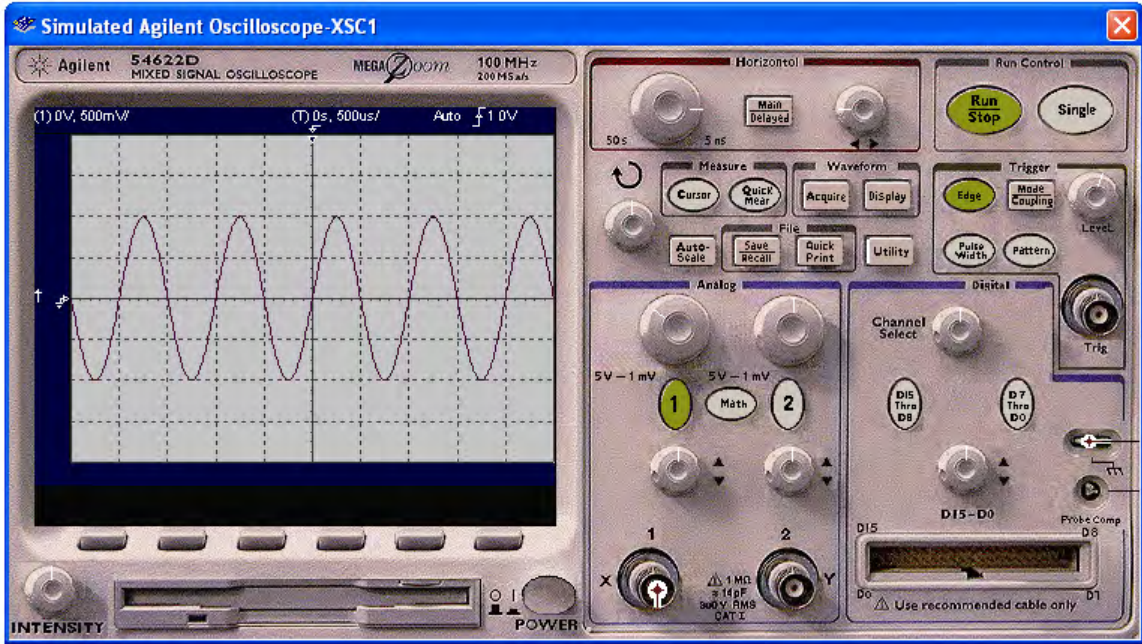
Figure 35 Agilent Oscilloscope Display

The Tektronix oscilloscope icon shown in Figure 36 has all of the functionality of the Model TDS2024 four channel Digital Storage oscilloscope. The color of the four channels is the same as the channel selection buttons on the display: Yellow, blue, purple, and green for channels one through four respectively. The Tektronix oscilloscope is controlled via the front panel as seen in Figure 37. The buttons are "pushed" by a mouse click. The dials can be turned by dragging the mouse over it or by placing the cursor over it and spinning the wheel on the mouse. The latter is by far the preferred method.
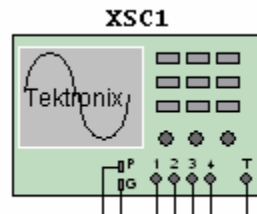
Figure 36 Tektronix Oscilloscope Icon

Figure 37 Tektronix Oscilloscope Display

Voltage and Current Meters

When voltage or current need to be measured, MultiSim provides very simple voltmeters and ammeters as shown in Figure 38. These meters can be placed throughout the circuit. The meters can be rotated to match the polarity needs of the circuit. The default is "DC." If the meters are to be used for AC measurement, then the configuration screen shown in Figure 39 must be opened and that parameter changed to reflect AC measurement. To open the configuration screen, double click on the meter.
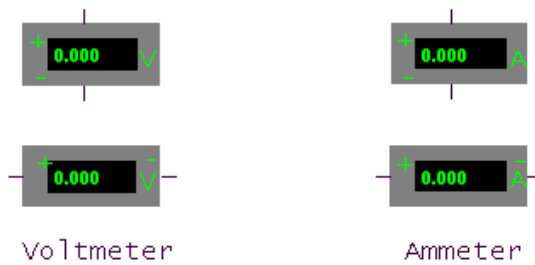
Figure 38 Voltage and Current Meters   **Figure 39 Voltmeter Configuration Screen**
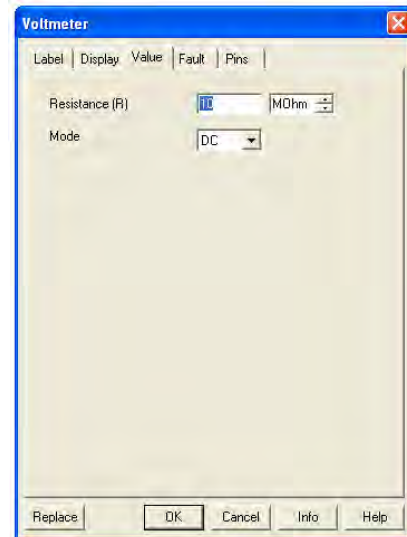
Bode Plotter

The Bode Plotter is used to view the frequency response of a circuit. In the actual lab setting, the circuit would be operated at a base frequency and the output of the circuit measured. The frequency would be incremented by a fixed amount and the measurement repeated. After operating the circuit at a sufficient number of incremental frequencies, the data would be graphed, with independent variable "frequency" on the X axis and dependant variable "amplitude" on the Y axis. This process can be very time consuming. MultiSim provides a simpler method of determining the frequency response of a circuit through the use of the virtual Bode Plotter.

In Figure 40, the positive terminal of the input is connected to the applied signal source. The positive terminal of the output is connected to the output voltage of the circuit. The other two terminals are connected to ground. The value of the AC source does not matter; the AC source just needs to be in the circuit. The Bode Plotter will provide the input signal.
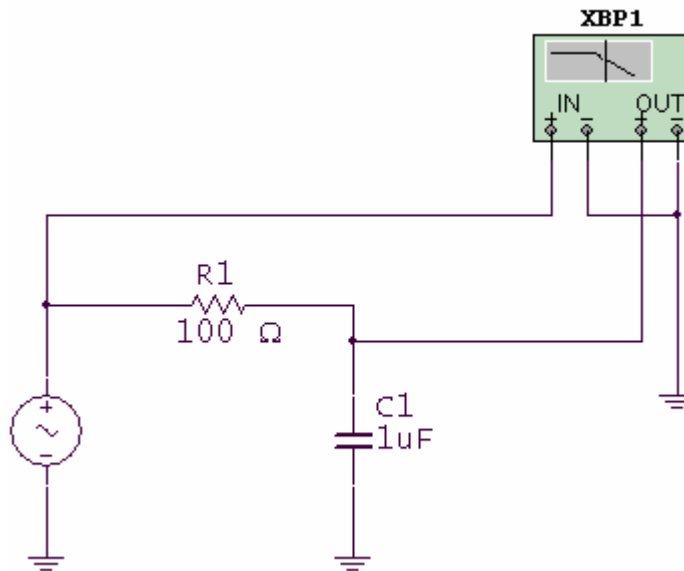
Figure 40 Bode Plotter
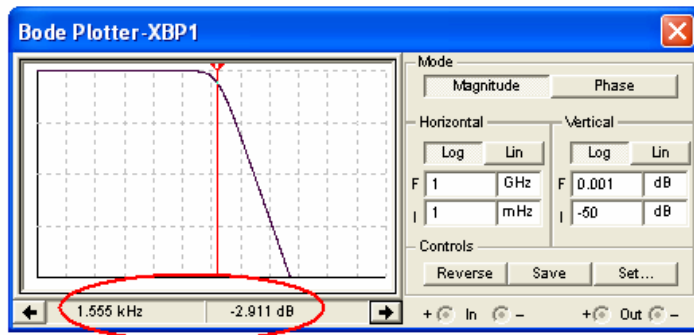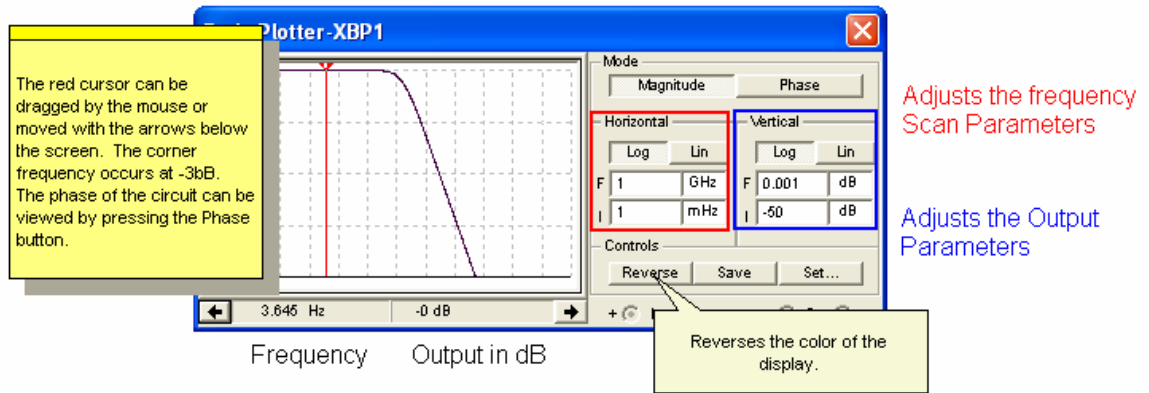
Figure 41 Bode Plotter Display

Circuit Examples

Example 1   Voltage measurement using a voltmeter in a series DC circuit.

A voltmeter in Figure 42 is placed in parallel with the resistor to measure the voltage across it. The default is set for "DC" measurement. If AC is required, double click on the meter to bring up the configuration screen. All circuits must have a ground. Figure 43 contains a *Quick Hint* on the use of the Voltmeter.
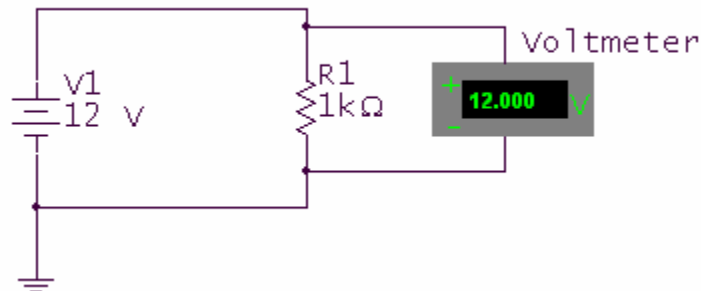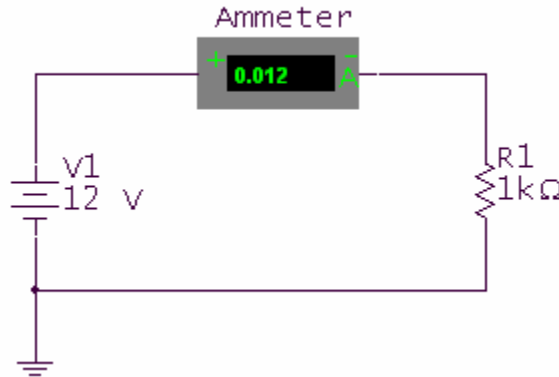

Figure 42 DC Voltage Measurement with a Voltmeter
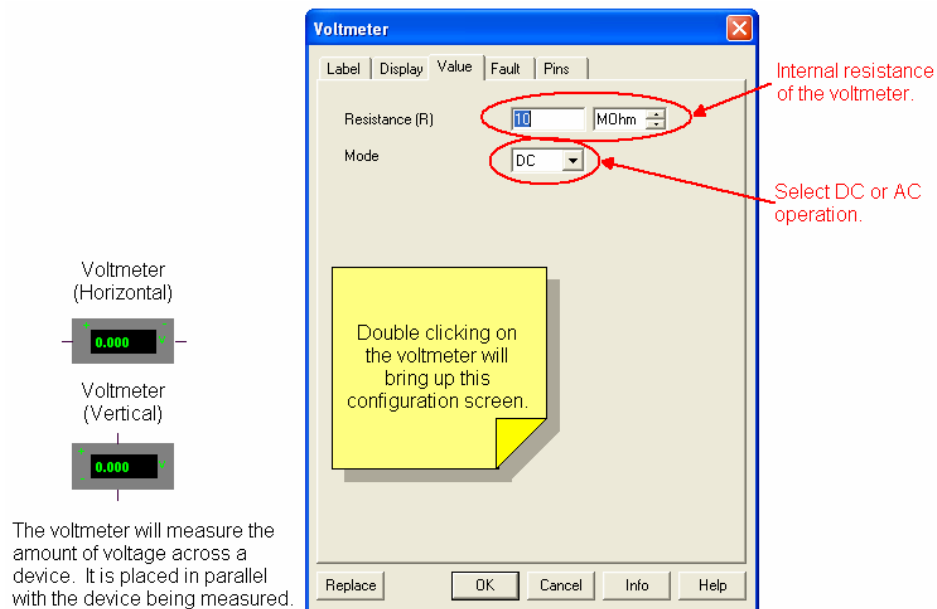
Figure 43 Voltmeter Quick Hint

Example 2 Voltage measurement using a Generic Multimeter in a series DC circuit.

A Generic Multimeter is placed in parallel with the resistor to measure the voltage across it. Be sure to double click the Generic Multimeter icon to bring up the meter display as shown in Figure 44. Press the appropriate buttons for "Voltage" and then "DC" or "AC" measurement. All circuits must have a ground. Figure 45 contains a *Quick Hint* on the use of the Generic Multimeter.
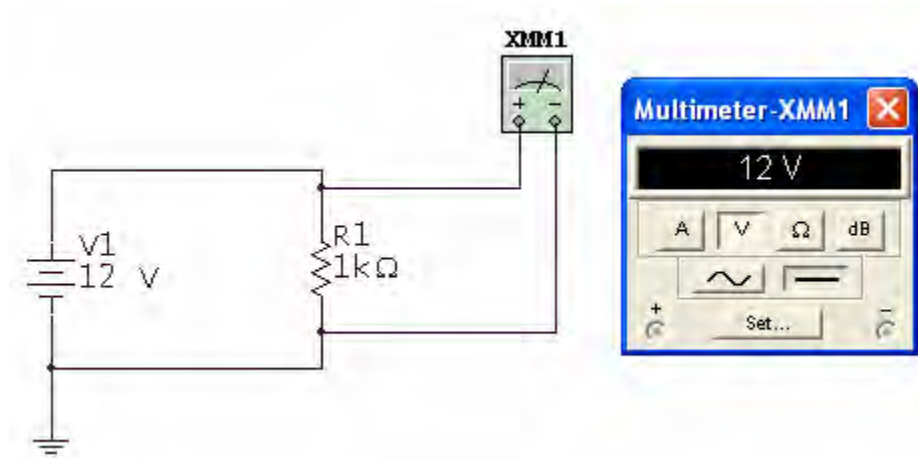
Figure 44 DC Voltage measurement with a Generic Multimeter
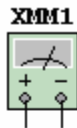
Figure 45 Generic Multimeter Quick Hint

Example 3 Voltage measurement using an Agilent Multimeter in a series DC circuit.

In Figure 46, an Agilent Multimeter is placed in parallel with the resistor to measure the voltage across it. Be sure to double click the Agilent Multimeter icon to bring up the meter display. Press the appropriate buttons for "Voltage" and then "DC" or "AC" measurement. All circuits must have a ground. Note the two white circles and black x's on the right side of the display to indicate a connection to the meter. Note: This instrument requires that its power button be pressed to "turn on" the meter. Figure 47 contains a *Quick Hint* on the use of the Agilent Multimeter.
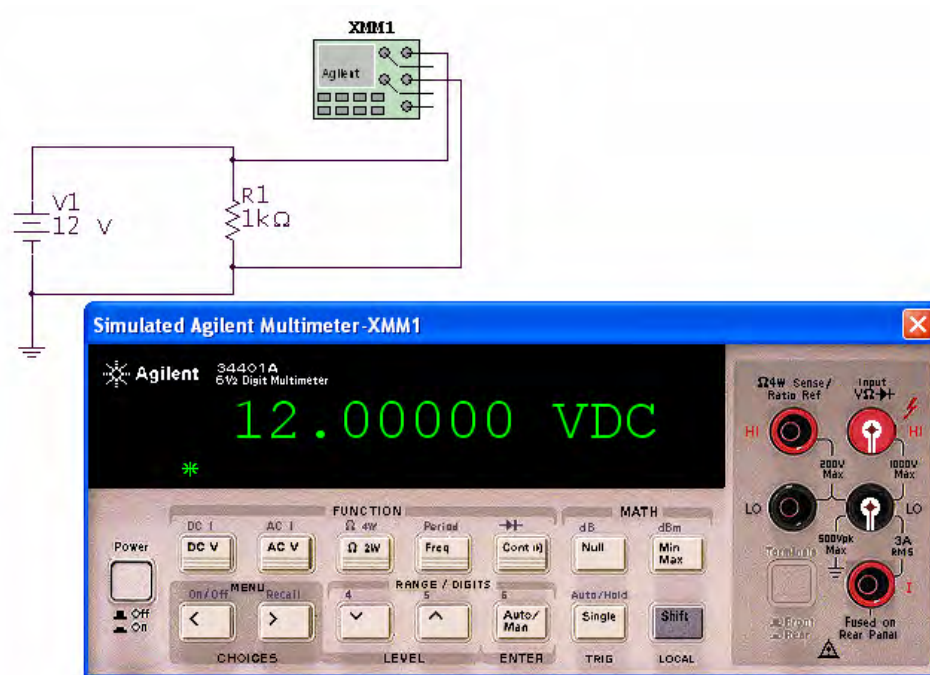
Figure 46 DC Voltage measurement with an Agilent Multimeter
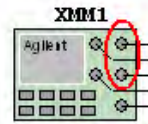
Figure 47 Agilent Multimeter Quick Hint

Example 4 Current measurement using an ammeter in a series DC circuit.

In Figure 48, an ammeter is placed in series with the resistor and DC source to measure the current flowing through the circuit. The default is set for "DC" measurement. If AC is required, double click on the meter to bring up the configuration screen. All circuits must have a ground. Figure 49 contains a *Quick Hint* on the use of the ammeter.

Figure 48 DC Current measurement with an Ammeter

Figure 49 Ammeter Quick Hint

Example 5 Current measurement using a Generic Multimeter in a series DC circuit.

In Figure 50, a Generic Multimeter is placed in series with the resistor and DC source to measure the current flowing through the circuit. Be sure to double click the Generic Multimeter icon to bring up the meter display. The current function is selected by clicking on the "A" on the meter display. Since the source is DC, the DC function of the meter is also selected, as indicated by the depressed button. All circuits must have a ground. Figure 51 contains a *Quick Hint* on the use of the Generic Multimeter.
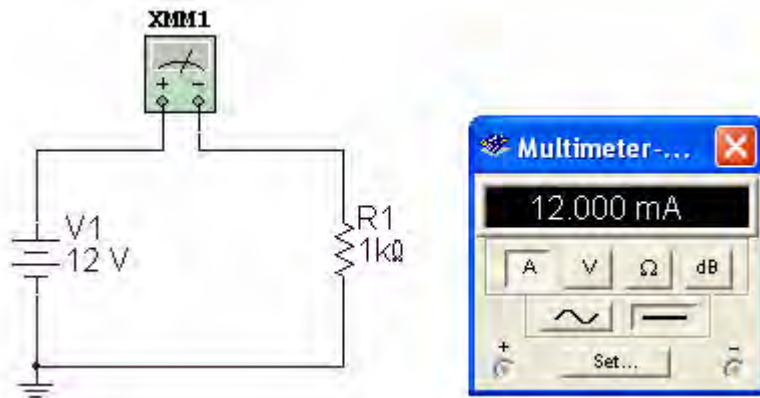
Figure 50 DC Current measurement with a Generic Multimeter

Figure 51 Generic Multimeter Quick Hint

Example 6 Current measurement using an Agilent Multimeter in a series DC circuit.

In Figure 52, an Agilent Multimeter is placed in series with the resistor and source to measure the current flowing through the circuit. Be sure to double click the Agilent Multimeter icon to bring up the meter display. Selection of DC current measurement is the second function of the DC voltage measurement button. Be sure to press the "Shift" button to access the second function of the voltage button. Note the two white circles and black x's on the right side of the display to indicate a connection to the meter. All circuits must have a ground.

This instrument requires that its power button be pressed to "turn on" the meter. Figure 53 contains a *Quick Hint* on the use of the Agilent Multimeter for current measurement.
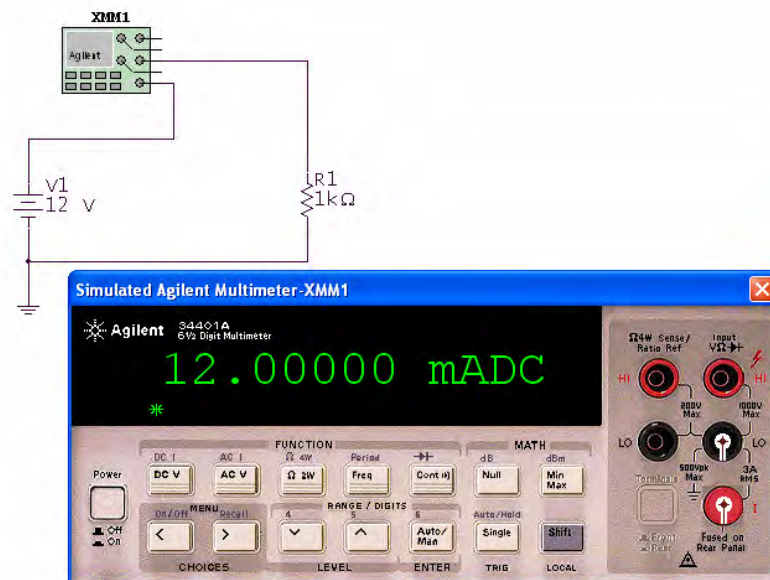
Figure 52 DC Current measurement with an Agilent Multimeter

Figure 53 Agilent Multimeter Quick Hint for Current Measurement

Example 7 Voltage measurement using a Generic Oscilloscope in a series AC circuit.

In Figure 54, channel 1 of the Generic oscilloscope is connected to the positive side of the resistor. The ground connection of the scope and the circuit must be grounded. The oscilloscope will use this ground as a reference point. This Generic oscilloscope's operation follows that of an actual oscilloscope. The oscilloscope display is brought up by mouse clicking on the oscilloscope icon. The settings can be changed by clicking in each box and bringing up the scroll arrows. Adjust the volts per division on the channel under measurement until the amplitude of the waveform fills the majority of the screen.   Adjust the Time-base such that a complete cycle or two are displayed. Figures 55, 56 and 57 contain *Quick Hints* on the use of the Generic oscilloscope.
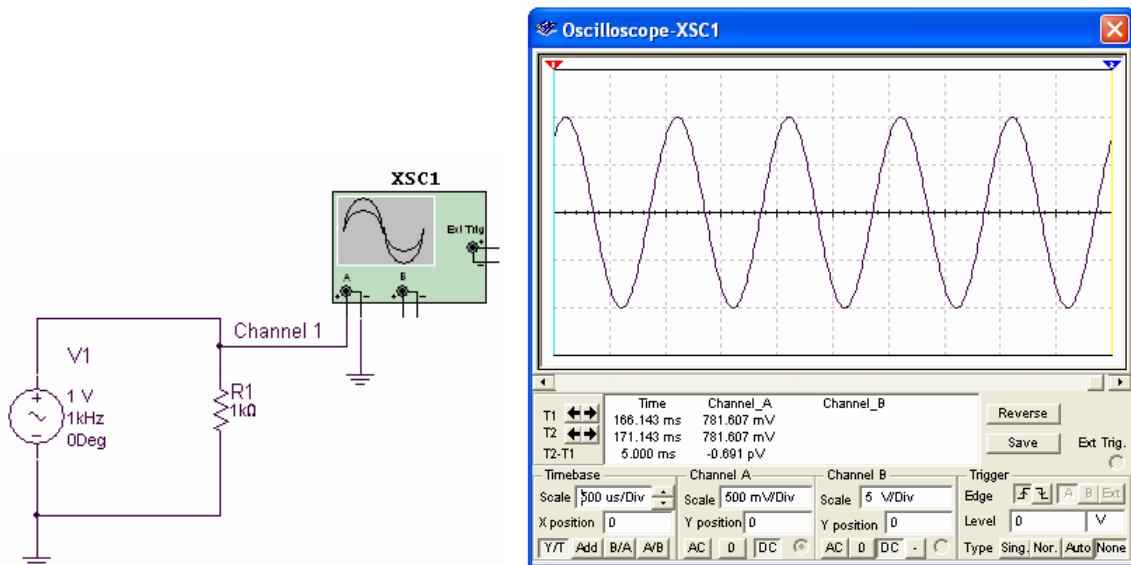
Figure 54 Voltage Measurement with a Generic Oscilloscope
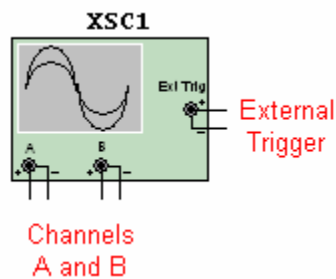
Figure 55 Generic Oscilloscope Icon Quick Hint
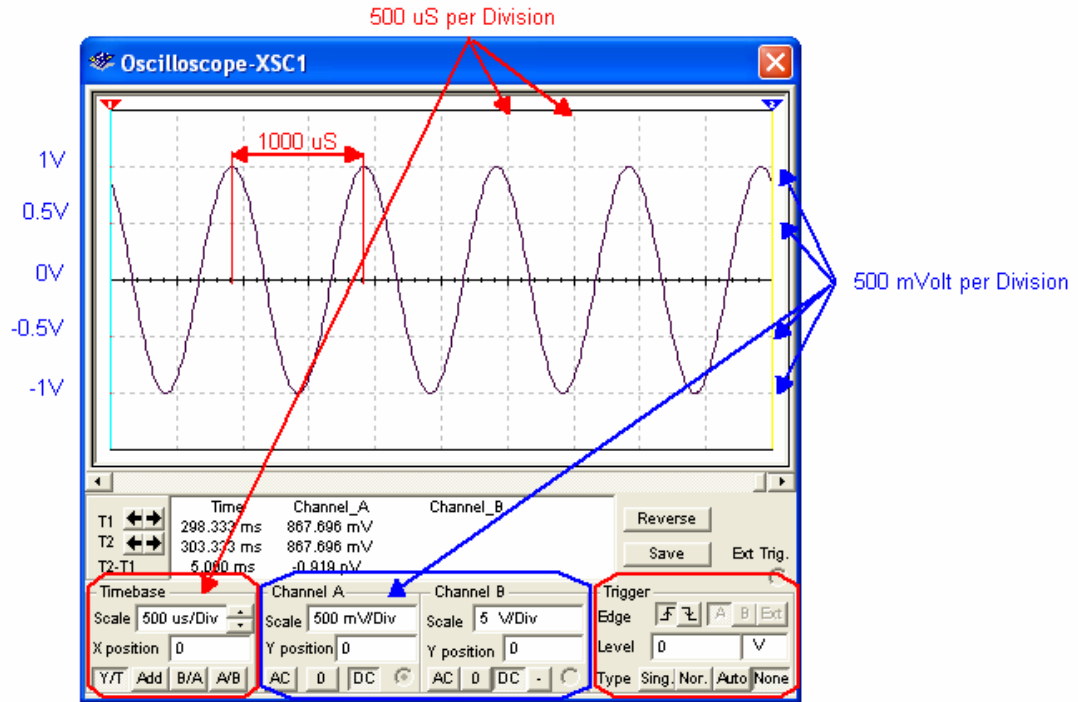
Figure 56 Generic Oscilloscope Quick Hint
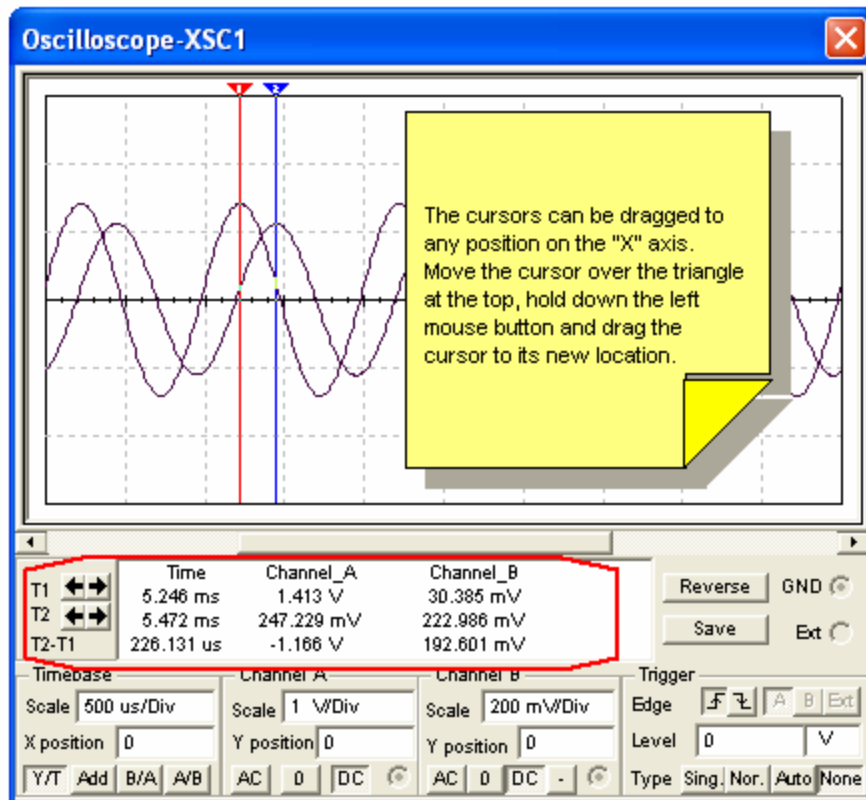
Figure 57 Generic Oscilloscope Quick Hint

Example 8 Voltage measurement using the Agilent Oscilloscope in a series AC circuit.

In Figure 58, channel 1 of the Agilent oscilloscope is connected to the positive side of the resistor. The ground connection of the scope and the circuit must be grounded. The oscilloscope will use this ground as a reference point. This Agilent oscilloscope's operation follows that of a dual channel, +16 logic channel, 100-MHz bandwidth Agilent Model 54622D Oscilloscope. The oscilloscope display as shown in Figure 59, is brought up by mouse clicking on the oscilloscope icon. The settings can be changed by placing the mouse over the dials and spinning the mouse wheel or by "pressing" the buttons with a mouse click. Adjust the volts per division on the channel under measurement until the amplitude of the waveform fills the majority of the screen. Adjust the Time-base in the Horizontal section such that a complete cycle or two are displayed. This instrument requires that its power button be pressed to "turn on" the oscilloscope. Figures 60, 61 and 62 contain *Quick Hints* on the use of the Agilent oscilloscope.
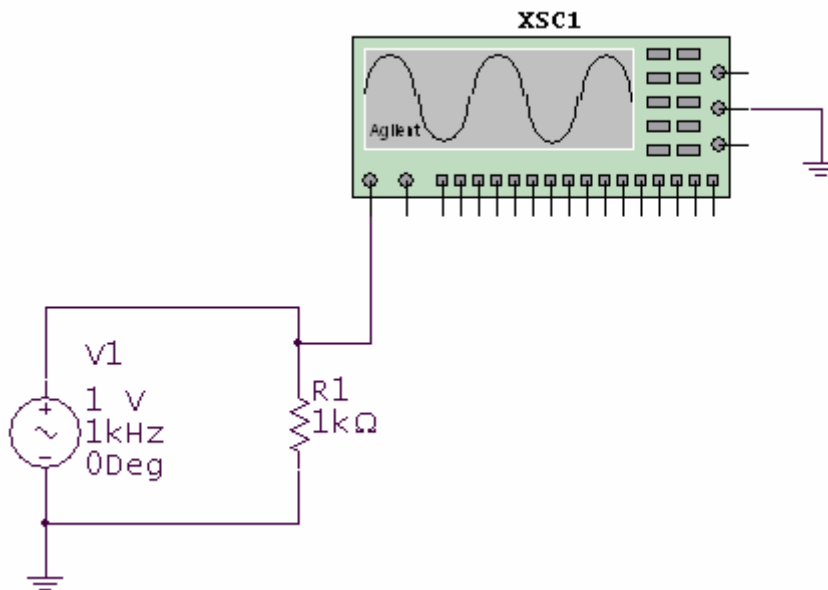

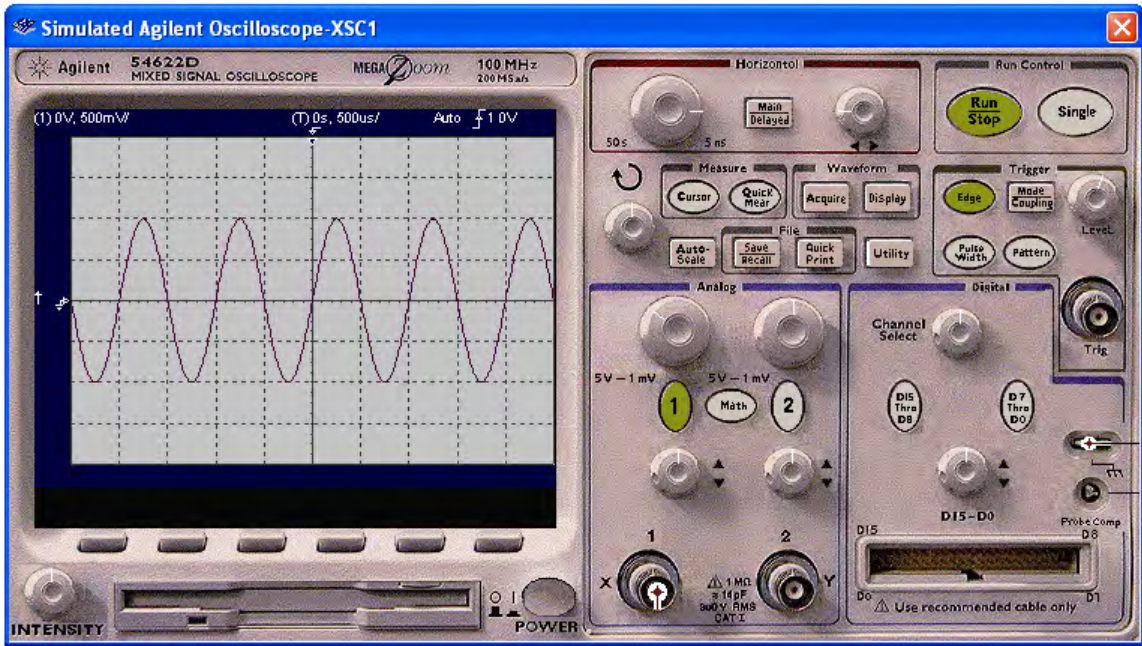**Figure 58 Voltage Measurement with an Agilent Oscilloscope**

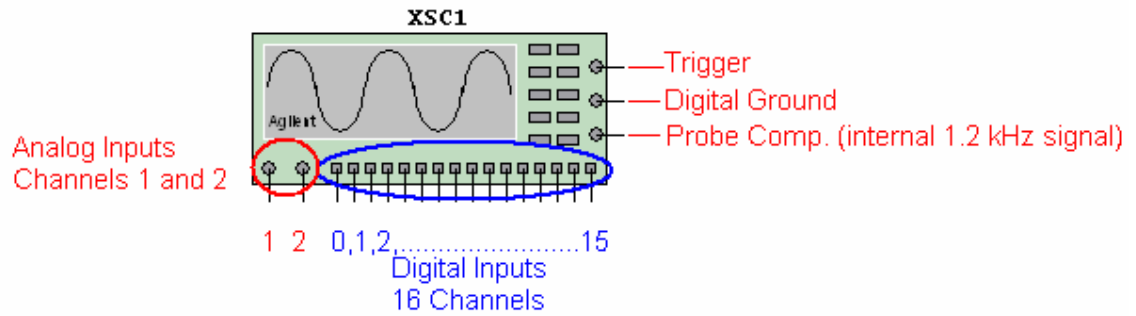**Figure 59 Agilent Oscilloscope Display**

Figure 60 Agilent Oscilloscope Icon Quick Hint
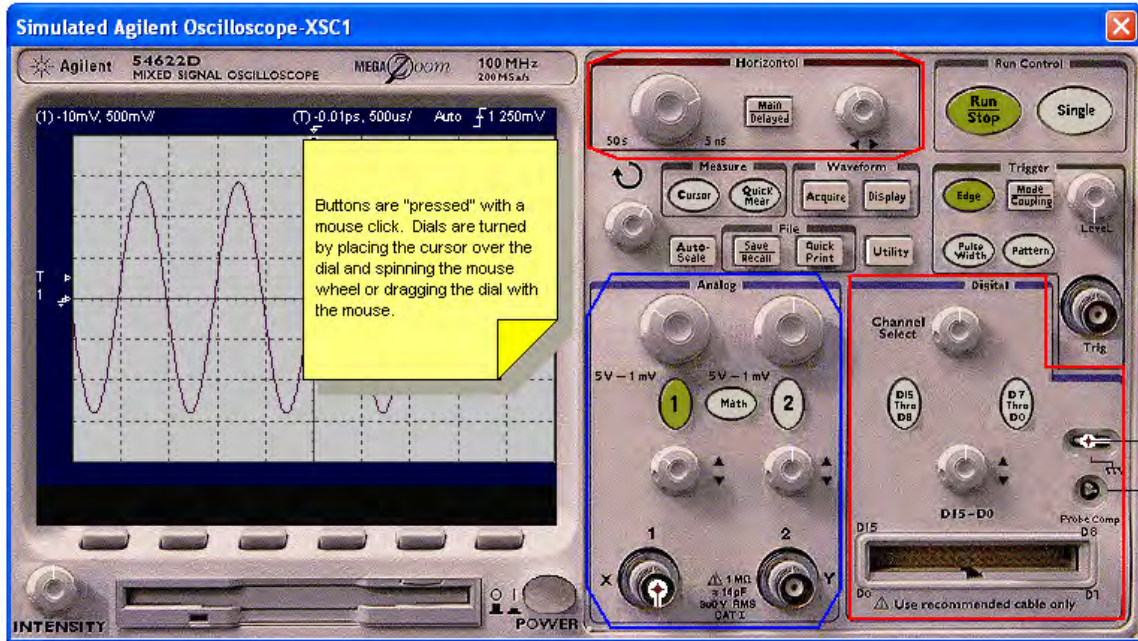
Figure 61 Agilent Oscilloscope Quick Hint
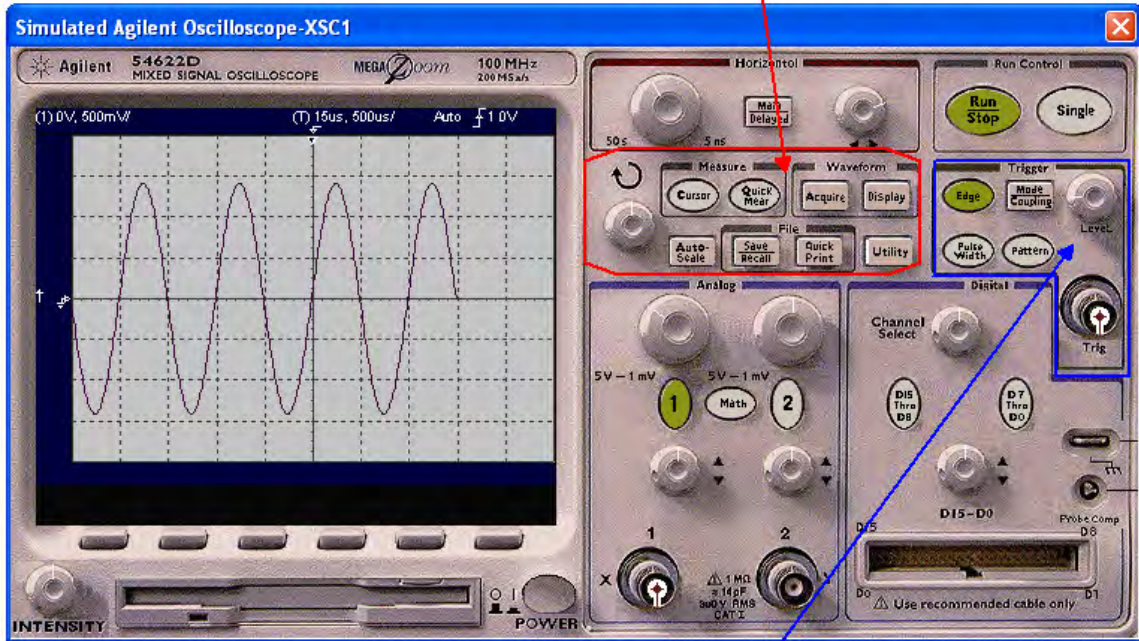
Figure 62 Agilent Oscilloscope Quick Hint

Example 9 Frequency and Voltage measurement using the Tektronix Oscilloscope in a series AC circuit.

In Figure 63, channel 1 of the Tektronix oscilloscope is connected to the positive side of the resistor. The ground connection of the scope and the circuit must be grounded. The oscilloscope will use this ground as a reference point. The Tektronix oscilloscope has all of the functionality of the Model TDS2024 four channel Digital Storage oscilloscope. The oscilloscope display is brought up by mouse clicking on the oscilloscope icon. The settings can be changed by placing the mouse over the dials and spinning the mouse wheel or by "pressing" the buttons with a mouse click. Adjust the volts per division on the channel under measurement until the amplitude of the waveform fills the majority of the screen.   Adjust the Time-base in the Horizontal section such that a complete cycle or two of the waveform is displayed. This instrument requires that its power button be pressed to "turn on" the oscilloscope. The voltage and frequency can be measured by the user or by using the "Measure" function of the oscilloscope.
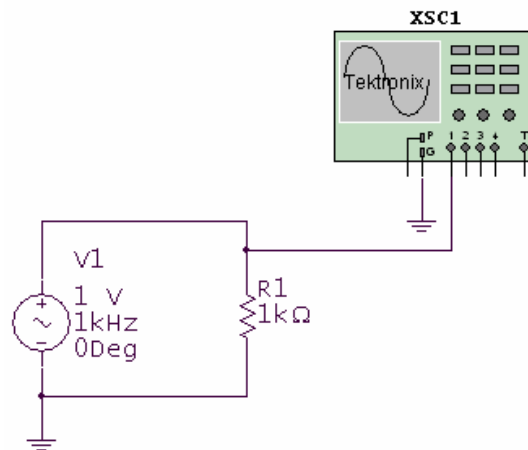

Figure 63 Voltage and Frequency measurement with a Tektronix Oscilloscope
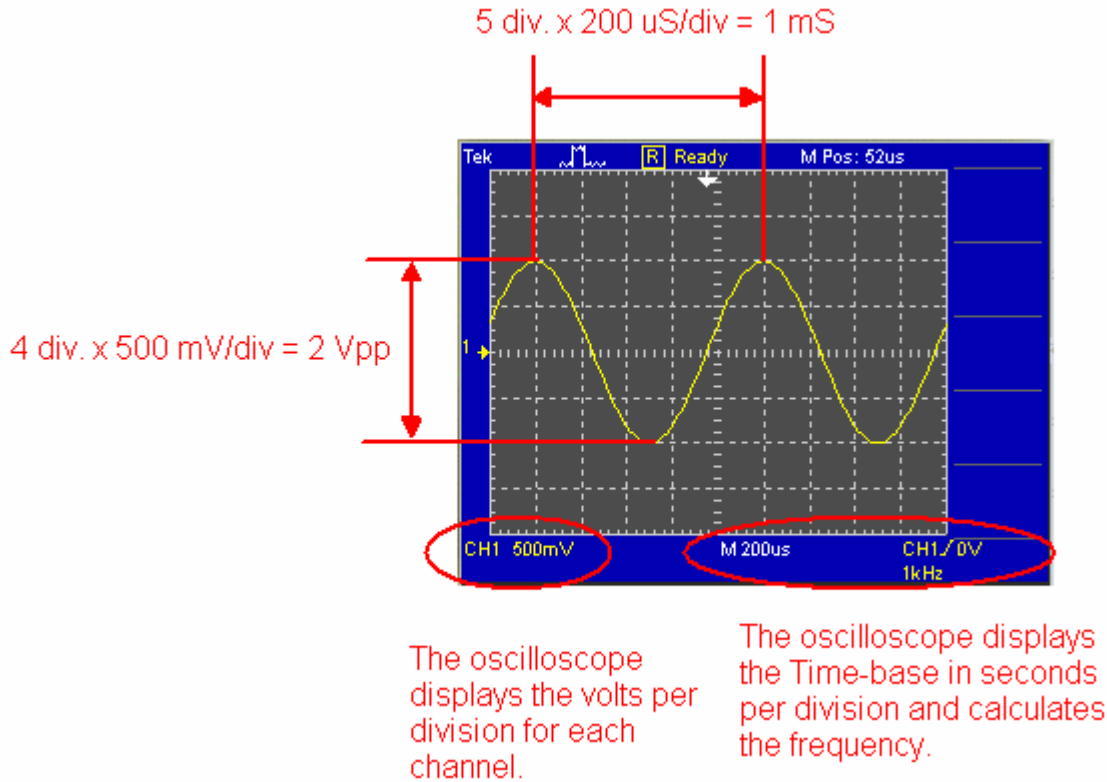
Figure 64 Measurement of the Period of the Waveform

Using volts per division and the seconds per division settings, the amplitude and frequency of the waveform in Figure 64 can be determined. The amplitude of the waveform is two divisions above zero volts. (The yellow arrow points to the zero reference point.) The volts per division setting is set to 500 mV per division.

$$V_P = 2 \ divisions \ x \ \frac{500 \, mV}{division}$$

$$V_P = 1V$$

$$V_{PP} = 4 \ divisions \ x \ \frac{500 \, mV}{division}$$

$$V_{PP} = 2V$$

The period of the waveform is measured to be 1 mS. Since frequency is the reciprocal of the period, the frequency can be calculated.

$$F = \frac{1}{T}$$

$$F = \frac{1}{1mS}$$

$$F = 1kHz$$

The Tektronix oscilloscope can also perform the voltage and frequency measurements automatically through the use of the "Measure" function. The four steps and the resulting display are shown in Figures 65 and 66. To set-up the oscilloscope to measure these values automatically:

1. Press the Measure button.
2. Select the channel to be measured.
3. Select what is to be measured: Vpp, Frequency, etc.
4. Return to the Main Screen.
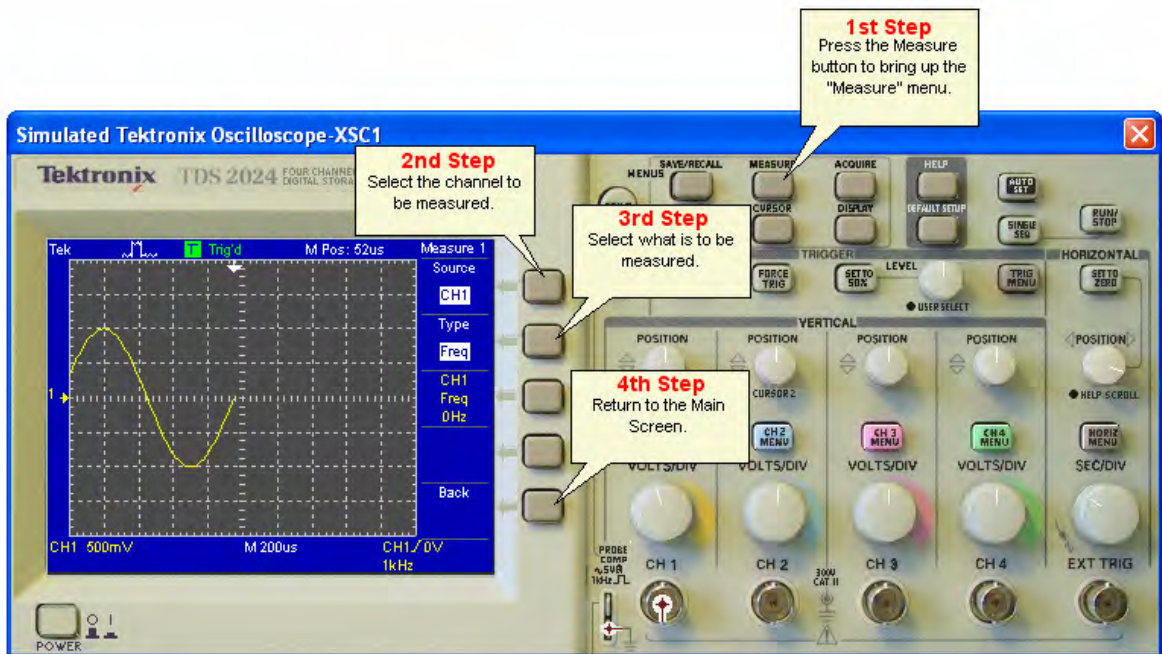5. Repeat for other channels and or values.

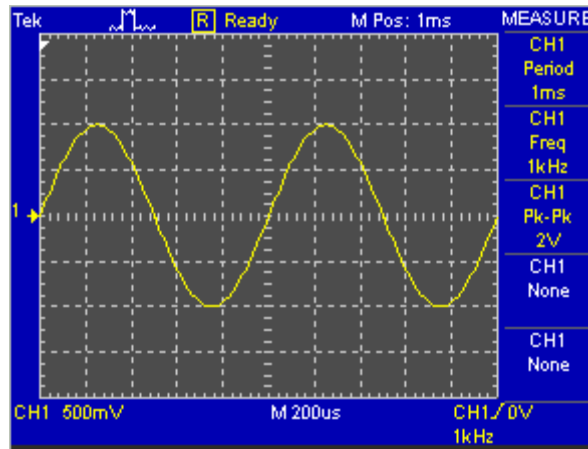**Figure 65 Tektronix Oscilloscope Measurement Function Set-up**

Figure 66 Tektronix Oscilloscope Measurement Display

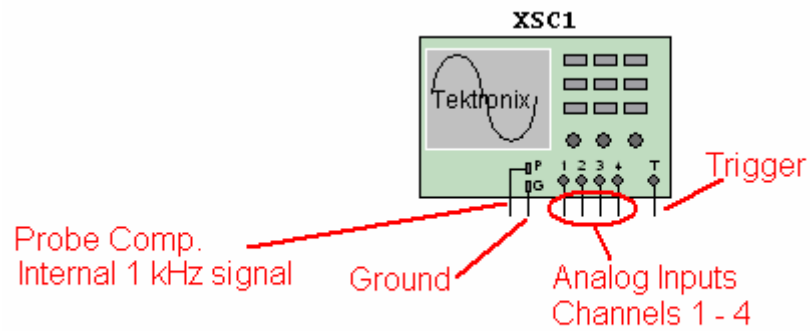Figures 67, 68 and 69 contain *Quick Hints* on the use of the Tektronix oscilloscope.

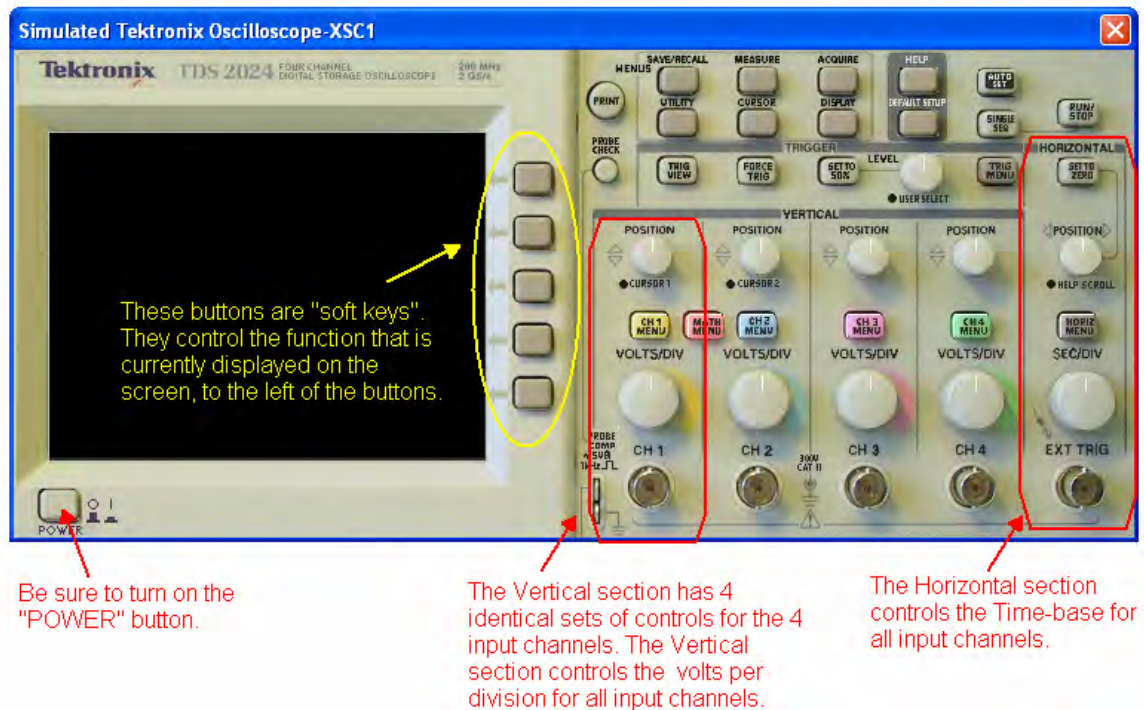**Figure 67 Tektronix Oscilloscope Icon Quick Hint**

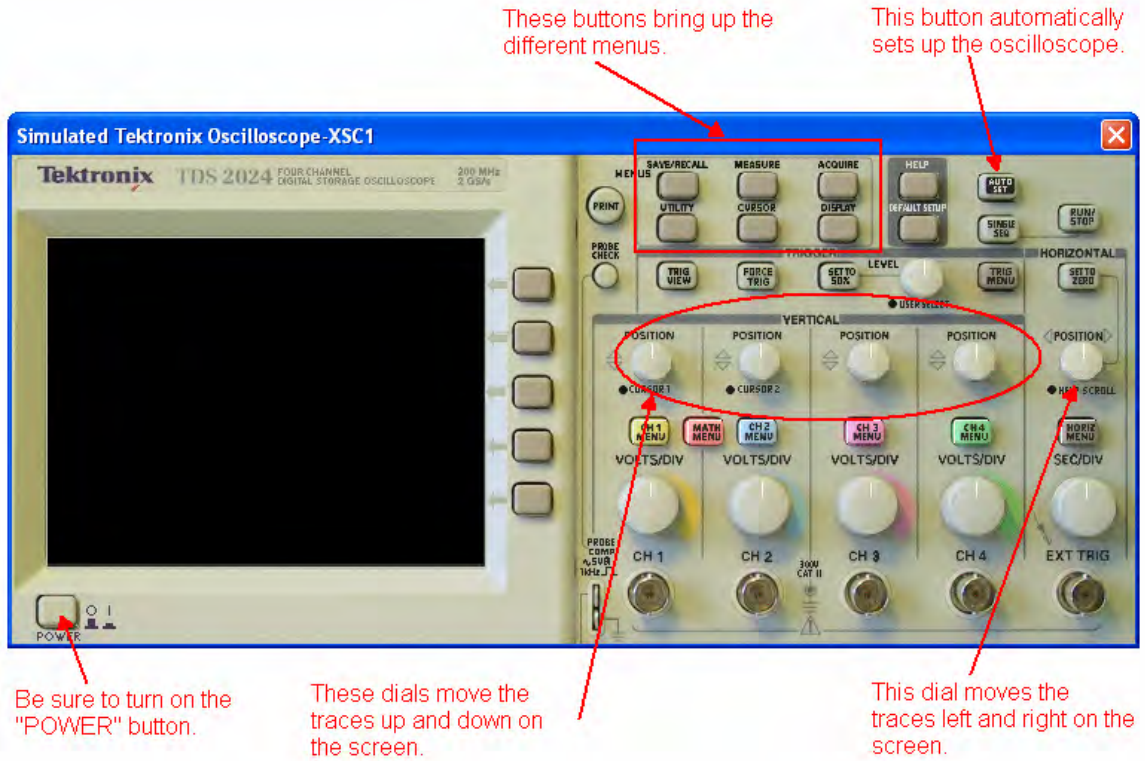**Figure 68 Tektronix Oscilloscope Quick Hint**

Figure 69 Tektronix Oscilloscope Quick Hint

Conclusion

The ability to create a schematic quickly and then analyze the circuit through simulation makes MultiSim a wonderful tool to help students understand the concepts covered in the study of DC, AC, and semiconductor electronics.

This appendix introduced the reader to the features of MultiSim that directly relate to the topics covered in this textbook. The features covered in this appendix will help the student to utilize the compact disk included with this textbook to its fullest extent.