

Multisim v11 Digital Primer

to accompany

**Electronics: Principles and Applications
Eighth Edition
By Charles Schuler**

Introductions

In an effort to help the reader understand the concepts presented in this textbook, key examples and problems will be presented through the use of computer simulation using Multisim. Multisim is an interactive circuit simulation package that allows users to view their circuit in schematic form while measuring the various parameters of the circuit. The ability to quickly create a schematic 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 following topics are covered:

- Work Area
- Opening a File
- Running a Simulation
- Saving a File
- Components
- Sources
- Measurement Equipment
- Circuit Examples
- User Customization
- Exporting Data to Excel
- Adding Text and Graphics

A note on “mouse clicking”: The instruction *click* means pressing the left mouse button. A *double-click* means pressing the left mouse button twice. A *right-click* means pressing the right mouse button.

Work Area

The power of Multisim 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 menu bar (with its drop-down menus), the toolbars, and the work area.

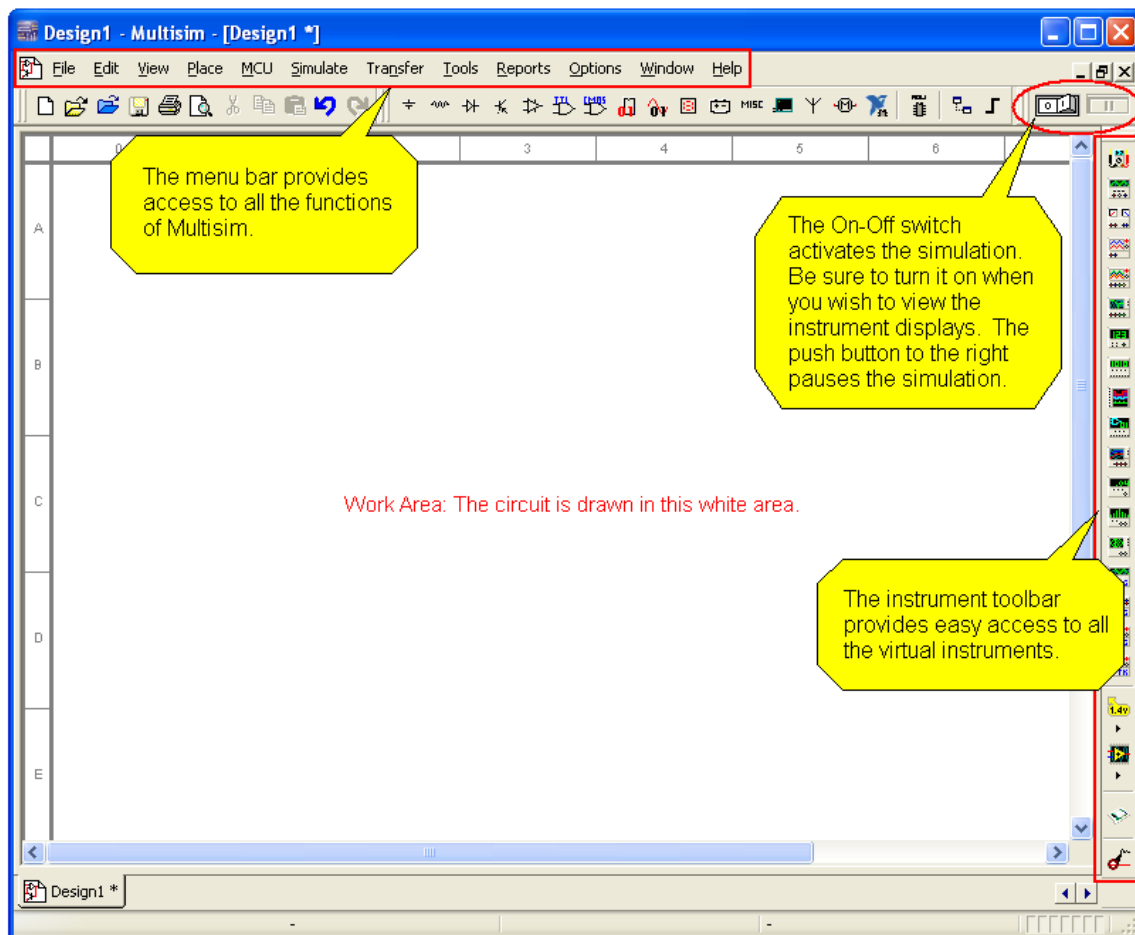


Figure 1 Main Screen

The menu bar gives the user access to all the functions of the program. Initially, the user will only utilize a few menu selections. Each drop-down menu is accessed by either clicking the menu name or by pressing the Alt key and the menu name's underlined letter. For example, to access the *File* menu, simply press Alt + F (holding down the Alt key while pressing the F key). The *File* menu will drop down as shown in Figure 2.

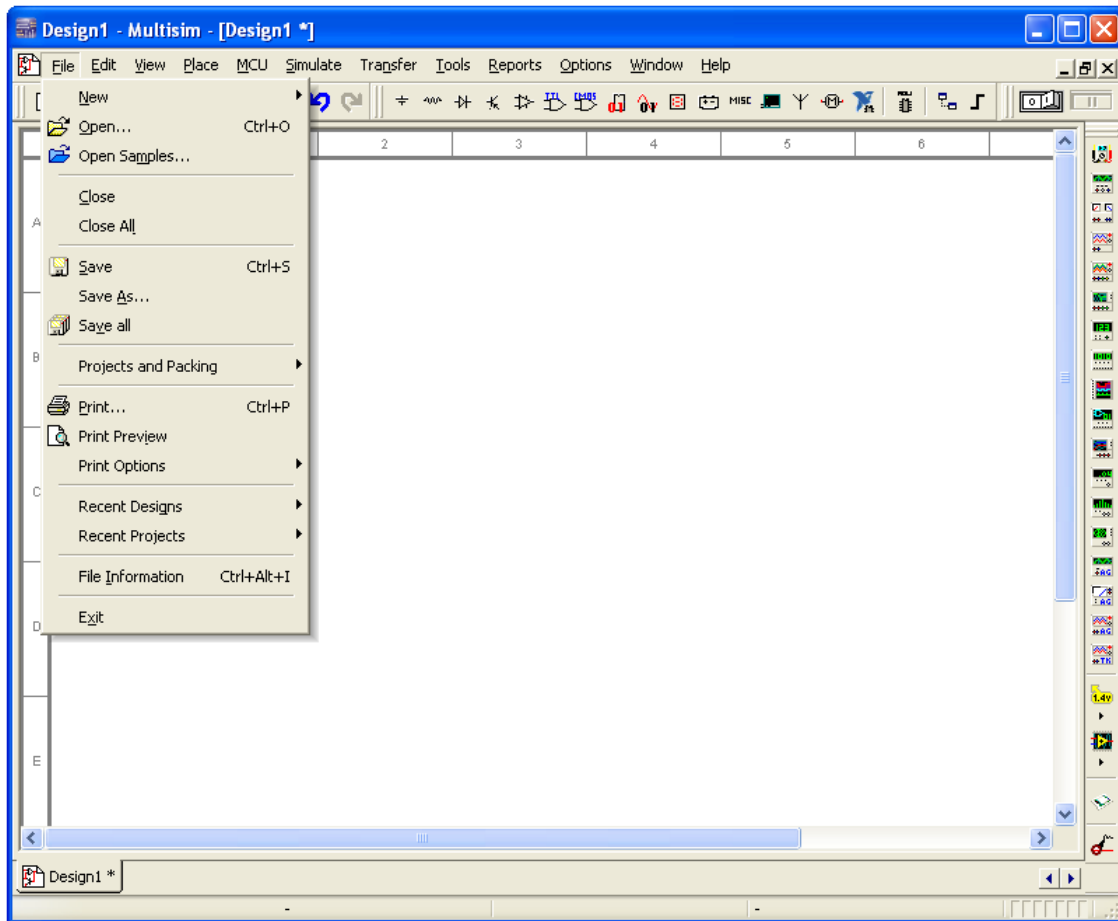


Figure 2 *File* Menu

Initially, only two commands from the *File* menu need be mastered: opening a file and saving a file. The remaining commands can be explored as time permits.

The toolbars beneath the menu bar provide access to all the commands contained in the menus. Typically, a user will access the commands through the toolbars instead of the

menu bar. The most important icon in the assorted toolbars 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 user to view 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 the 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 3 Problems 3-1*. This first part of the file name tells the user that the file is located in the folder labeled *Chapter 3*. The second part of the file name tells the user that it is question 1 out of the Problems section at the end of Chapter 3.

To open a file, either click *File* on the menu bar and then click the *Open* command or click the open folder icon located on the toolbar (below the menu 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.

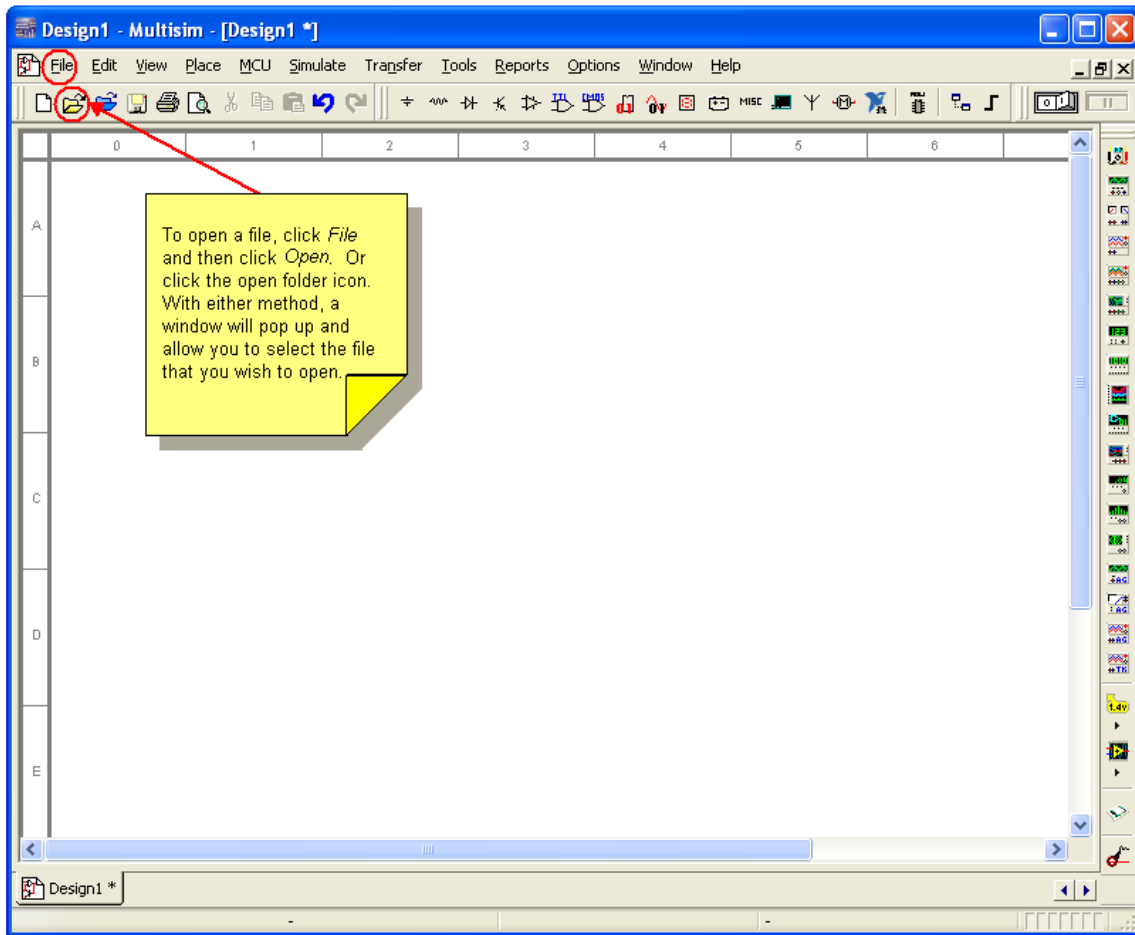


Figure 3 Opening a File

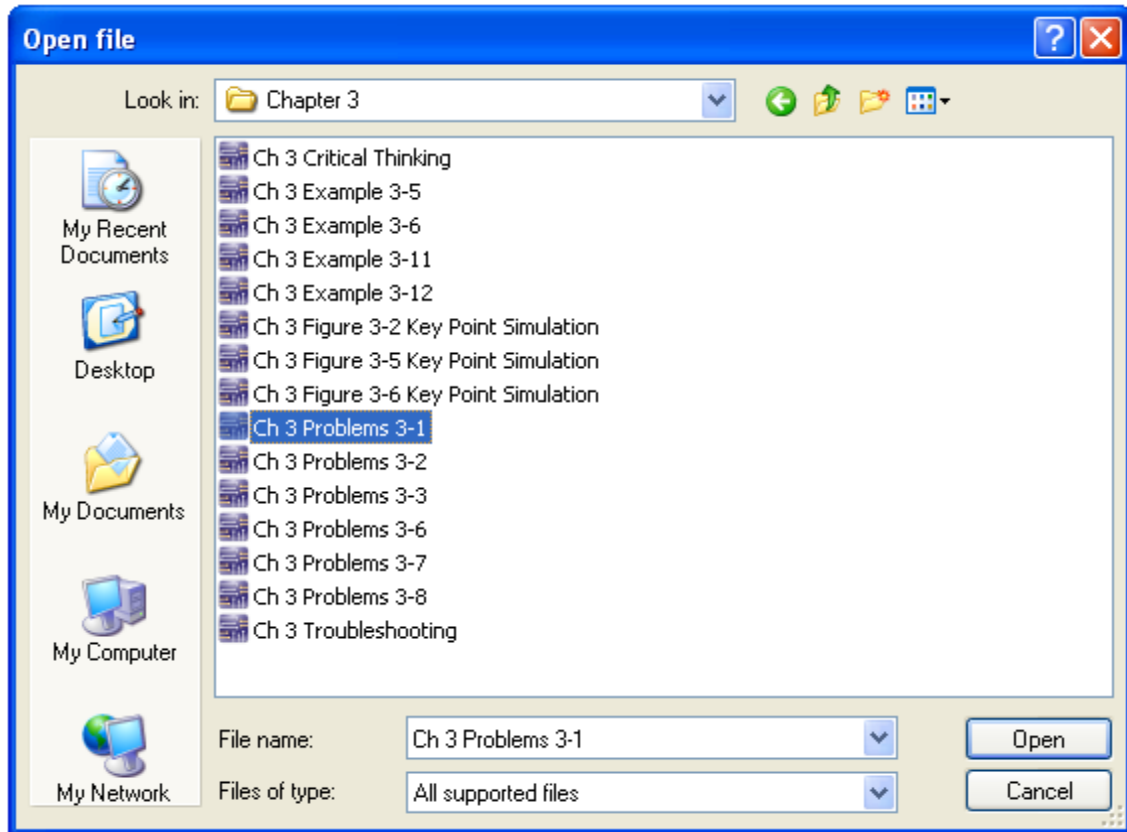


Figure 4 *Open File* Dialog Box

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-click the instrument icon and the display will pop up. There is a help screen referred to as the *Description Box* with information relating to the circuit and the instruments within the circuit. This *Description Box* is opened by pressing Ctrl + D while in Multisim.

The simulation can be started three ways:

1. Selecting *Simulate* from the menu bar and then selecting *Run*.
2. Pressing the F5 key.
3. Clicking the toggle switch.

All three ways are illustrated in Figure 5.

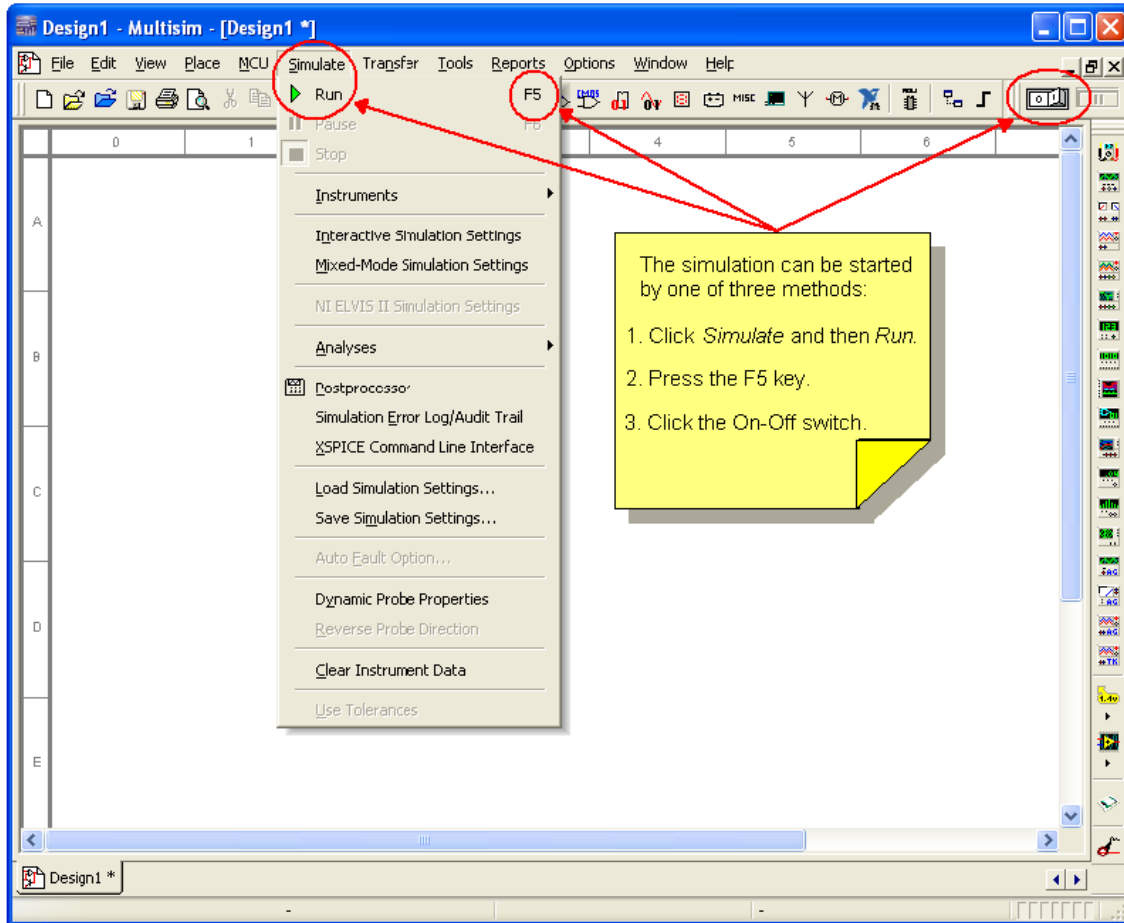


Figure 5 Starting the Simulation

Saving a File

If the file has been modified, it needs to be saved under a new file name. As shown in Figure 6, click *File* on the menu bar. Then click the *Save As* command. This will cause the *Save As* dialog box to open. Give the file a new name, and click the Save button. The process is demonstrated in Figure 7.

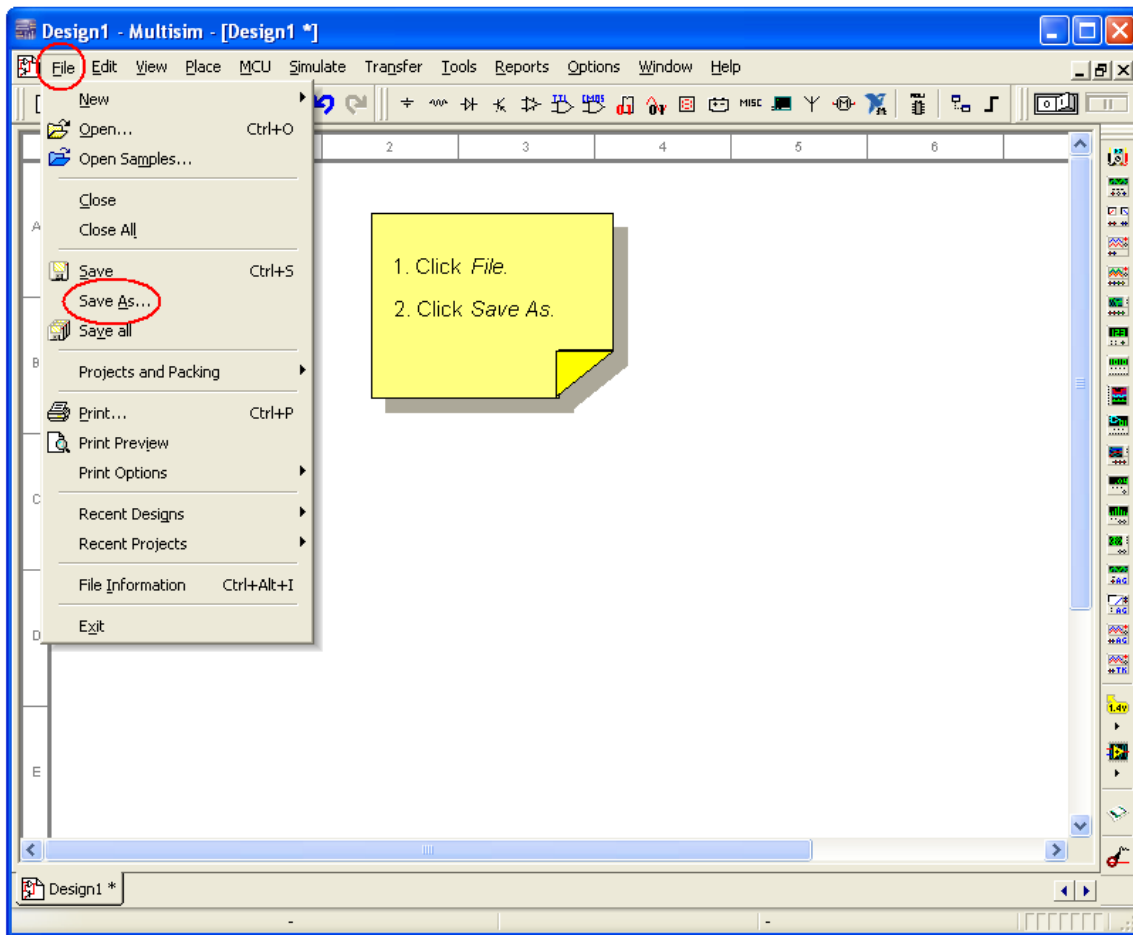


Figure 6 *Save As* Command

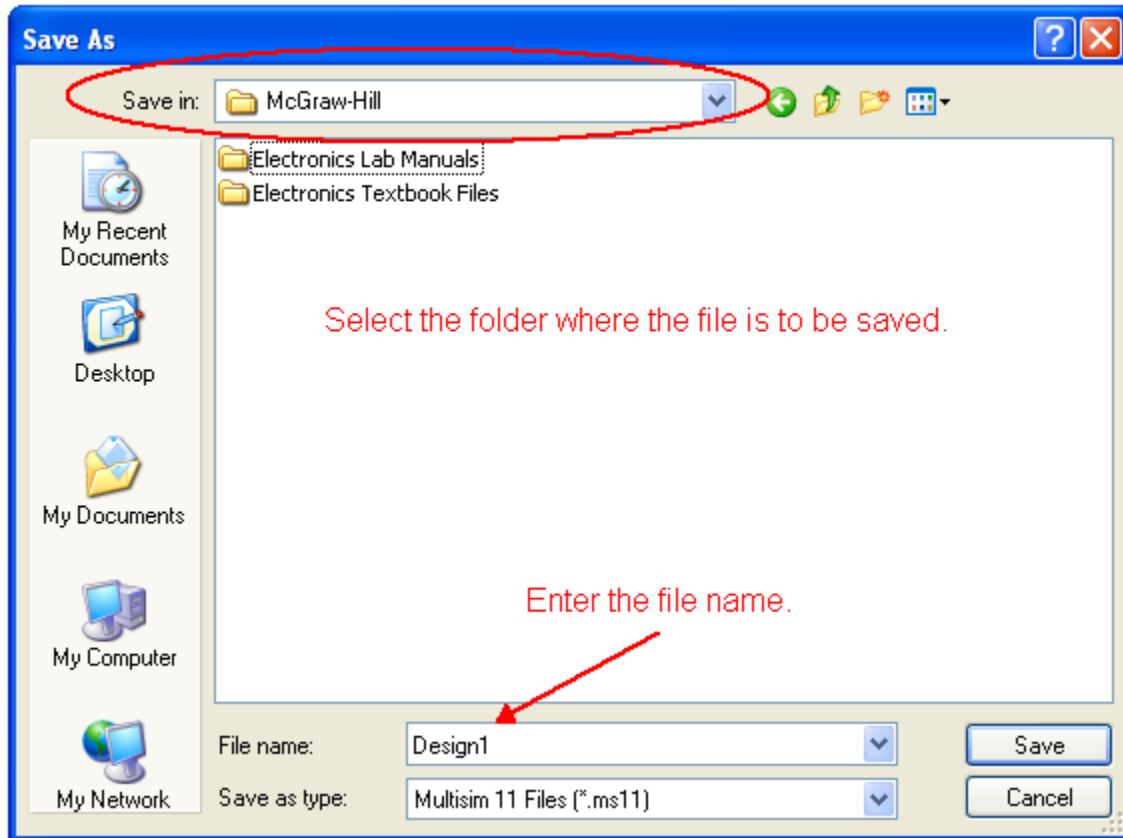


Figure 7 *Save As* Dialog Box

Components

Multisim has two kinds of component models: 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.

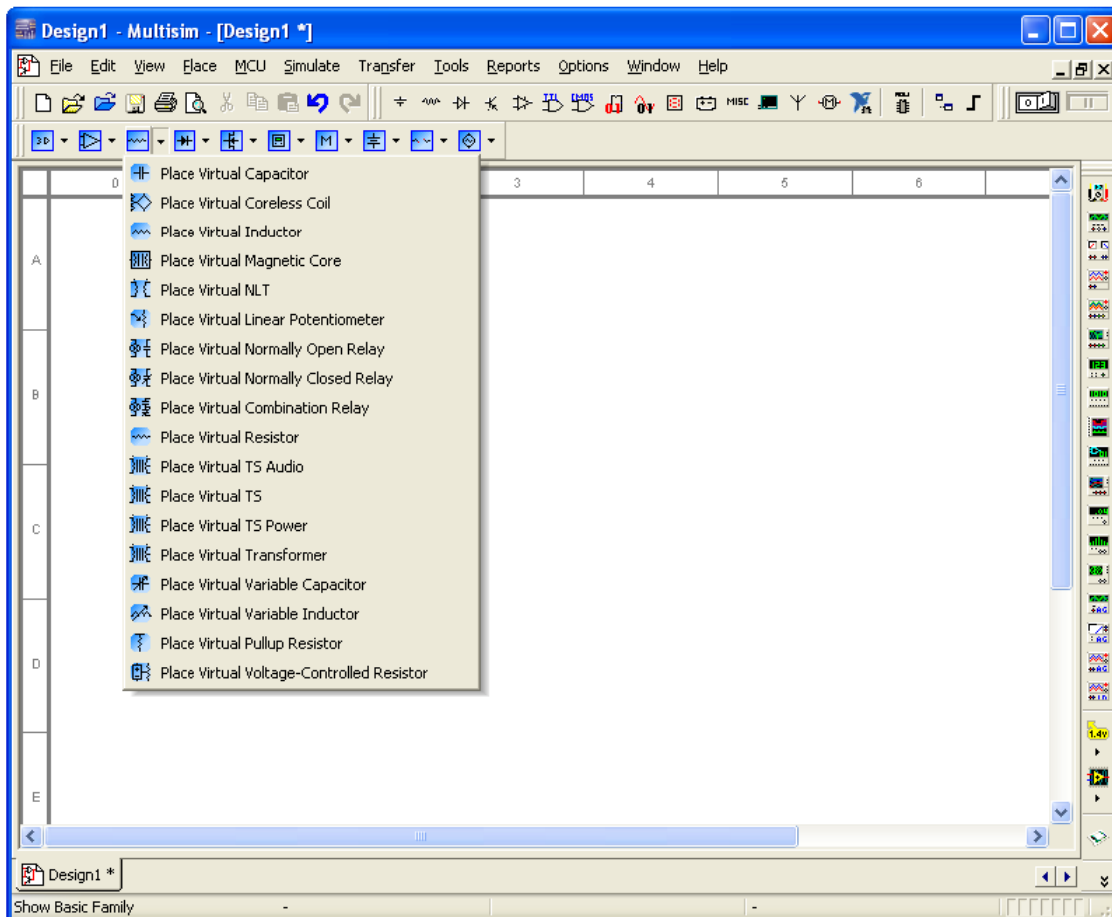


Figure 8 Virtual Component List

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 percentage tolerance, as shown in Figure 9.

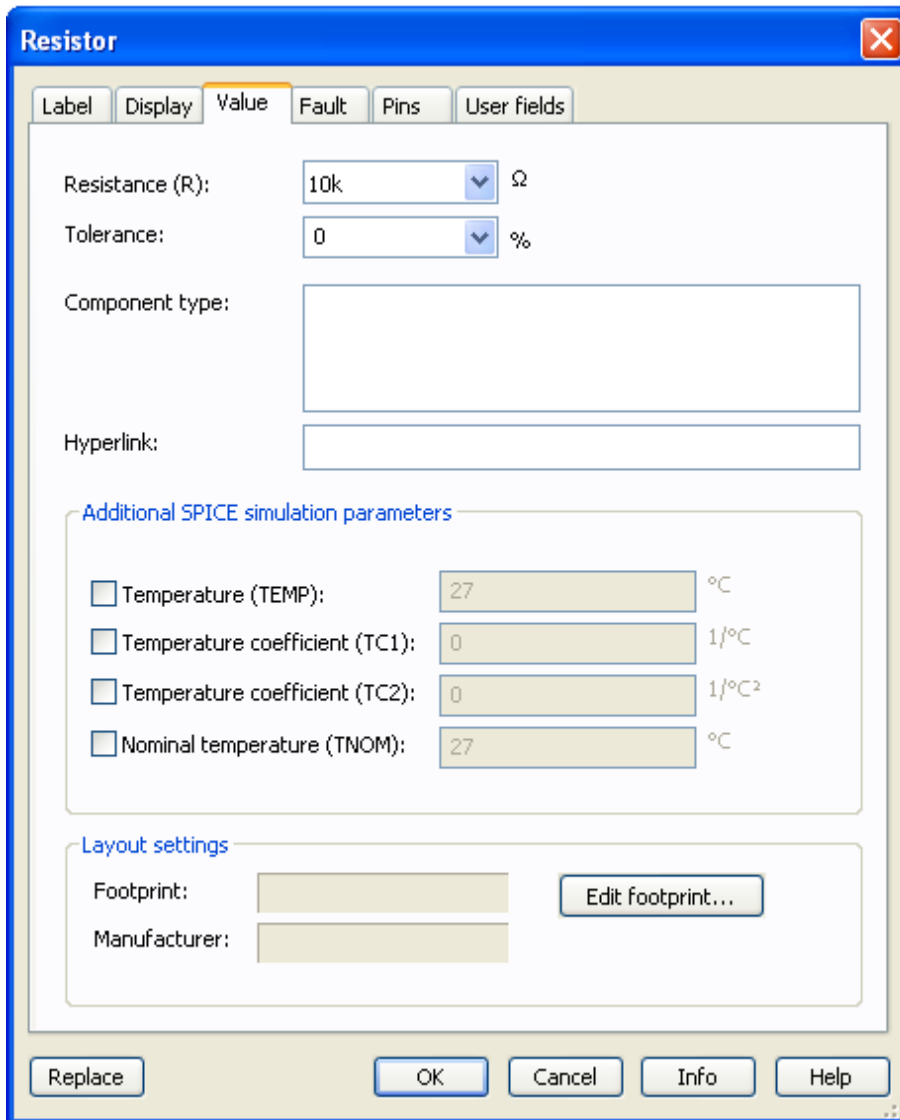


Figure 9 Configuration Screen for a Virtual Resistor

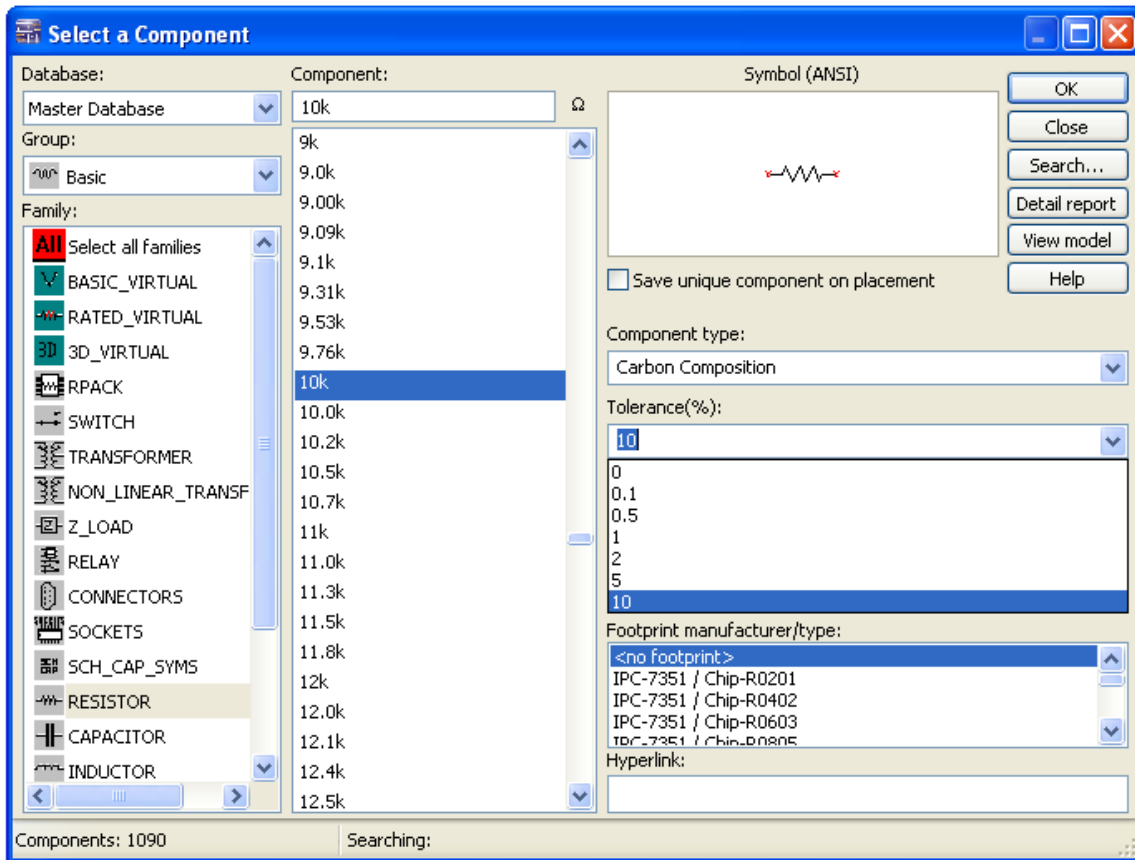


Figure 10 Component Listing for Resistors

The models of the actual resistors are available with tolerance values of 0, 0.1, 0.5, 1, 2, 5, and 10 percent. 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 0 percent tolerance.

Two 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. The DIP switch packs are available in two to ten switch configurations. Each individual switch within the switch pack is activated by either pressing the associated key or by clicking it. When the cursor is placed over an individual switch within a switch pack, a white rectangle appears around the switch and the cursor changes into a pointing hand symbol. When the white rectangle is present and the cursor resembles a pointing hand, a single click will cause the switch to change positions. The black dot on the switch pack signifies the ON position.

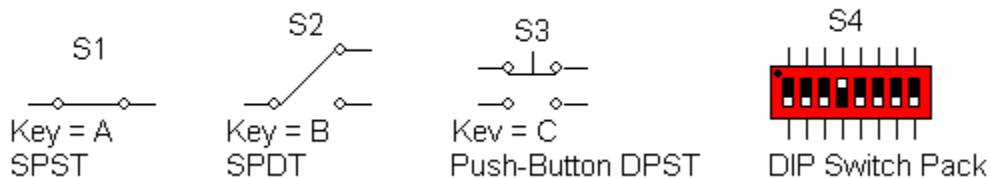


Figure 11 Switches

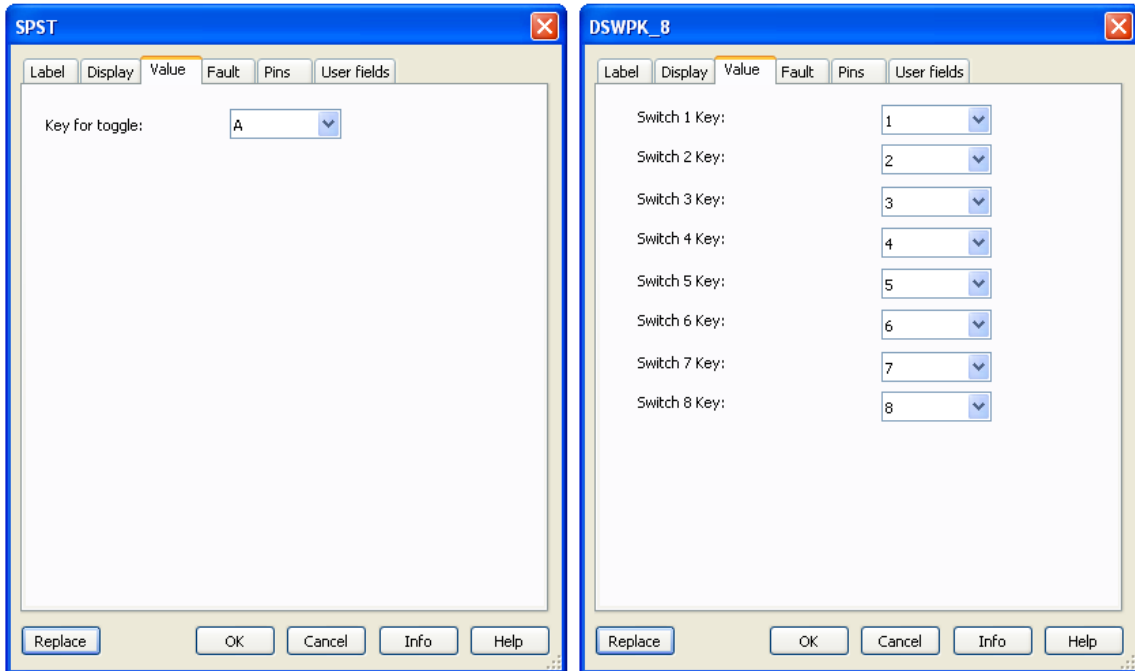


Figure 12 Switch Configuration Screens

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, whereas pressing the Shift key and the letter will decrease the resistance. A slider located to the right of the potentiometer can also be used to adjust the resistance value by dragging it with the mouse. 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 on the configuration screen. The associated key is also set in the configuration screen, as shown in Figure 14.

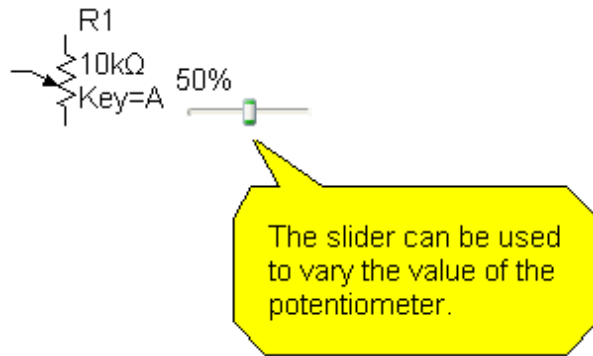


Figure 13 Potentiometer

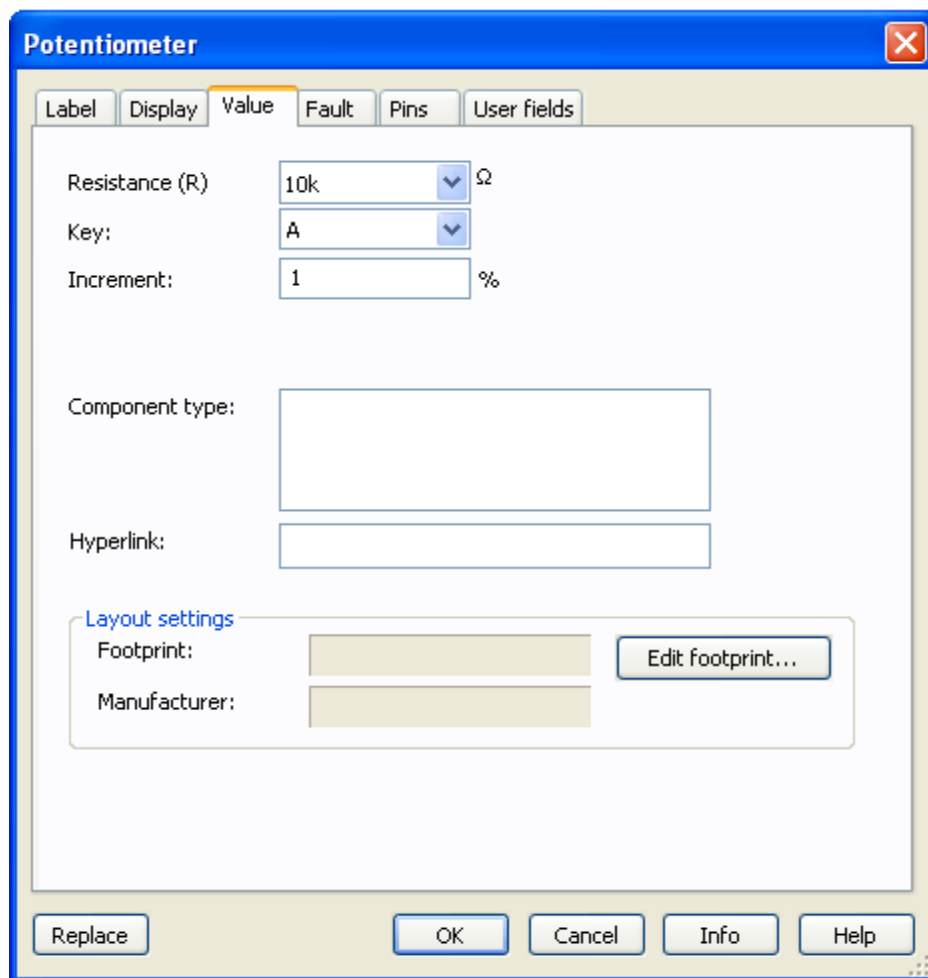


Figure 14 Potentiometer Configuration Screen

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 V dc. If the component is double-clicked, the configuration screen shown in Figure 16 will pop up and the voltage value can be changed.

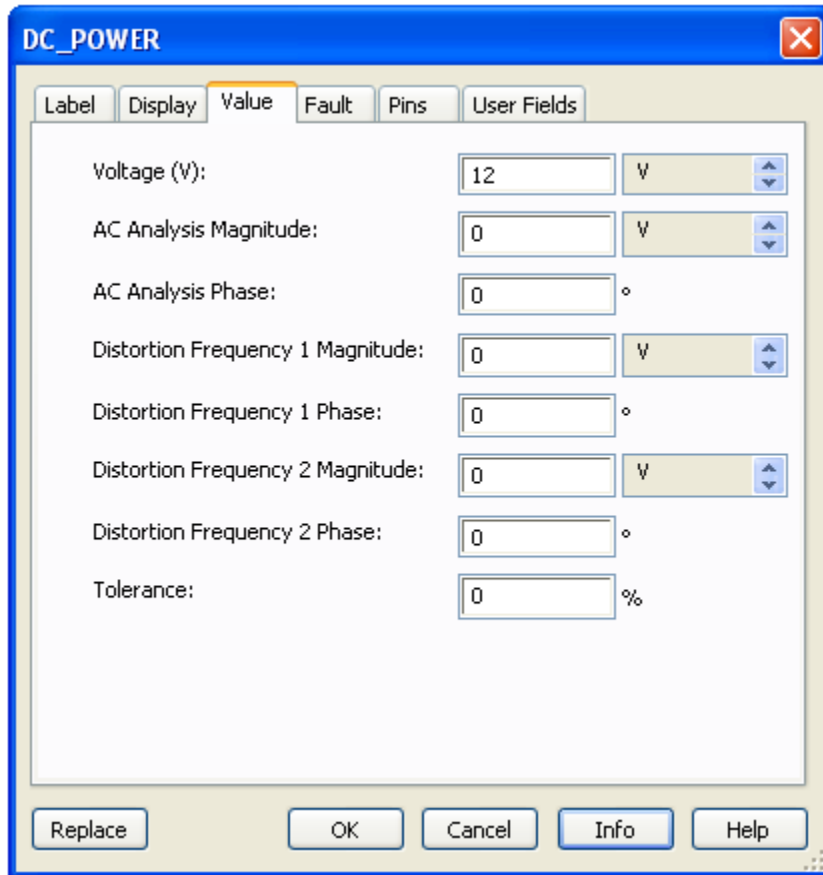


Figure 16 Configuration Screen for the DC Source

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. Field-effect transistor (FET) circuits will utilize the V_{DD} voltage supply as illustrated in Figure 18.



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 (op-amp) circuits. Operational amplifiers typically have two voltage supplies: a negative (V_{EE}) and a positive (V_{CC}) voltage supply, as shown in Figure 20.



Figure 19 V_{CC} and V_{EE} Voltage Sources

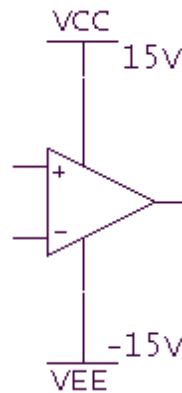


Figure 20 V_{CC} and V_{EE} Op-Amp Example

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

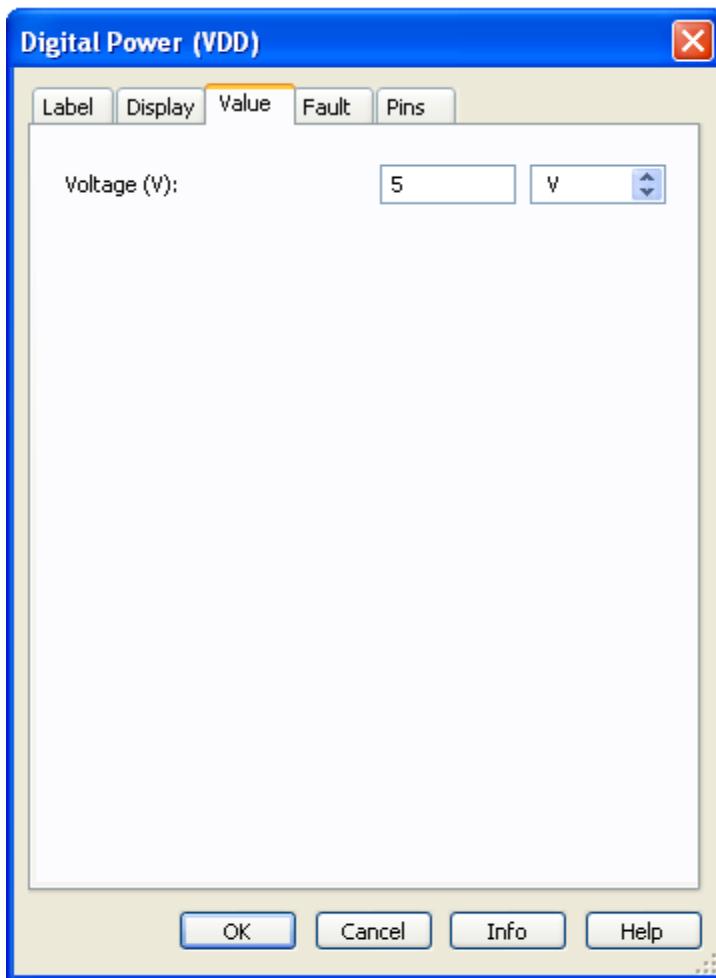


Figure 21 V_{DD} Configuration Screen

The ac source can be represented as either a schematic symbol or a function generator.

The schematic symbol for an ac source can represent either a power source or a signal voltage source. The amplitude of the power source will be in V_{RMS} , whereas the amplitude of the signal voltage source will be in V_P .

The schematic symbols for the two ac sources, as shown in Figure 22, will include information about the ac source. This information will include the device reference number, V_{RMS} value or V_P value, frequency, and phase shift. These values are fully adjustable. The default values for the two sources are shown in Figure 23. If the component is double-clicked, the configuration screen will pop up and the value can be changed.

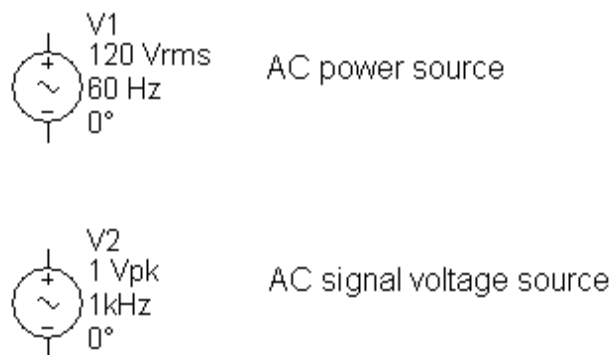


Figure 22 AC Sources

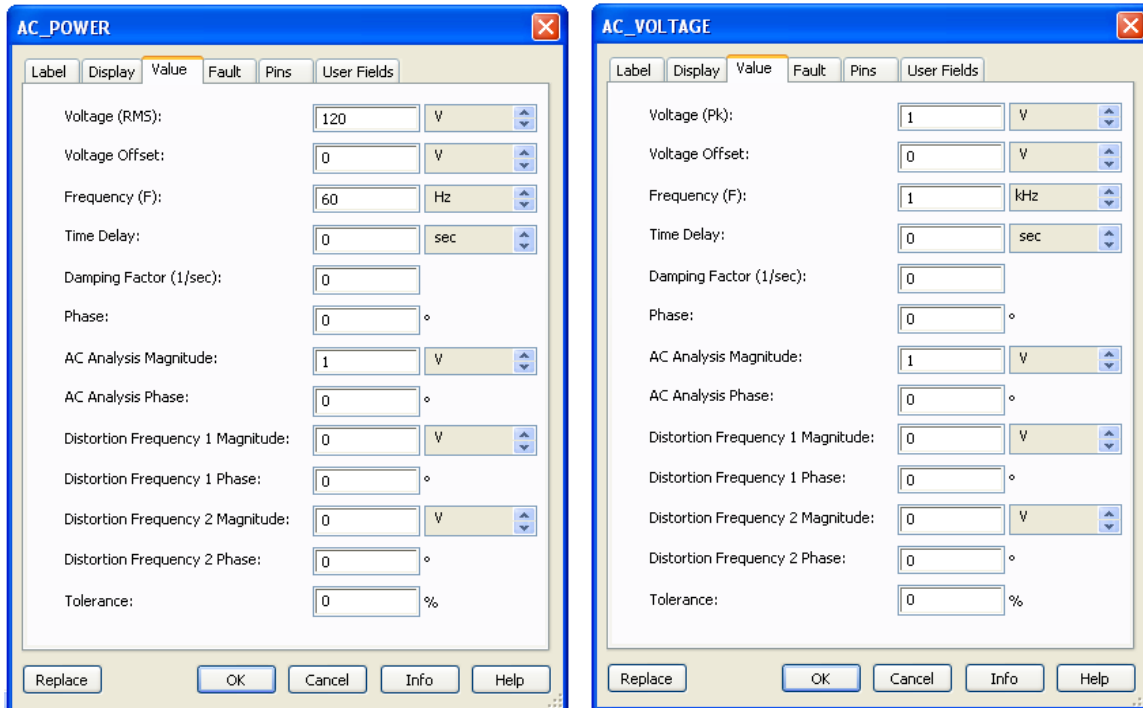


Figure 23 Configuration Screens for the Two AC Sources

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 double-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.

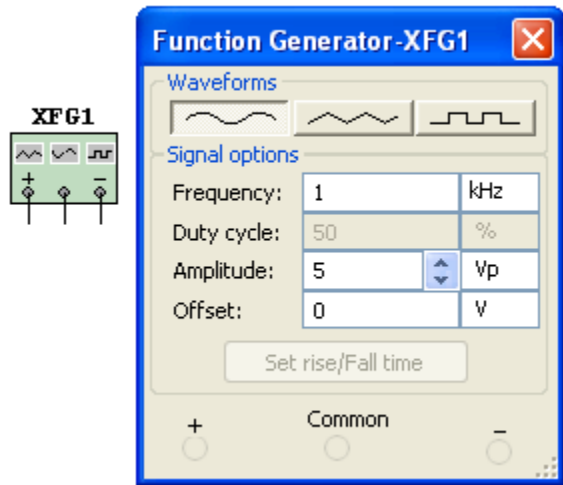


Figure 24 Generic Function Generator and Configuration Screen

The Agilent function generator is controlled via the front panel, as shown in Figure 25.

The buttons are “pushed” by clicking. The dial can be turned by dragging the mouse over it or by placing the cursor over it and spinning the mouse wheel. The latter is by far the preferred method.

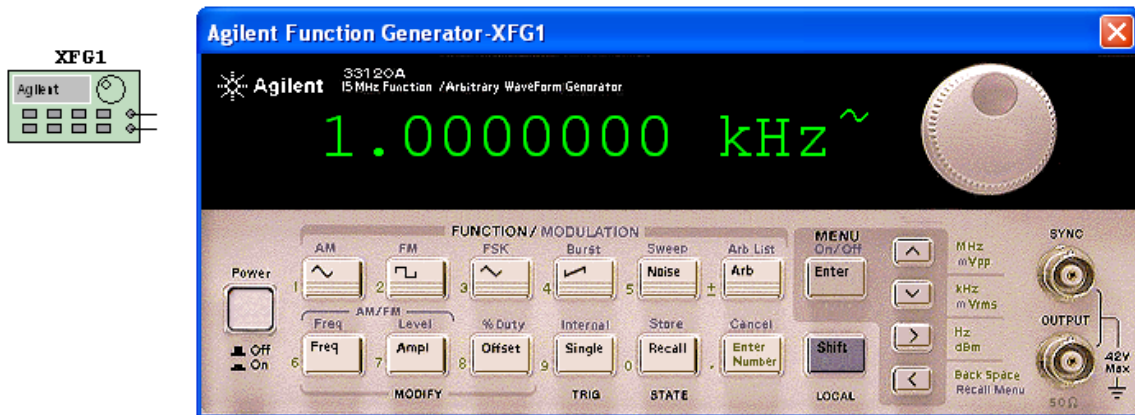


Figure 25 Agilent Function Generator

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

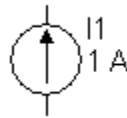


Figure 26 DC Current Source

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.

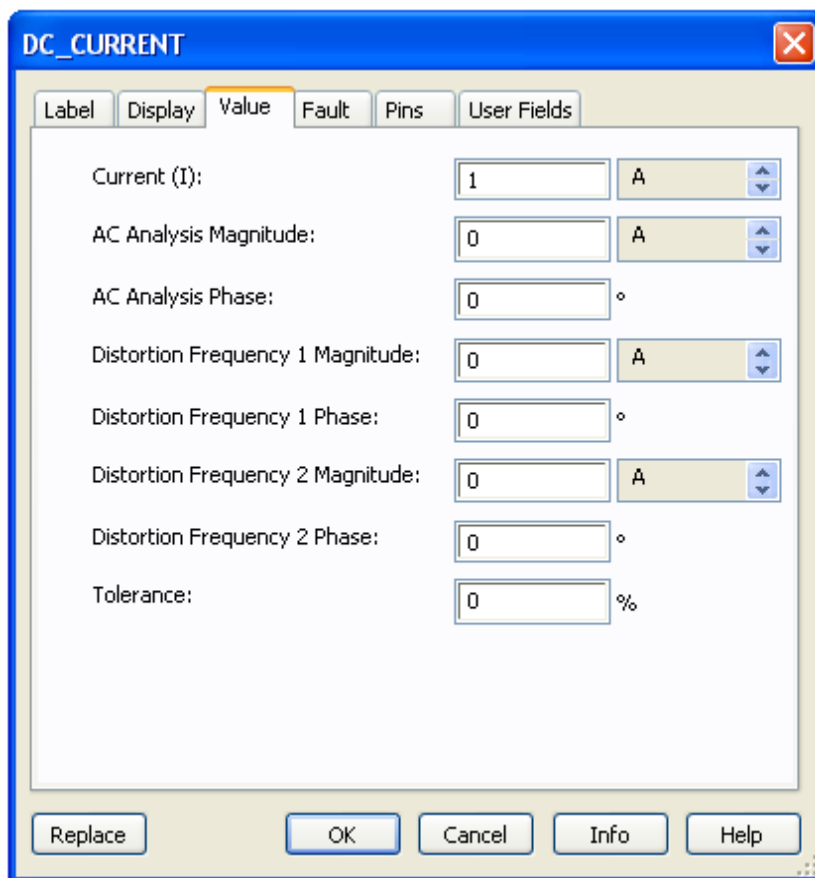


Figure 27 DC Current Source Configuration Screen

The ac current source is represented as a circle with an upward 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.

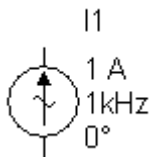


Figure 28 AC Current Source

| Parameter | Value | Unit |
|-----------------------------------|-------|------|
| Current (Pk): | 1 | A |
| Current Offset: | 0 | A |
| Frequency (F): | 1 | kHz |
| Time Delay: | 0 | sec |
| Damping Factor (1/sec): | 0 | |
| Phase: | 0 | ° |
| AC Analysis Magnitude: | 1 | A |
| AC Analysis Phase: | 0 | ° |
| Distortion Frequency 1 Magnitude: | 0 | A |
| Distortion Frequency 1 Phase: | 0 | ° |
| Distortion Frequency 2 Magnitude: | 0 | A |
| Distortion Frequency 2 Phase: | 0 | ° |
| Tolerance: | 0 | % |

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 versus frequency. This is usually done by taking many measurements and plotting the results in a spreadsheet. The Bode plotter automates this time-consuming process.

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 clicking the icon to the left in Figure 31. Clicking will cause the multimeter display to pop up. The different functions on the display can be selected “pushing” the different buttons via clicking.

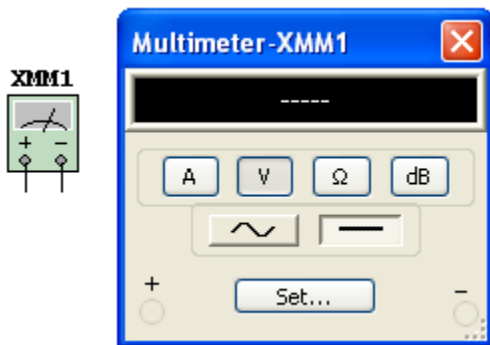


Figure 31 Generic Multimeter Icon and Configuration Screen

The Agilent multimeter icon and meter display are shown in Figure 32. The display is brought up by clicking 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 clicking a 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.

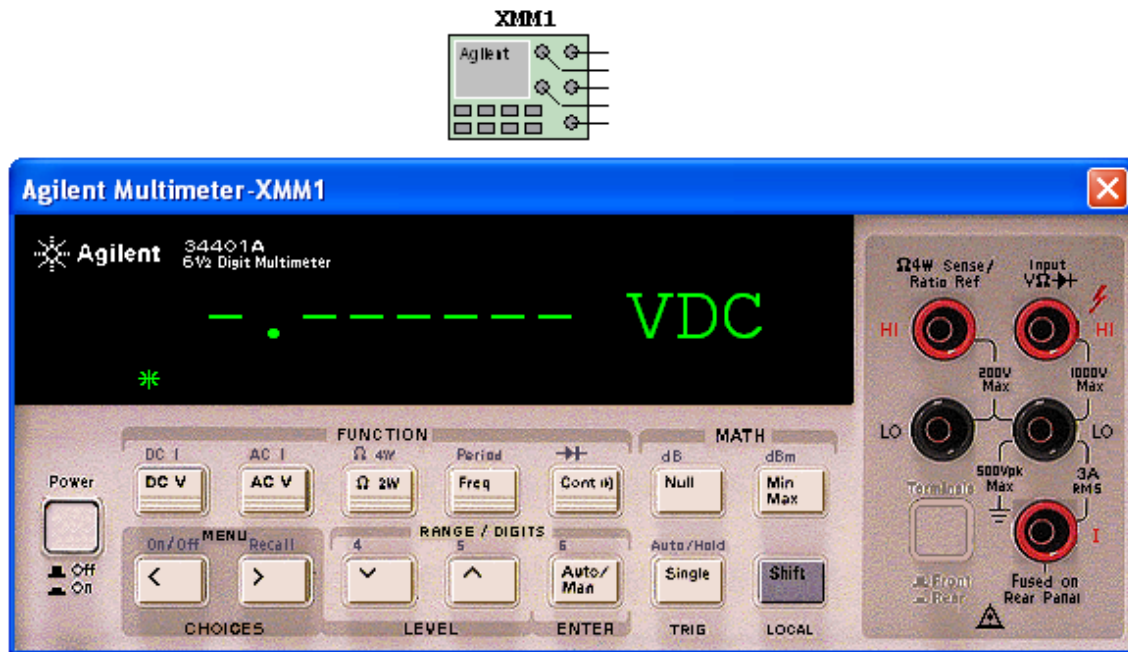


Figure 32 Agilent Multimeter Icon and Meter Display

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 clicking the oscilloscope icon. The settings can be changed by clicking each box and bringing up the scroll arrows. The color of the traces will match the color of the wire segments connecting the oscilloscope to the circuit. To change the color of the trace, right-click the wire segment connecting the oscilloscope to the circuit and then select *Color Segment* from the pop-up box. Select the color for the wire segment and corresponding trace from the color palette.

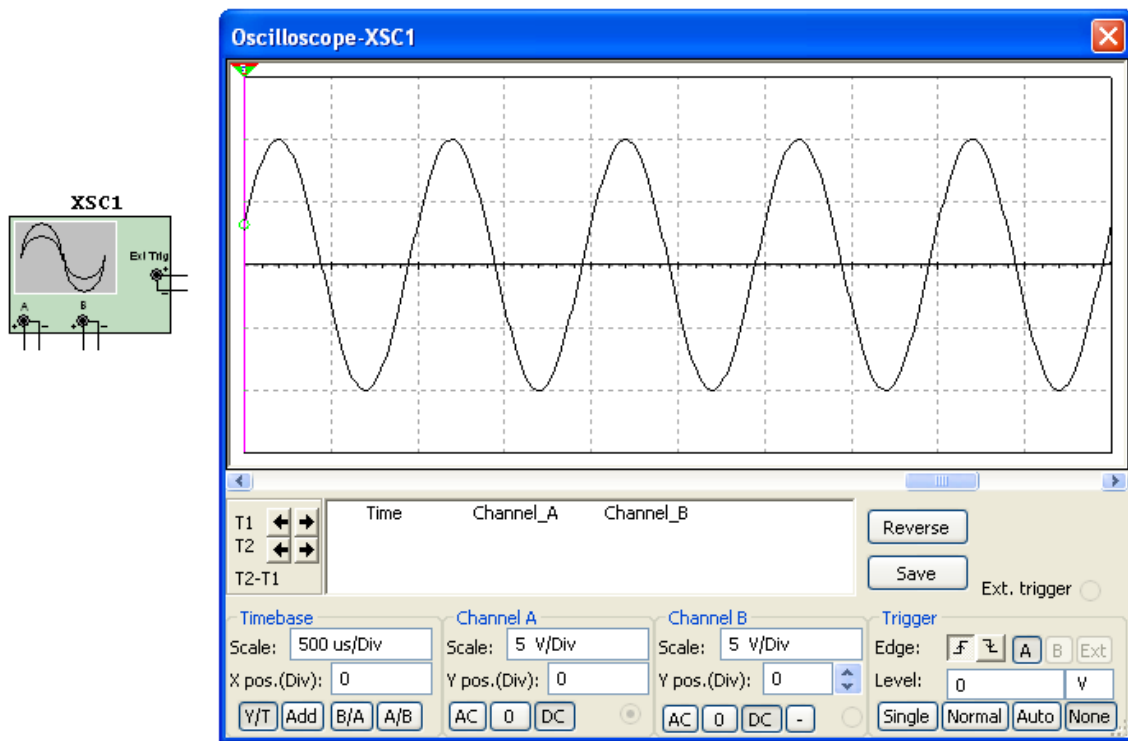


Figure 33 Generic Oscilloscope Icon and Oscilloscope Display

The Agilent oscilloscope icon in Figure 34 has all 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 clicking. The dials can be turned by dragging the mouse over it or by placing the cursor over it and spinning the mouse wheel.

The latter is the preferred method.

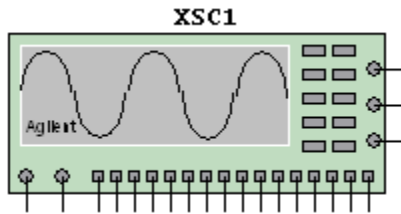


Figure 34 Agilent Oscilloscope Icon

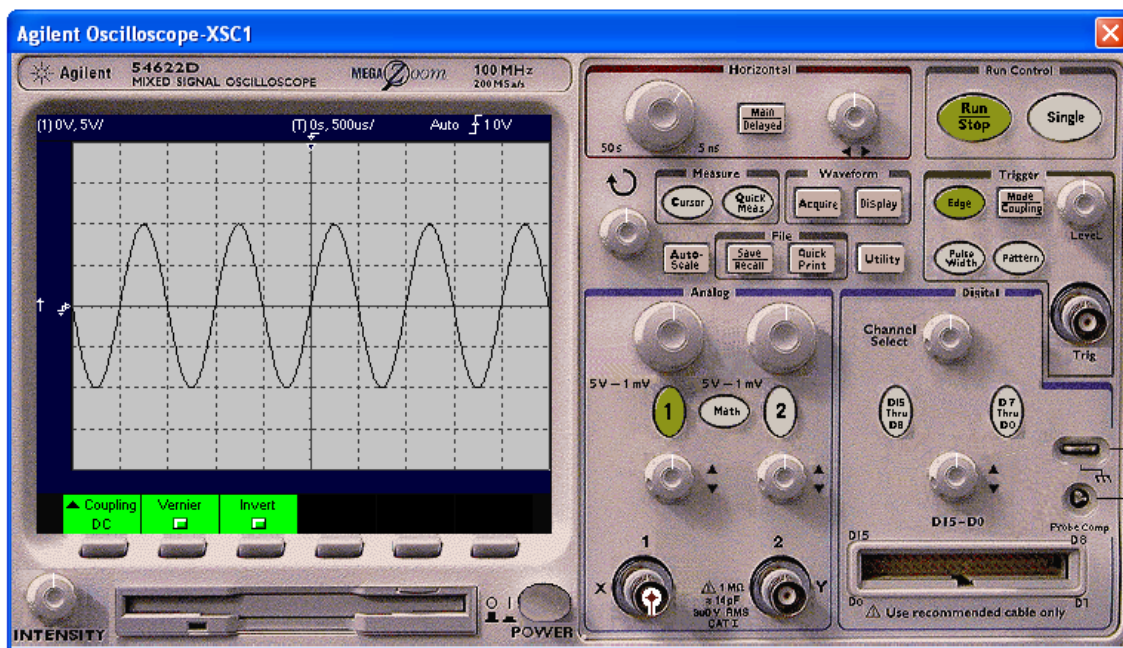


Figure 35 Agilent Oscilloscope Display

The Tektronix oscilloscope icon shown in Figure 36 has all 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 1 through 4, respectively. The Tektronix oscilloscope is controlled via the front panel as seen in Figure 37. The buttons are “pushed” by clicking. The dials can be turned by dragging the mouse over it or by placing the cursor over it and spinning the mouse wheel. The latter is the preferred method.

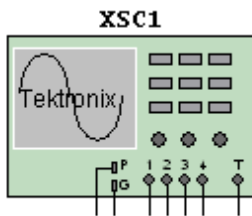


Figure 36 Tektronix Oscilloscope Icon

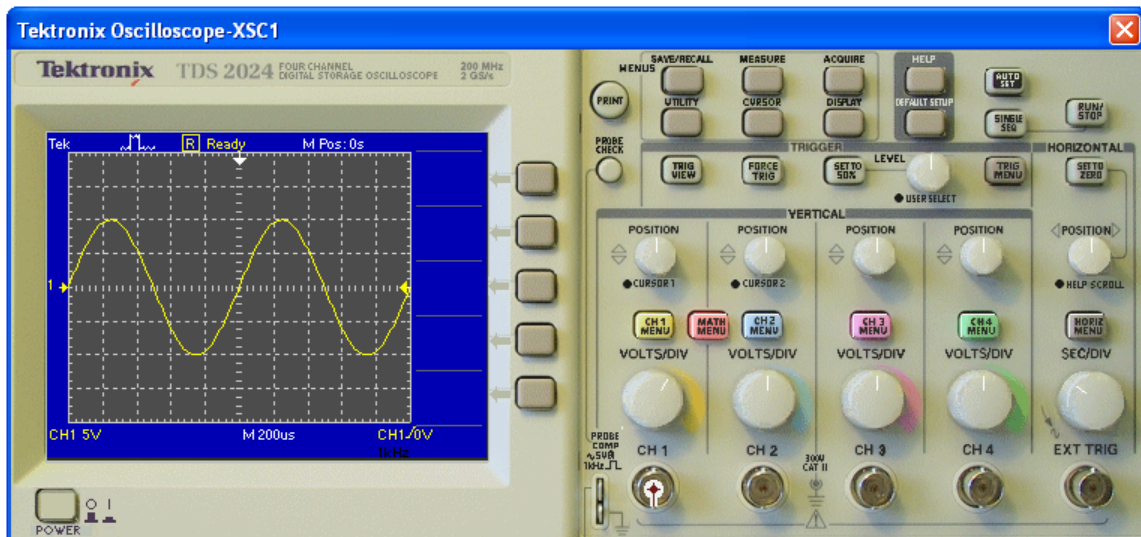


Figure 37 Tektronix Oscilloscope Display

Voltage and Current Meters

Multisim provides simple voltmeters and ammeters, as shown in Figure 38, for use when voltage or current need to be measured. These meters can be placed throughout the

circuit. The meters are available in both vertical and horizontal orientation to match the layout 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 the meter.

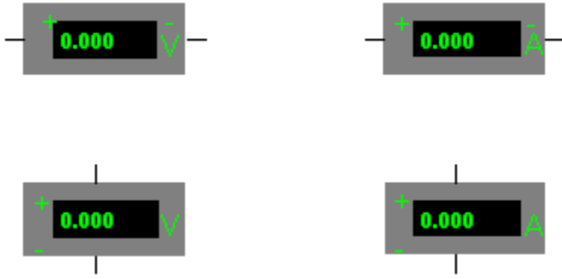


Figure 38 Voltage and Current Meters

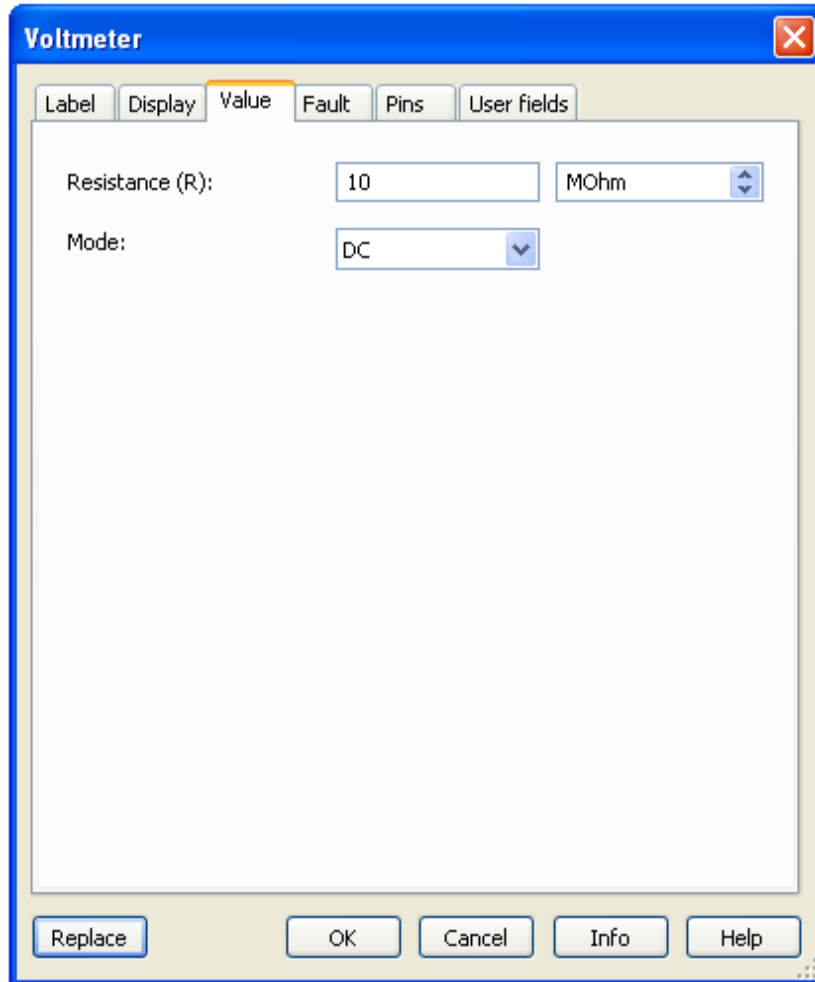


Figure 39 Voltmeter Configuration Screen

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.

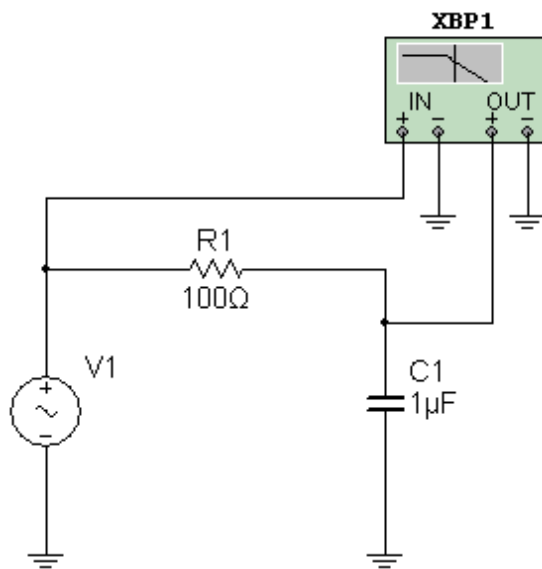
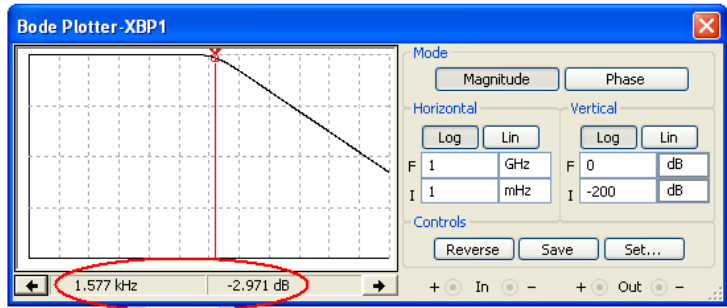
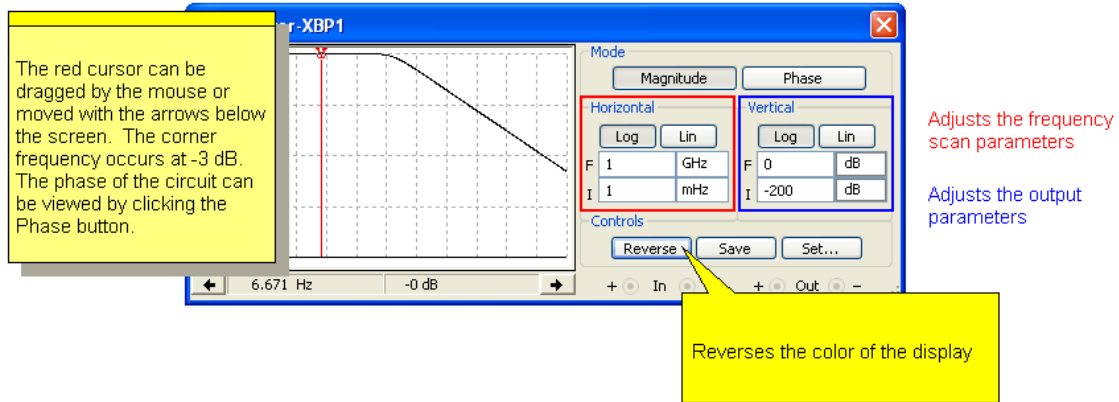


Figure 40 Bode Plotter



The corner frequency is approximately 1.577 kHz.

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 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.

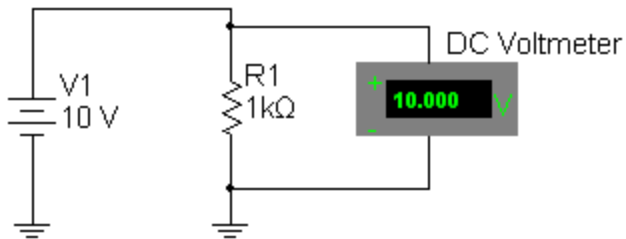


Figure 42 DC Voltage Measurement with a Voltmeter

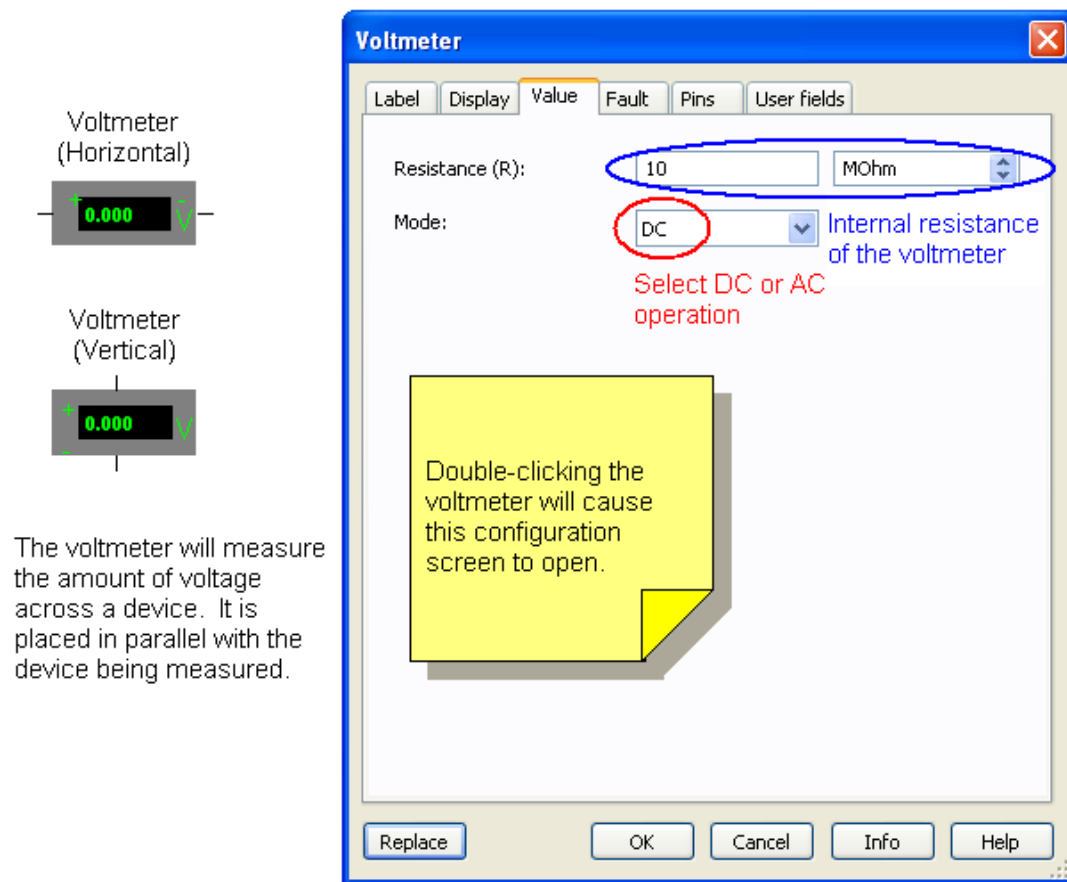


Figure 43 Voltmeter Quick Hint

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.

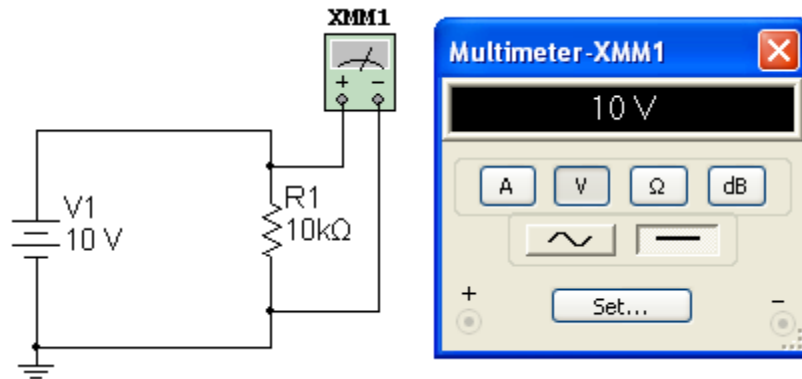
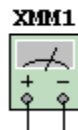
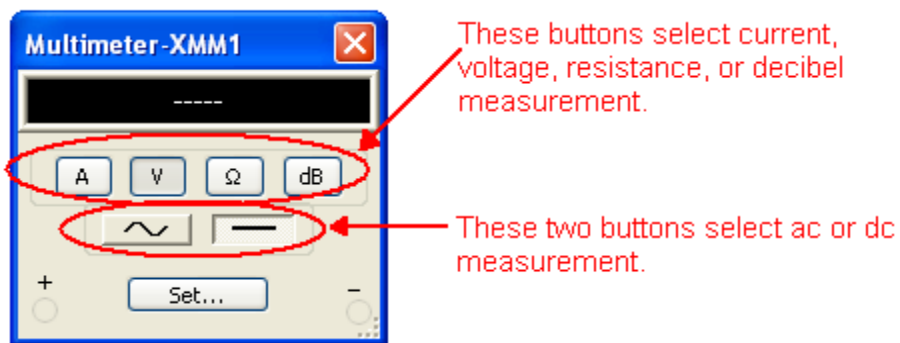


Figure 44 DC Voltage Measurement with a Generic Multimeter

Generic Multimeter Icon



Double-clicking the multimeter icon brings up this display.



These buttons select current, voltage, resistance, or decibel measurement.

These two buttons select ac or dc measurement.

Simply click on the appropriate buttons to set up the multimeter for the type of measurement you wish to make.

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.

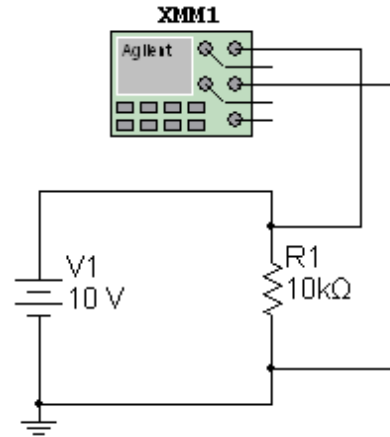
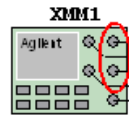


Figure 46 DC Voltage Measurement with an Agilent Multimeter

Agilent Multimeter



These five connections represent the connections on the meter face below. The top two of the right-hand column are used for voltage measurement.

Double-click the Agilent multimeter icon to bring up the meter face.



Selects dc voltage

Be sure to turn on the power to the meter by clicking this button.

The buttons select the type of measurement. Click on a button to "depress" it.

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 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.

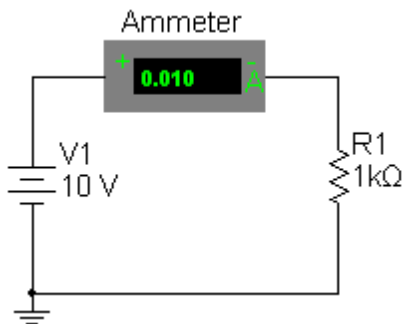


Figure 48 DC Current Measurement with an Ammeter

The image shows two ammeter symbols: 'Ammeter (Horizontal)' and 'Ammeter (Vertical)', both displaying '0.000'. To the right is the 'Ammeter' configuration dialog box. It has tabs for 'Label', 'Display', 'Value', 'Fault', 'Pins', and 'User Fields'. The 'Value' tab is active, showing 'Resistance (R)' set to '1e-009' and 'Ohm', and 'Mode' set to 'DC'. Red circles highlight these two fields, with a red arrow pointing to the 'Resistance (R)' field and the text 'Internal resistance of the ammeter'. Below the dialog box, a yellow note says 'Double-clicking the ammeter will bring up this configuration screen.' At the bottom of the dialog box are buttons for 'Replace', 'OK', 'Cancel', 'Info', and 'Help'.

The ammeter will measure the amount of current flowing through it. The meter will display the appropriate prefix, if needed (m = milli, n = nano, p = pico).

The ammeter is placed in series with the load.

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.

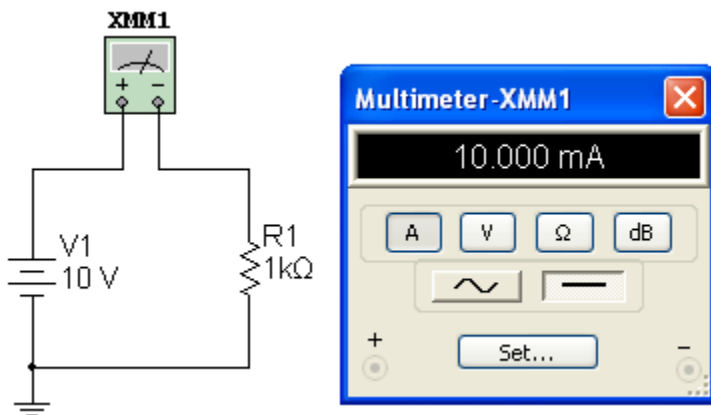
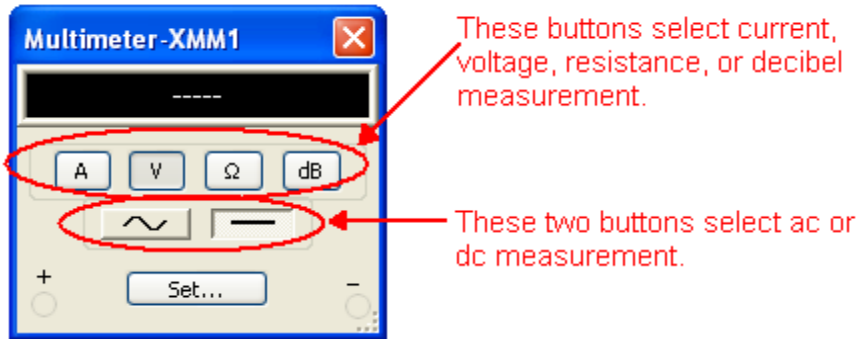


Figure 50 DC Current Measurement with a Generic Multimeter

Generic Multimeter Icon



Double-clicking the multimeter icon brings up this display.



Simply click on the appropriate buttons to set up the multimeter for the type of measurement you wish to make.

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.

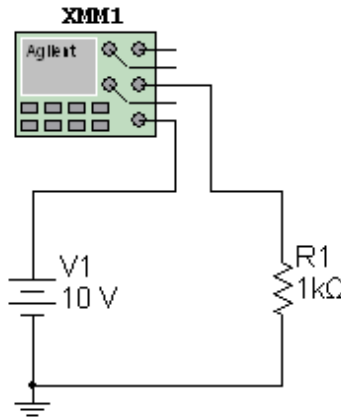
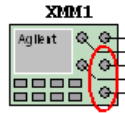


Figure 52 DC Current Measurement with an Agilent Multimeter

Agilent Multimeter



These five connections represent the connections on the meter face below. The bottom two of the right-hand column are used for current measurement.

Double-click the Agilent multimeter icon to bring up the meter face.



The second level selects dc current.

Be sure to turn on the power to the meter by clicking this button.

Pressing the Shift button provides access to the second level of the Function buttons.

The buttons select the type of measurement. Click on a button to "depress" it.

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 the ground as a reference point. This generic oscilloscope's operation follows that of an actual oscilloscope. The oscilloscope display is brought up by clicking 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 Timebase such that a complete cycle or two are displayed. Figures 55, 56, and 57 contain *Quick Hints* on the use of the generic oscilloscope.

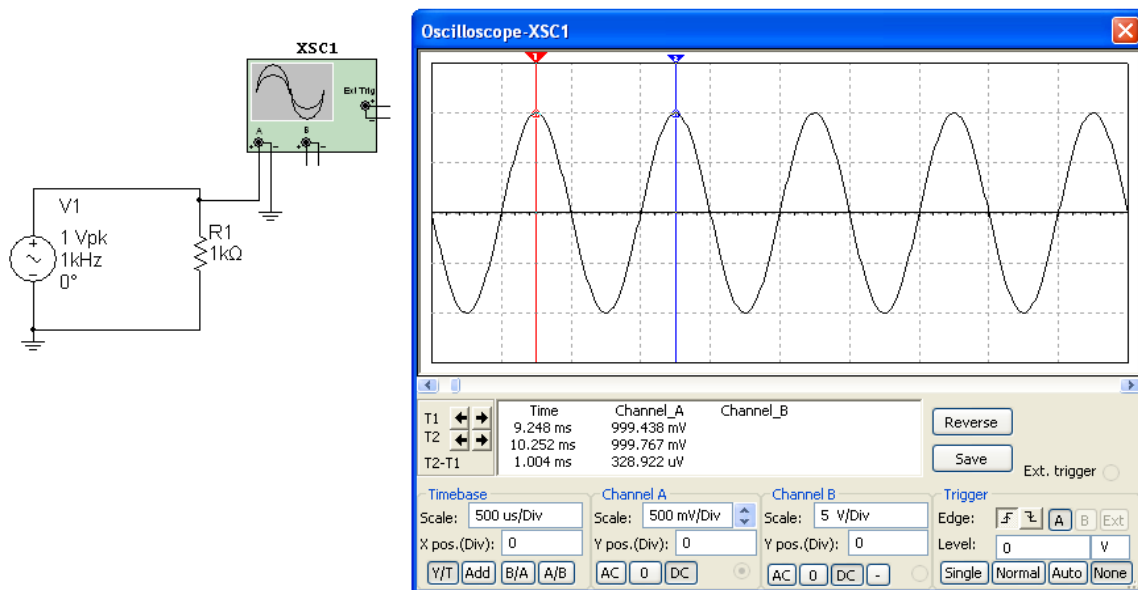


Figure 54 Voltage Measurement with a Generic Oscilloscope

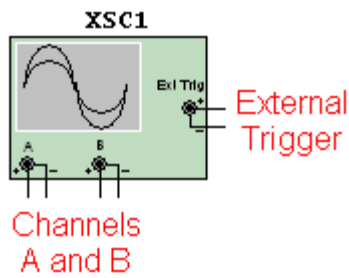
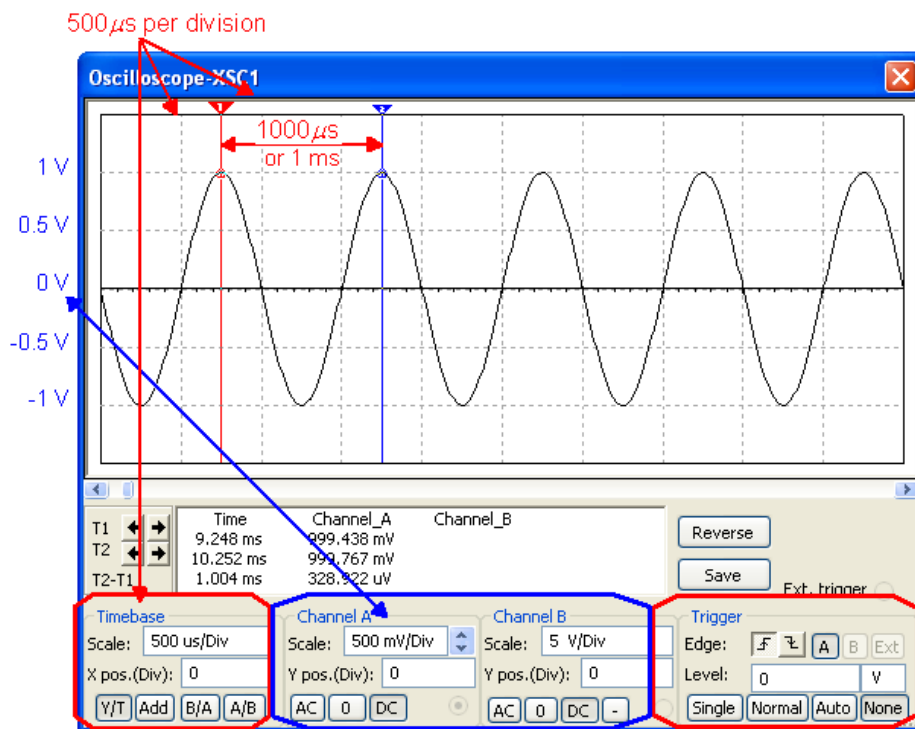


Figure 55 Generic Oscilloscope Icon Quick Hint

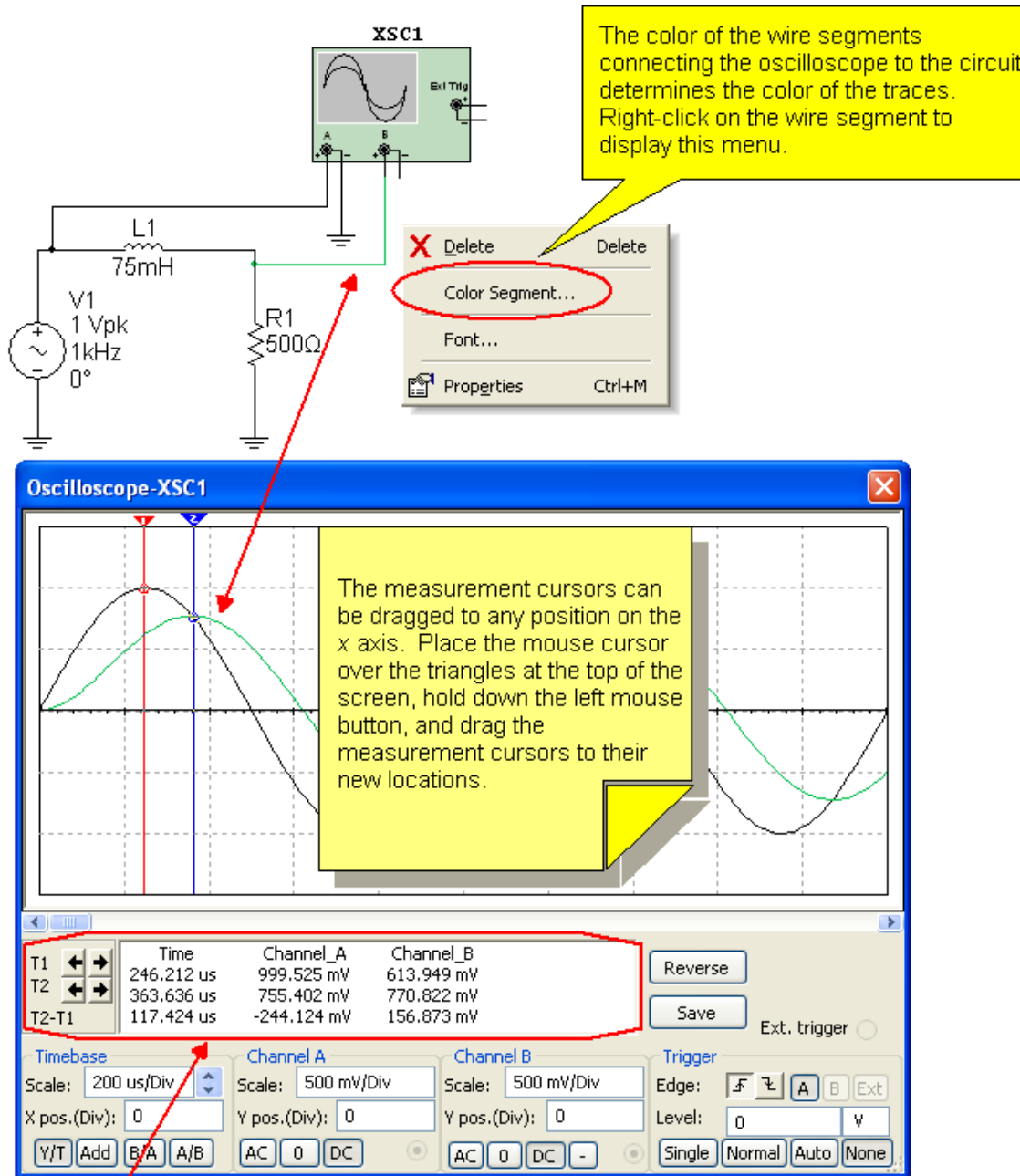


The Timebase controls the units for the x axis. The x axis represents time, and the units are seconds per division.

Channel A and channel B control the units for the y axis. The y axis represents voltage, and the units are volts per division.

The Trigger controls when the oscilloscope starts to display the waveform. The waveform can be displayed on either the rising or falling edge.

Figure 56 Generic Oscilloscope Quick Hint



This box and data are from the measurement cursors (1 and 2). The voltage level and time are measured at the point where the waveform passes through the measurement cursor.

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 clicking 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 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 Timebase 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.

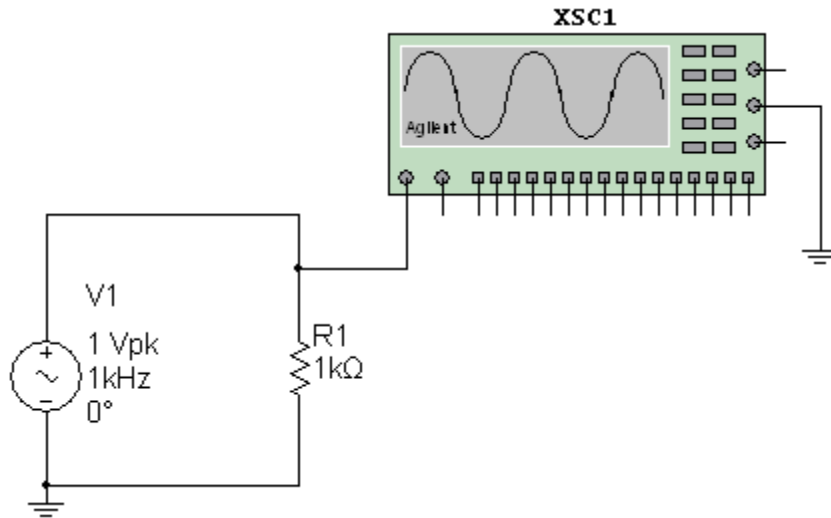


Figure 58 Voltage Measurement with an Agilent Oscilloscope

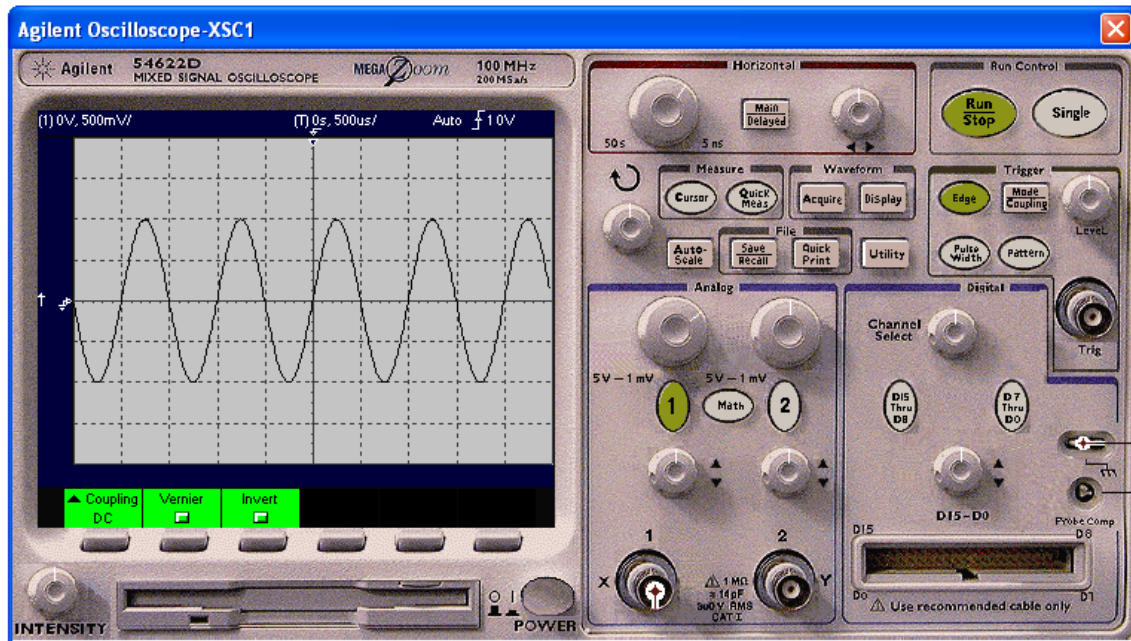


Figure 59 Agilent Oscilloscope Display

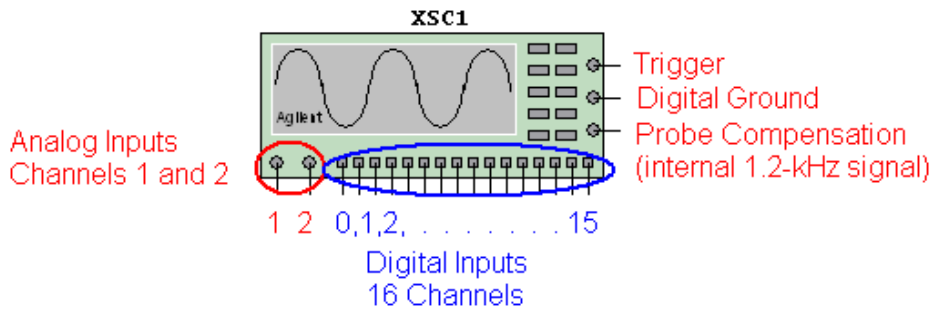
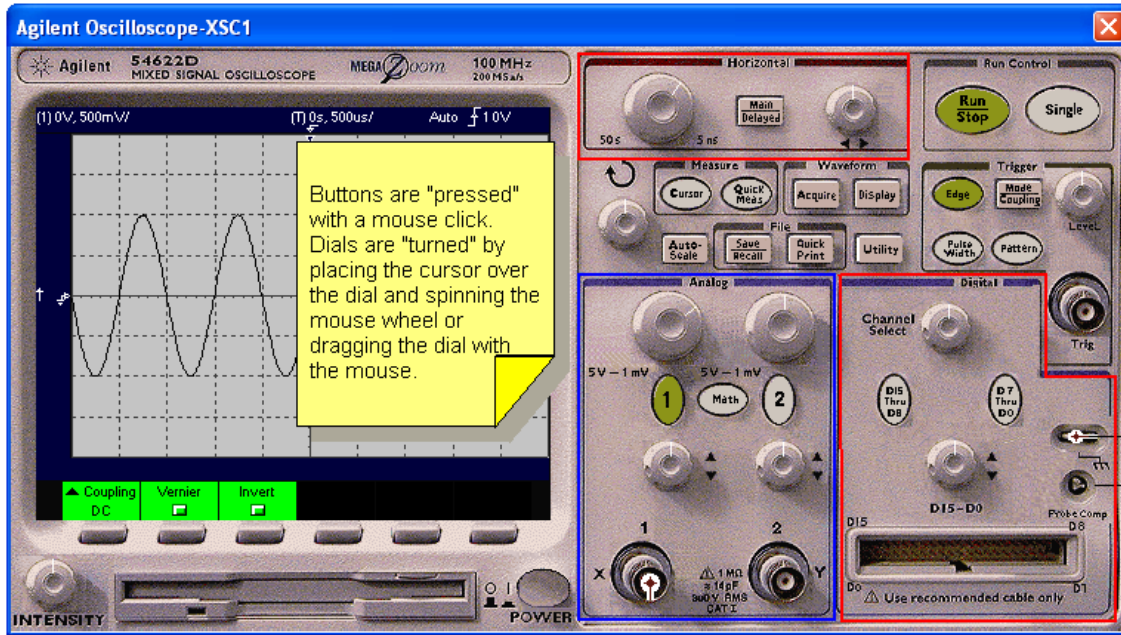


Figure 60 Agilent Oscilloscope Icon Quick Hint

The Horizontal section controls the units for the x axis. The x axis represents time and the units are seconds per division.

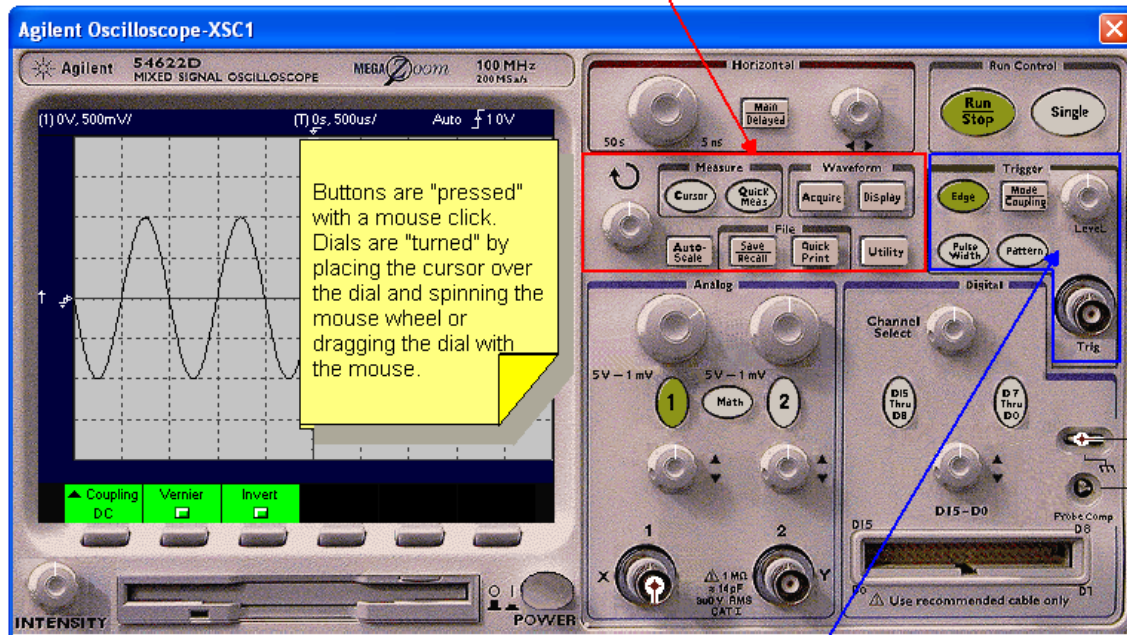


Channel's 1 and 2 control the units for the y axis. The y axis represents voltage, and the units are in volts per division.

16-Channel Digital Logic Input

Figure 61 Agilent Oscilloscope Quick Hint

The Measure section included cursor operation.
The Waveform section allows waveform storage.
The File section saves, recalls, and prints.



The Trigger controls when the oscilloscope starts to display the waveform. It can display it on the rising edge or the falling edge. The voltage level at which the oscilloscope is triggered is also selected here.

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 clicking the oscilloscope icon. The settings can be changed by placing the mouse over the dials and spinning the mouse wheel or by clicking the buttons. Adjust the volts per division on the channel under measurement until the amplitude of the waveform fills the majority of the screen. Adjust the Timebase 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.

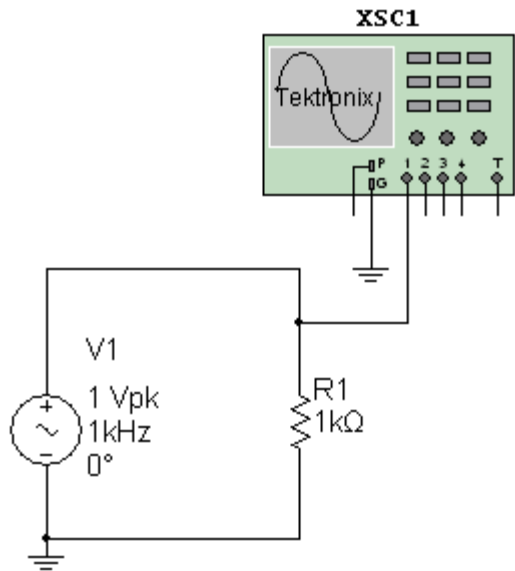


Figure 63 Voltage and Frequency Measurement with a Tektronix Oscilloscope

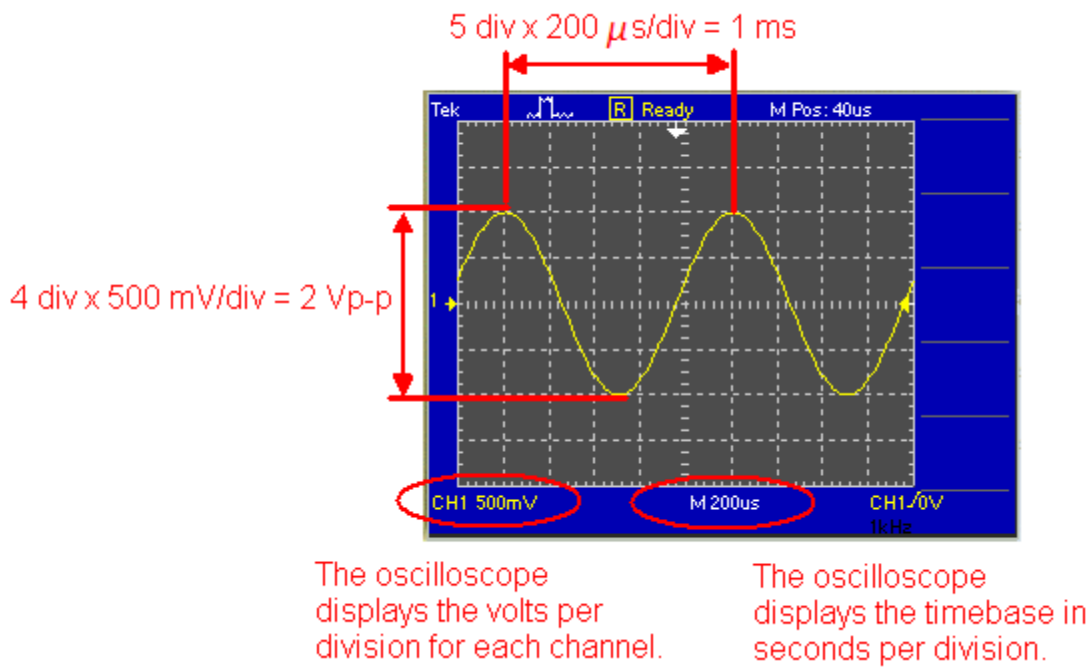


Figure 64 Measurement of the Period of the Waveform

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.

$$V_p = 2 \text{ divisions} \times \frac{500 \text{ mV}}{\text{division}}$$

$$V_p = 1 \text{ V}$$

$$V_{p-p} = 4 \text{ divisions} \times \frac{500 \text{ mV}}{\text{division}}$$

$$V_{p-p} = 2 \text{ V}$$

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

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

$$f = \frac{1}{1 \text{ ms}}$$

$$f = 1 \text{ kHz}$$

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: V_{p-p} , frequency, etc.
4. Return to the main screen.
5. Repeat for other channels and or values.

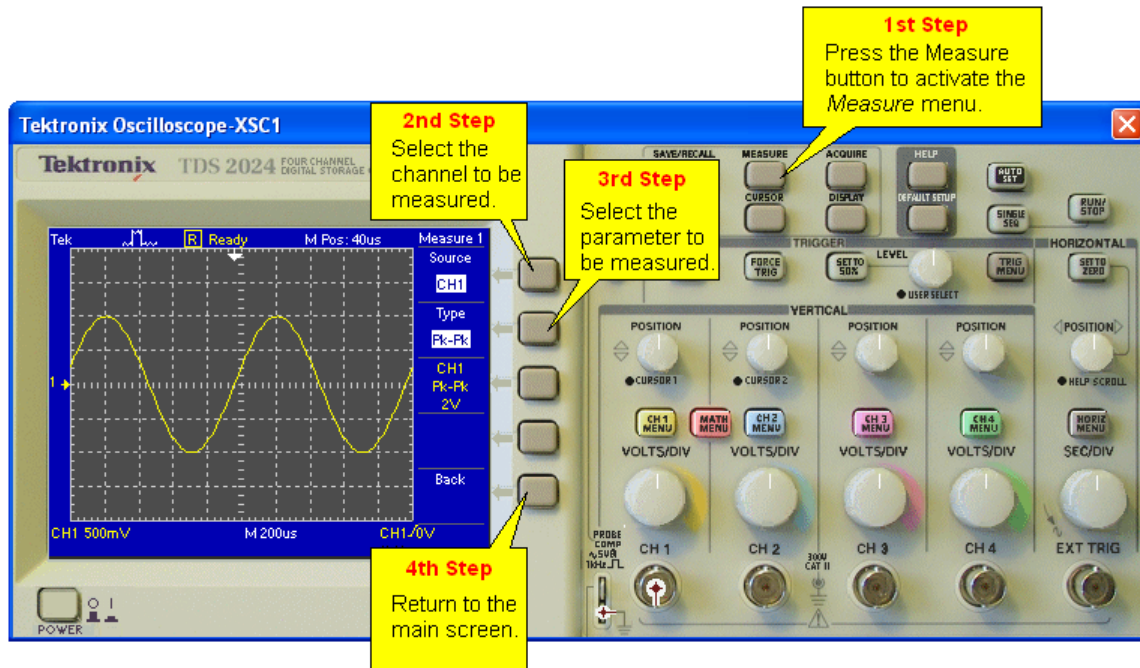


Figure 65 Tektronix Oscilloscope Measurement Function Setup

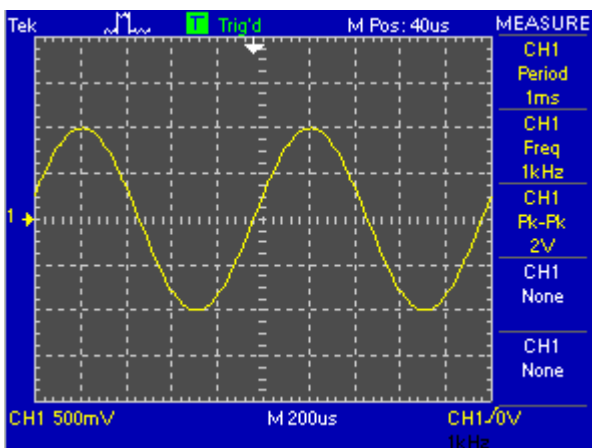


Figure 66 Tektronix Oscilloscope Measurement Display

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

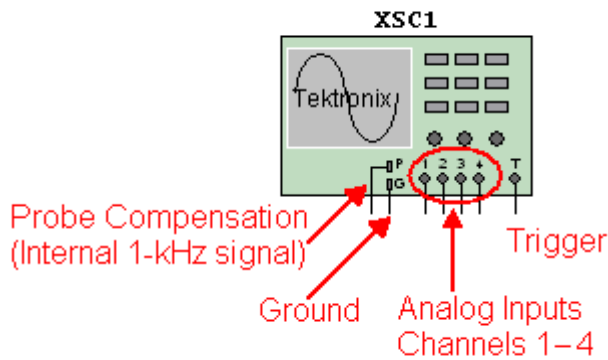


Figure 67 Tektronix Oscilloscope Icon Quick Hint

Hint: Buttons are "pressed" with a mouse click. Dials are "turned" by placing the cursor over the dial and spinning the mouse wheel or dragging the dial with the mouse.

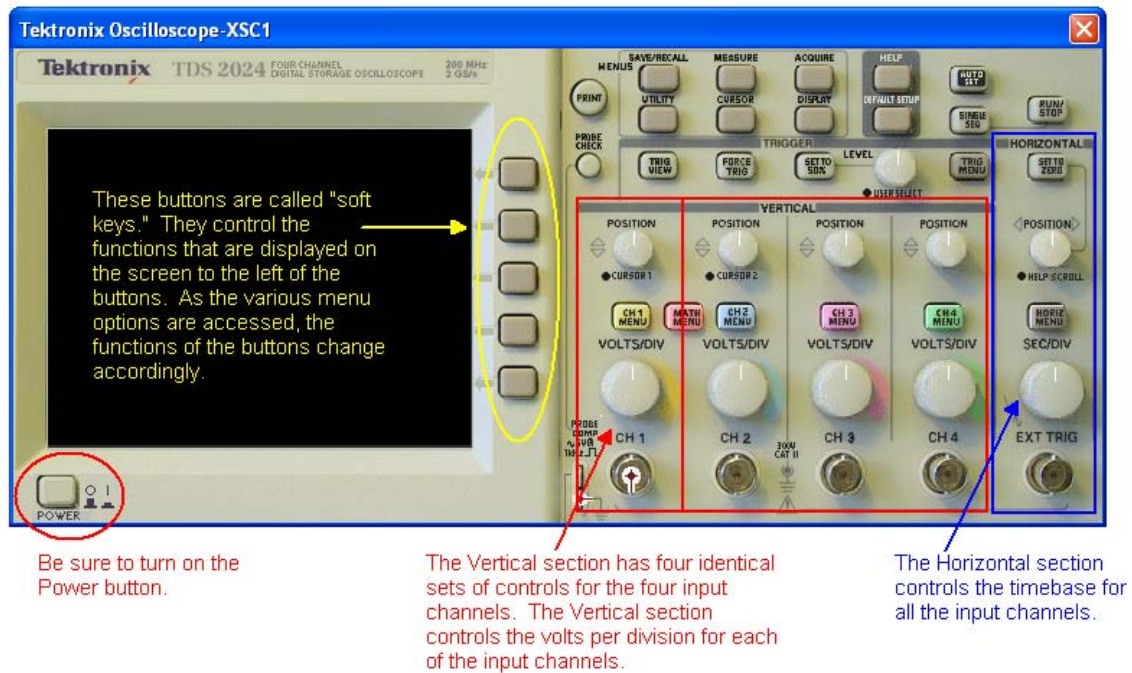


Figure 68 Tektronix Oscilloscope Quick Hint

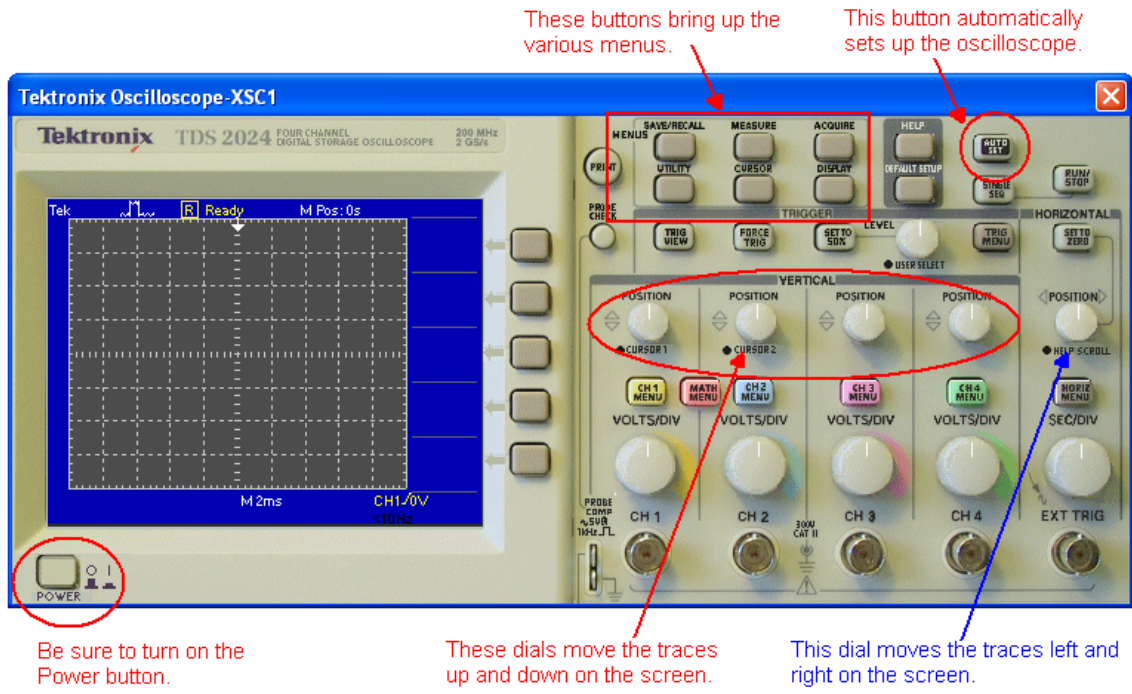


Figure 69 Tektronix Oscilloscope Quick Hint

User Customization

There are several changes to the base configuration of Multisim that will make it easier for you to use. Multisim uses a grid system to align the various components in the work area. When the program is first installed, the grid will be visible in the work area. It will appear as a pattern of dots, as shown in Figure 70.

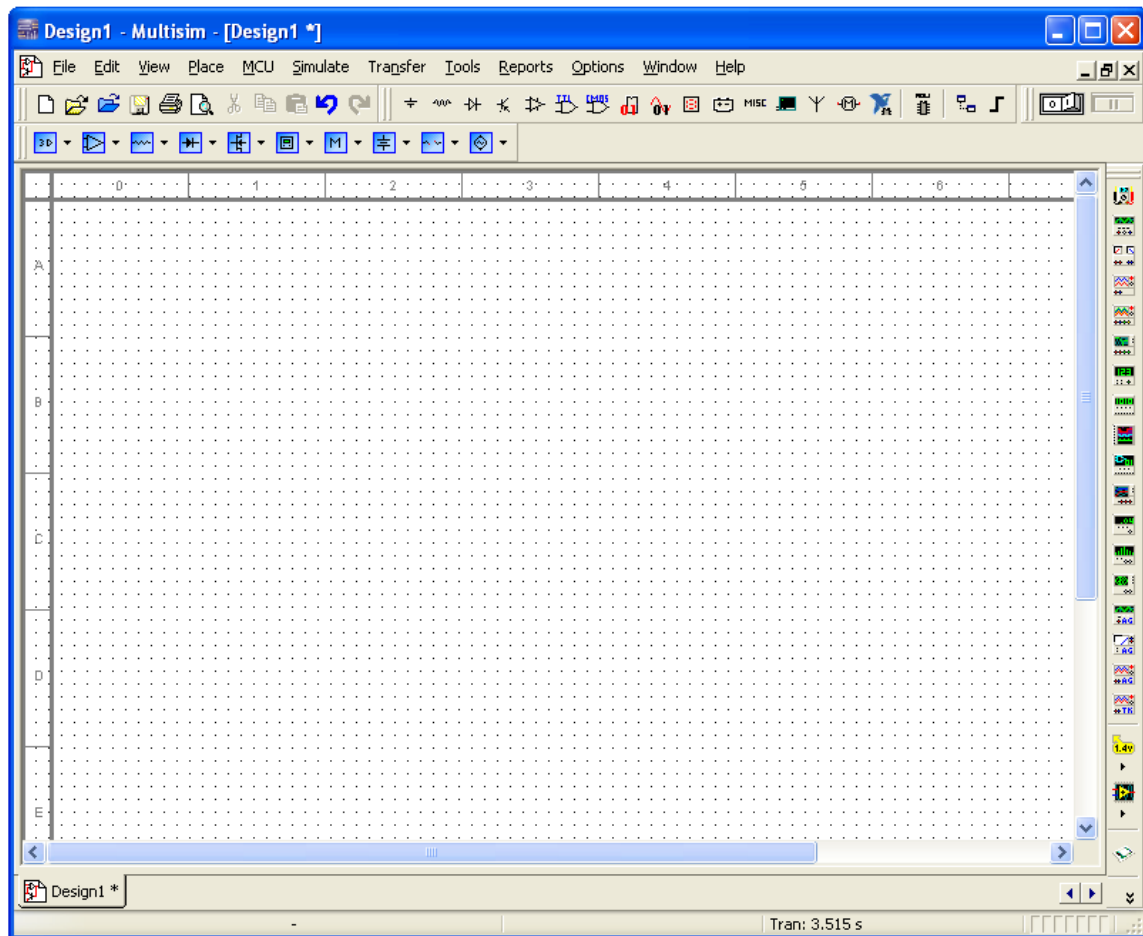


Figure 70 The Work Area with the Grid Displayed

The grid pattern can be turned off in the *Sheet Properties* submenu of the *Options* menu. The grid will still be used to align the components; however, the dots will not be visible.

To access the *Sheet Properties* submenu, either press Alt + O or click *Options* on the menu bar, as shown in Figure 71.

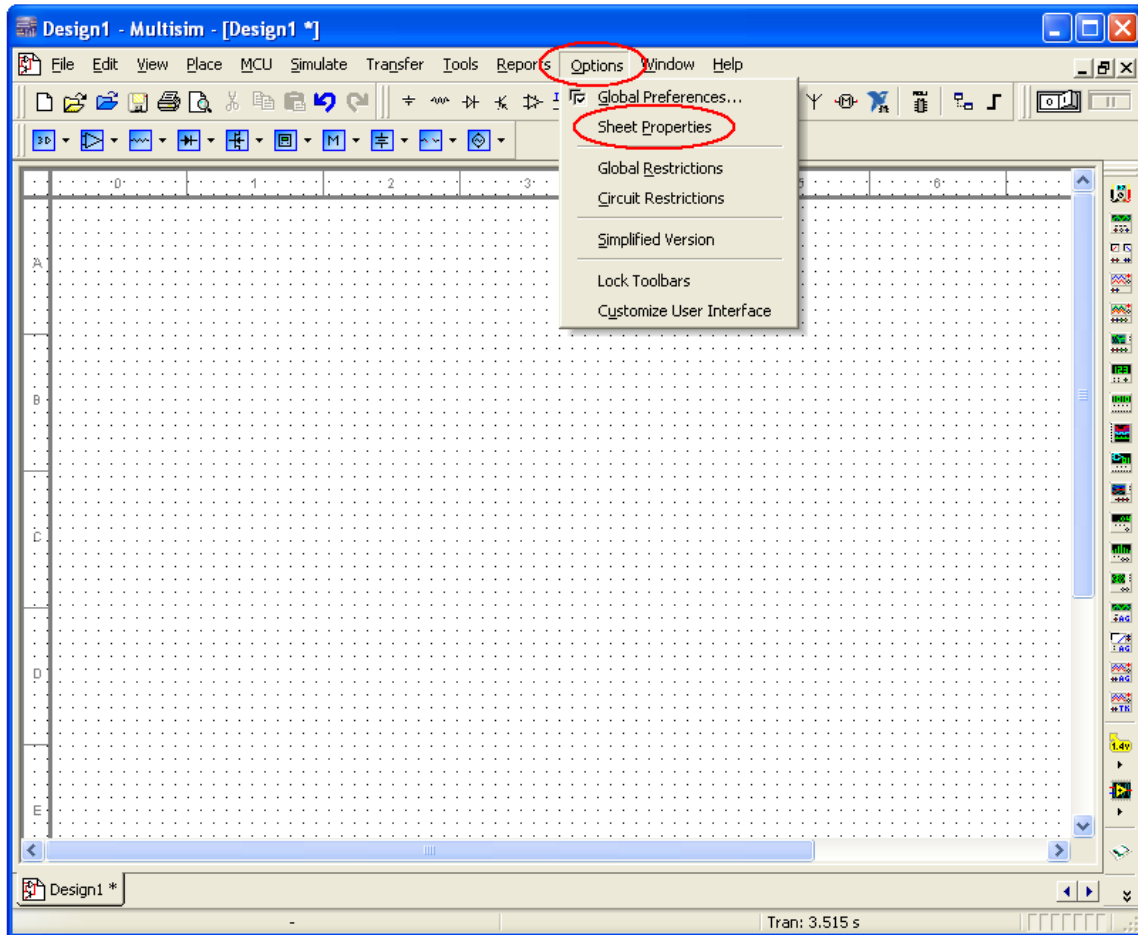


Figure 71 Access to the *Sheet Properties* Submenu

The *Sheet Properties* submenu shown in Figure 72 will allow the user to control the work area environment. The *Workspace* tab gives the user access to the grid controls, the sheet size, and orientation.

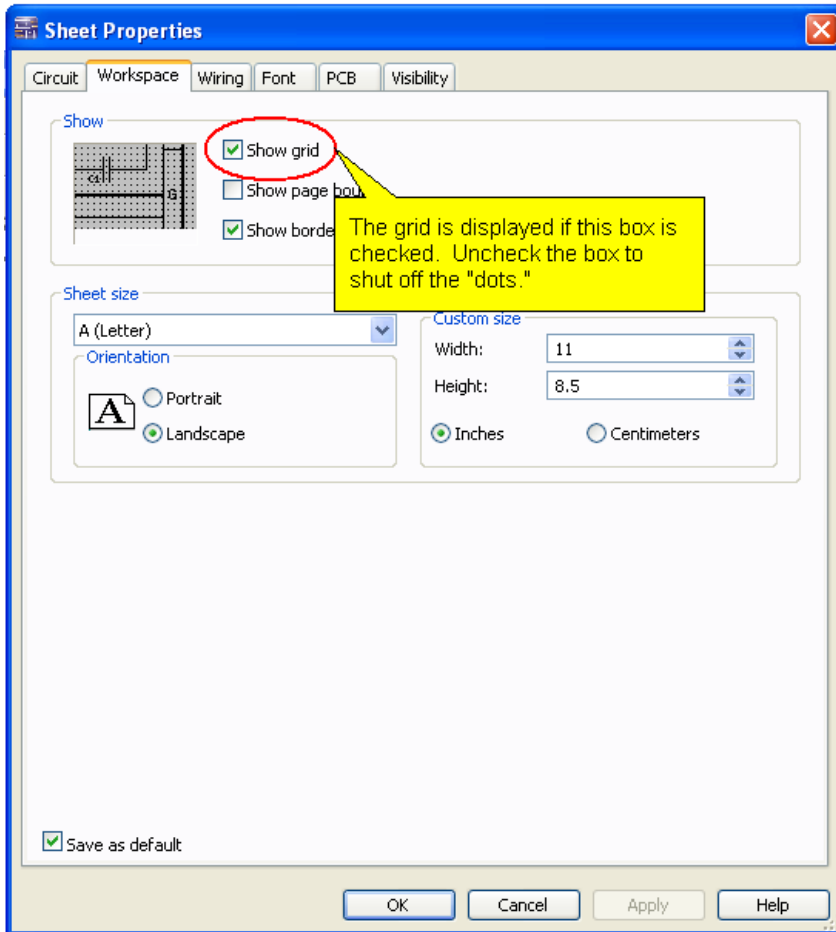


Figure 72 *Workspace Tab of the Sheet Properties Submenu*

The initial configuration of Multisim will display the Net Names within the circuit. The Net Names are the numbers circled in red in Figure 73.

These are the Net Names.

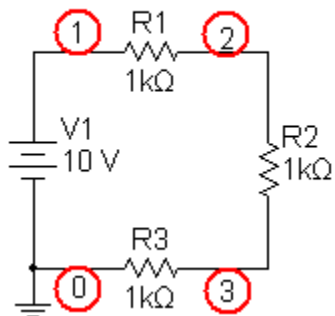


Figure 73 *Net Names Displayed*

If the Net Names are a distraction, they can be turned off from within the *Circuit* tab of the *Sheet Properties* submenu, as shown in Figure 74. In addition to controlling the display of the Net Names, this tab gives the user access to which component parameters are displayed at the global level and the color of the various components and wiring. A custom color scheme can be defined on this screen as well.

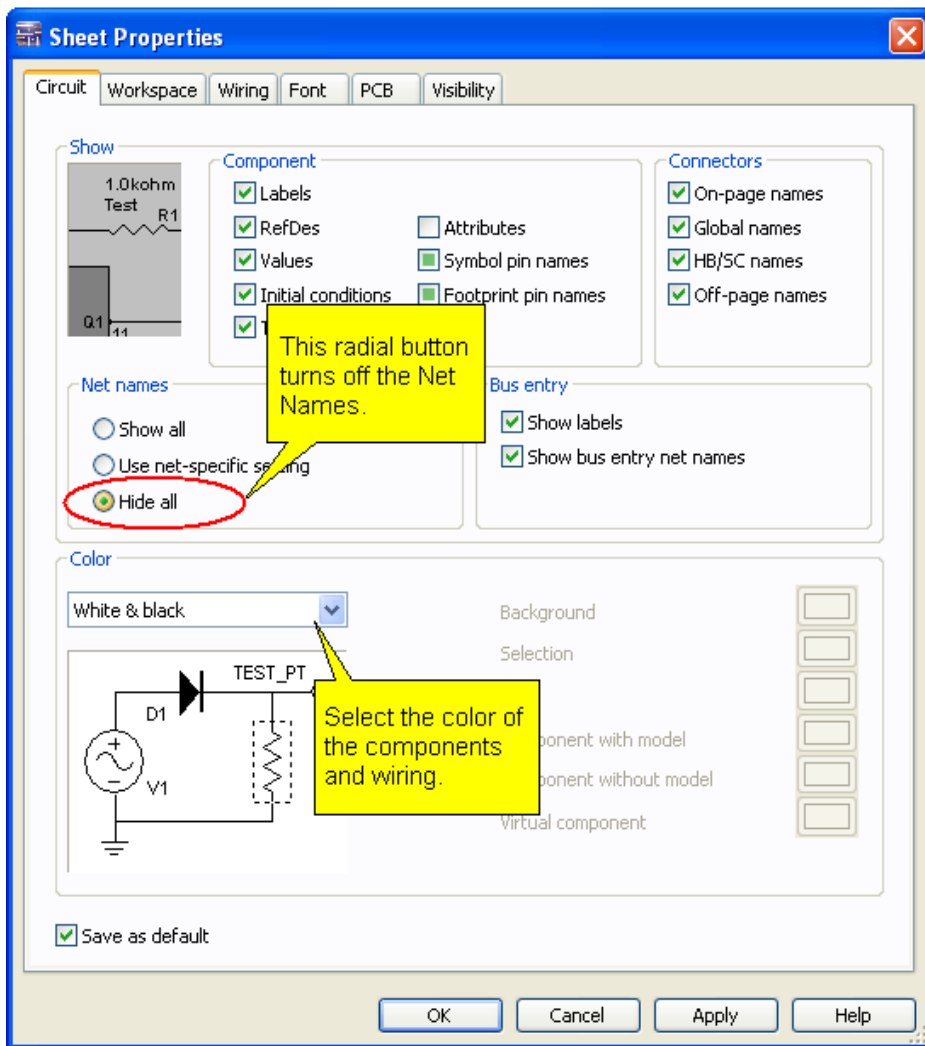


Figure 74 The *Circuit* Tab of the *Sheet Properties* Submenu

Exporting Data to Excel

The Bode plotter and the oscilloscope were introduced in the section, “Measurement Equipment.” Although the displays of the Bode plotter and the oscilloscope are quite informative, it may be useful to have the data in numerical form for further analysis.

After the Bode plotter simulation is run or a waveform is viewed on the oscilloscope, the data can be viewed in the Grapher. The command to initiate Grapher is on the *View* menu. To access the *View* menu, either press Alt + V or click *View* in the menu bar, as shown in Figure 75.

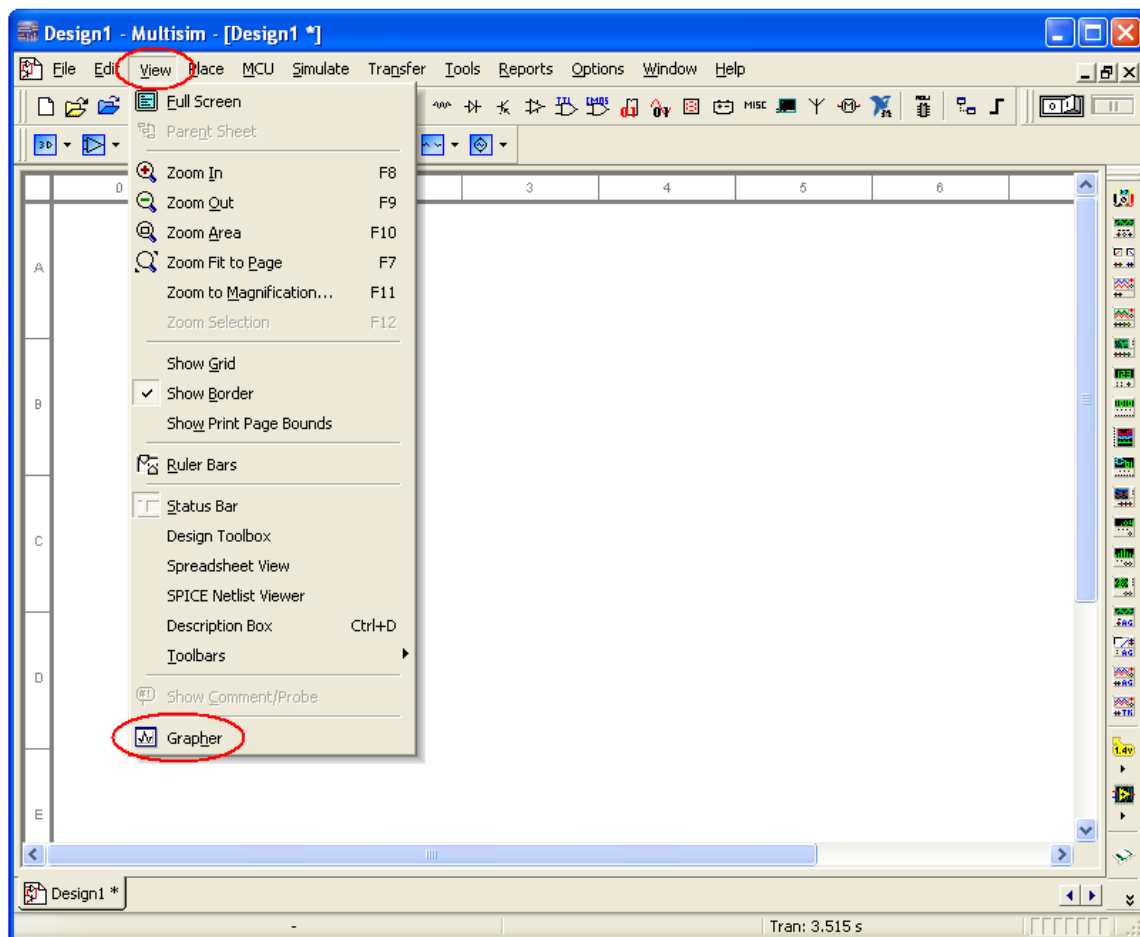


Figure 75 Accessing the Grapher via the *View* Menu

If Excel is also loaded on the computer, the data used to generate the plot in the Grapher can be exported to an Excel worksheet for further analysis. The plot to be exported must be selected by clicking it. The red indicator arrow on the Grapher screen, as shown in Figure 76, points to the plot to be exported.

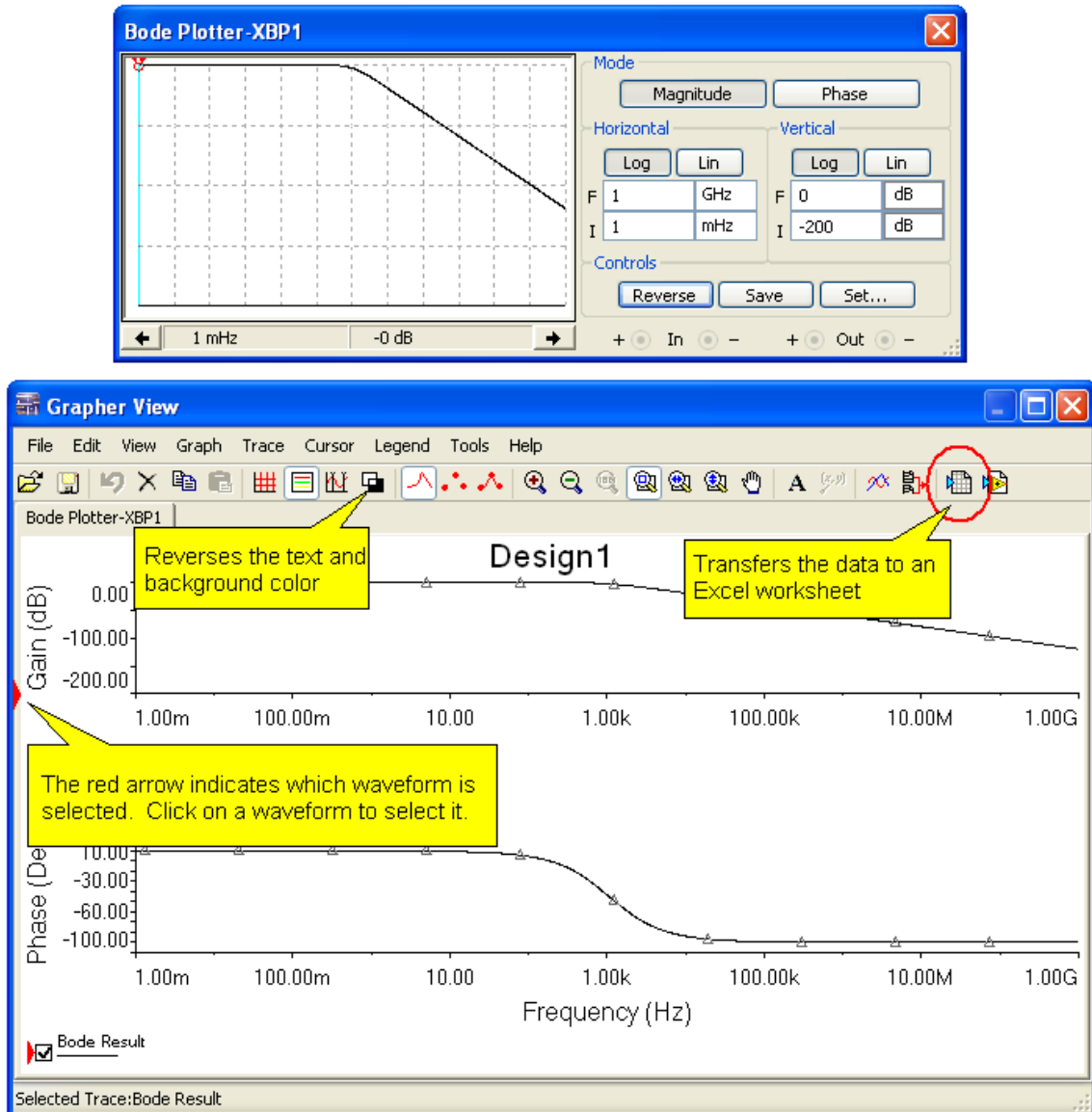


Figure 76 The Grapher

Once the plot is selected, the data can be exported to an Excel worksheet by clicking the Excel icon in the upper right-hand side of the toolbar, as shown in Figure 76. The export feature will create a new worksheet with the x and y data in adjacent columns. Data from the generic oscilloscope were captured by the Grapher in Figure 77.

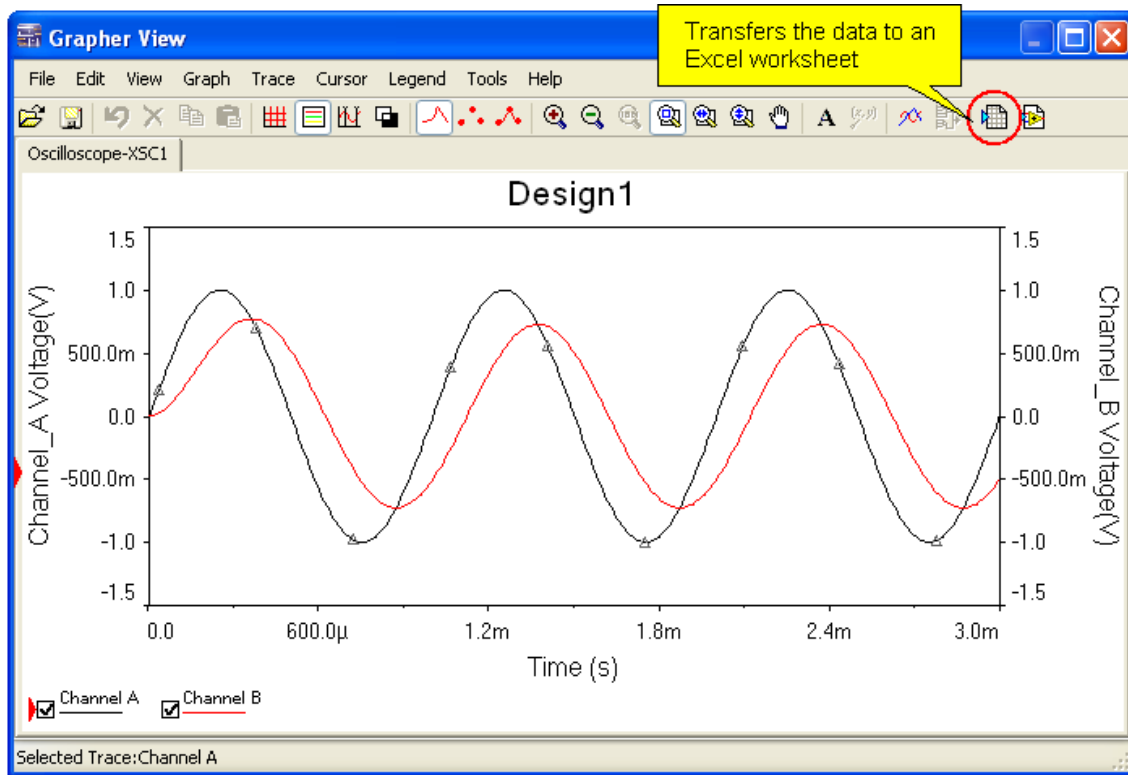


Figure 77 Exporting Oscilloscope Data to Excel

After the export to Excel icon is clicked, Multisim will prompt the user for the desired trace to be exported, as shown in Figure 78. As part of the export process, Excel will be started, and a new worksheet will be created with the data from the x axis in column A and the data from the y axis in column B. The data used to create the waveform within Multisim can now be examined using the analytical capabilities of Excel.

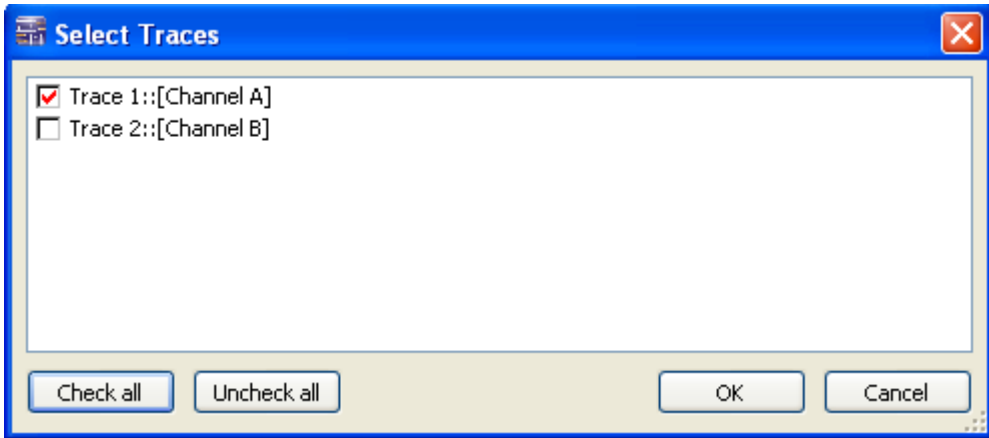


Figure 78 Selecting the Trace for Export

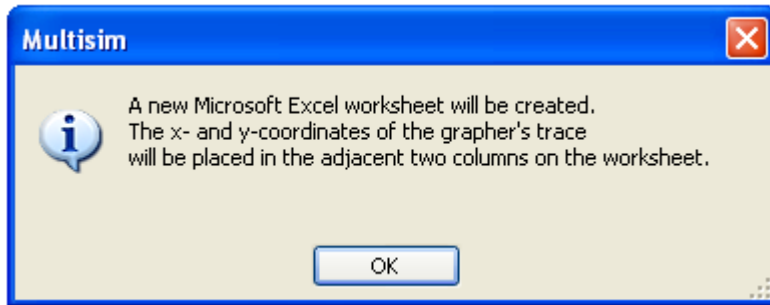


Figure 79 Creating a New Excel Worksheet

| | A | B |
|----|-------------------------|-------------------------|
| 1 | X--Trace 1::[Channel A] | Y--Trace 1::[Channel A] |
| 2 | 0 | 0 |
| 3 | 0.0000001 | 0.000628318 |
| 4 | 1.08404E-07 | 0.000681121 |
| 5 | 1.25211E-07 | 0.000786725 |
| 6 | 1.58826E-07 | 0.000997934 |
| 7 | 2.26056E-07 | 0.001420351 |
| 8 | 3.60516E-07 | 0.002265186 |
| 9 | 6.29435E-07 | 0.003954849 |
| 10 | 1.16727E-06 | 0.007334136 |
| 11 | 2.24295E-06 | 0.01409242 |
| 12 | 4.39431E-06 | 0.027606749 |
| 13 | 8.69702E-06 | 0.054617806 |
| 14 | 1.73024E-05 | 0.108500459 |
| 15 | 2.73024E-05 | 0.170706187 |
| | 3.73024E-05 | 0.232238215 |

Figure 80 The Excel Worksheet with Exported Data

Adding Text and Graphics

Simply constructing the circuit and running the simulation are only part of the overall process. It is also very important to create circuits that include information that clearly explains what is taking place. Both annotation and graphics can be added to a circuit schematic to aid in the understanding of it by the viewer. To add text to the work area, either press Ctrl + Alt + A or click *Place* on the menu bar, as shown in Figure 81. Move the cursor to the location within the work area where the text should begin, and click once. To exit the “text edit” mode after the text has been entered, click anywhere else on the work area. The text can be edited by double-clicking the text. The text can be repositioned in the work area by placing the cursor over the text, holding down the left mouse button, and dragging the text its new location.

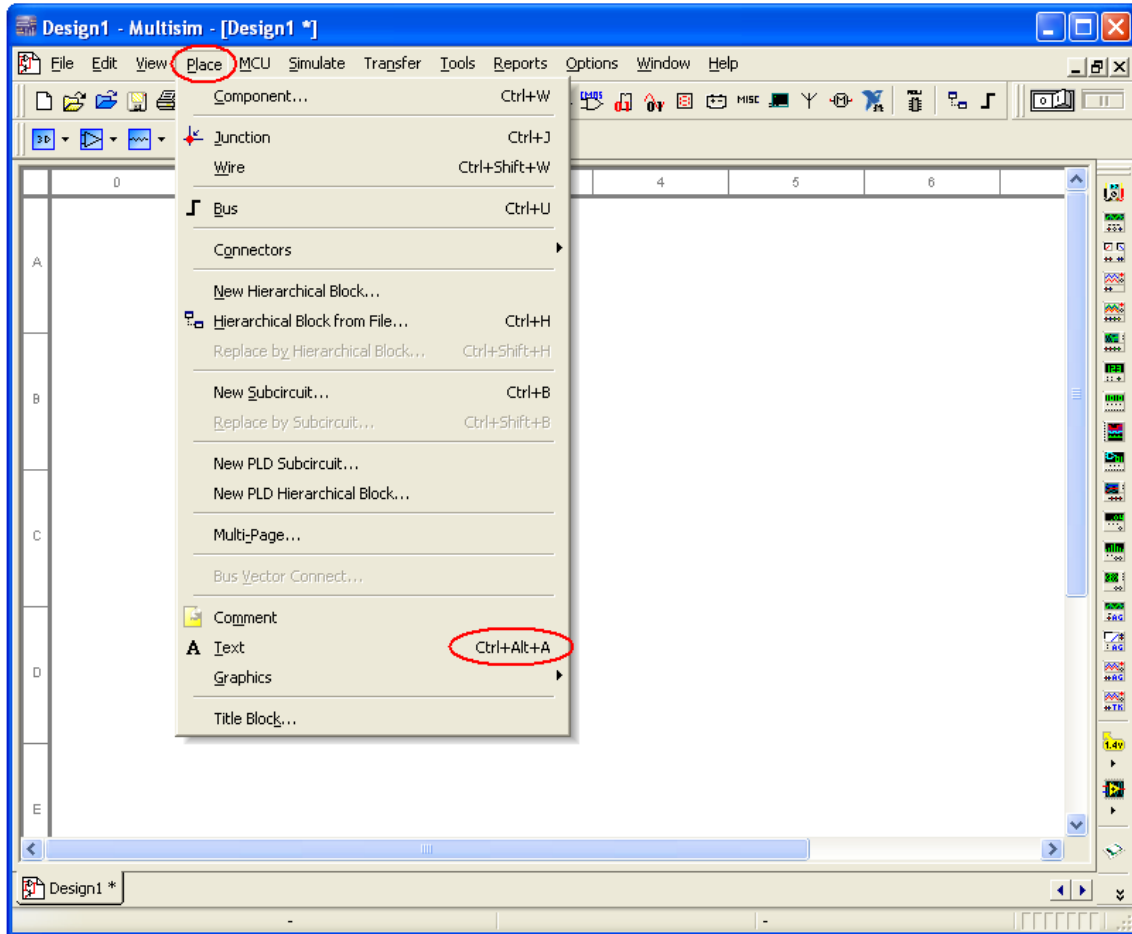


Figure 81 Placing Text in the Work Area

Basic drawing tools are available to annotate the work area. The option to import a picture into the work area is also accessible from the *Graphics* dialog box. As shown in Figure 82, click *Place*, which is located on the menu bar. Then select *Graphics* from the drop-down menu. This will cause the *Graphics* dialog box to open. The default file format for pictures to be inserted into the drawing is bitmap (.bmp).

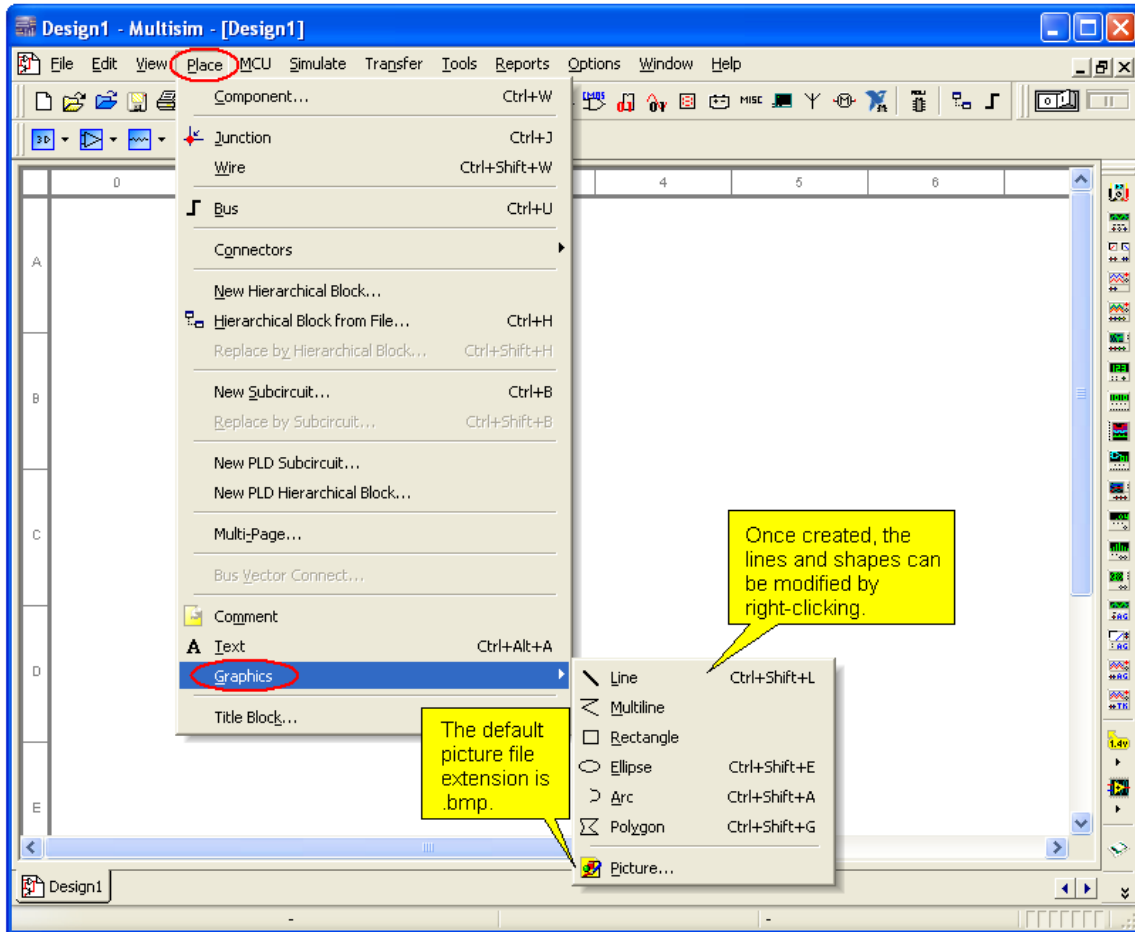


Figure 82 Placing Graphics in the Work Area

Once the lines or shapes are drawn, they can be modified or repositioned. A right-click on a line or shape will cause the edit menu to open as shown in Figure 83. The properties of the object can be changed in the edit menu. The line or shape can be repositioned by placing the cursor over the object and holding the left mouse button down while it is dragged to its new location.

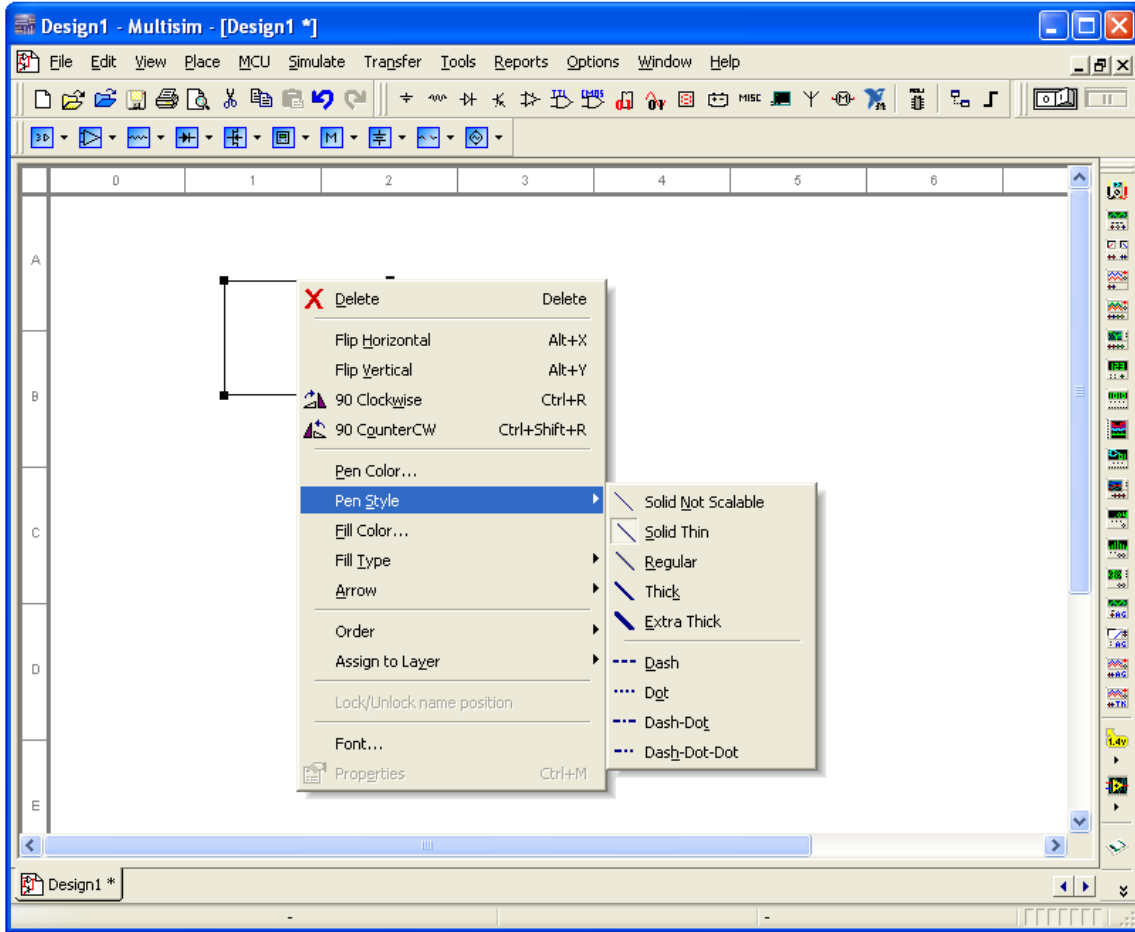


Figure 83 Modifying Graphics in the Work Area

The text and graphics tools within Multisim provide the ability to add graphics and text to the work area. This feature enhances the readability of the schematic, especially when it is going to be used in a laboratory report. Figure 84 provides an example of an annotated schematic that could be used within a laboratory report. Notice how the addition of the color, text, and graphics helps document what is being measured within the circuit.

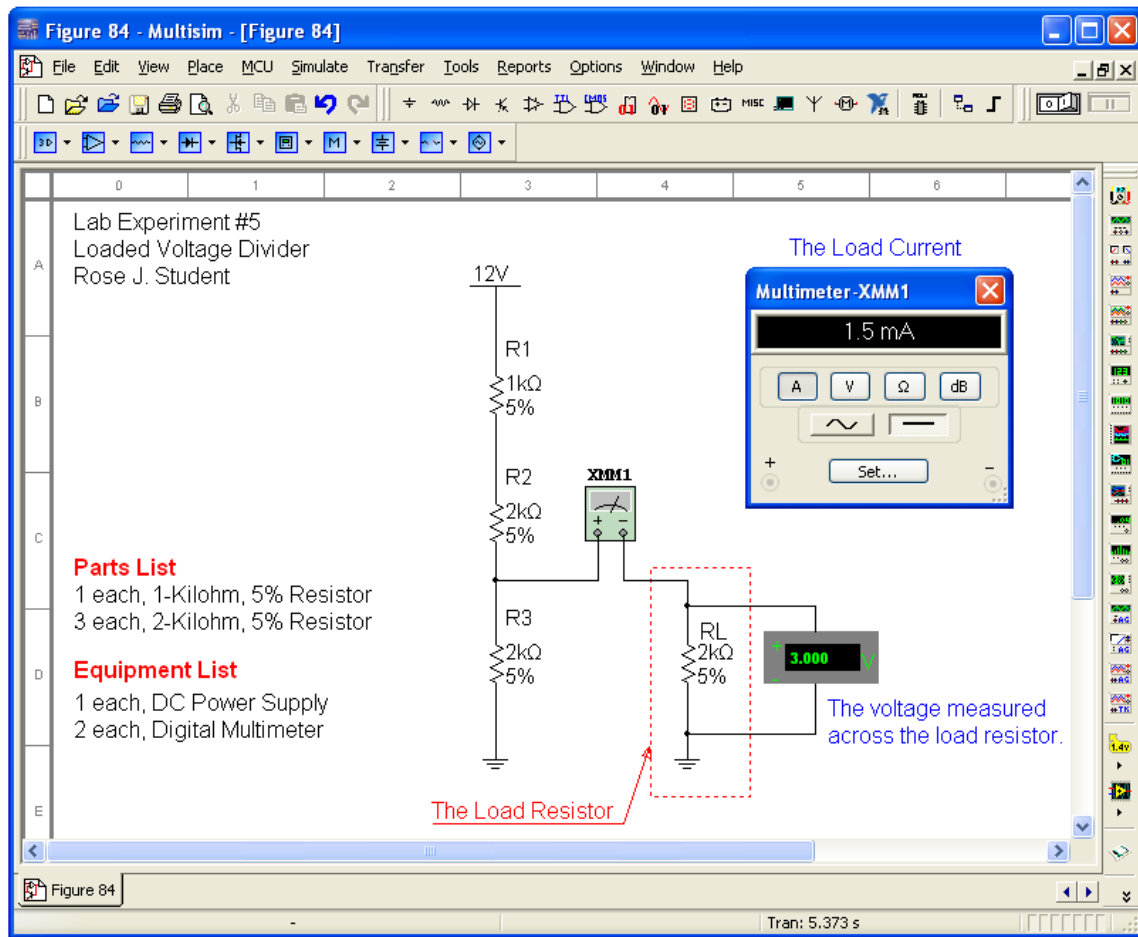


Figure 84 The Use of Annotation and Graphics in the Work Area

Conclusion

Multisim is an interactive circuit simulation package that allows users to view their circuit in schematic form while measuring the various parameters of the circuit. The ability to create a schematic quickly and then analyze the circuit through simulation makes Multisim a wonderful tool for students studying the concepts covered in this textbook.