

ISBN 1-55169-085-3

© 1999 Interactive Image Technologies Ltd. All rights reserved. Published July 1999. Printed in Canada.

Preface

Congratulations on choosing Multisim from Electronics Workbench. We are confident that it will deliver years of increased productivity and superior designs.

Electronics Workbench is the world's leading supplier of circuit design tools. Our products are used by more customers than those of any other EDA vendor, so we are sure you will be pleased with the value delivered by Multisim, and any other Electronics Workbench products you may select.

Documentation Conventions

When Multisim manuals refer to a toolbar button, an image of the button appears in the left column.

The manuals show circuits in black and white, although Multisim is configured to use color by default. (You can customize these to your preferred settings.)



When you see this icon, the functionality described is only available in certain versions of Multisim, or to users who have purchased add-in modules.

Multisim manuals use the convention **Menu/Item** to indicate menu commands. For example, **File/Open** means choose the **Open** command from the **File** menu.

Multisim manuals use the convention of an arrow (➤) to indicate procedural information.

The Multisim Documentation Set

Multisim documentation consists of this *Getting Started and Tutorial* manual, a *User Guide*, and on-line help. All Multisim users receive Adobe Acrobat™ PDF versions of the *Getting Started and Tutorial* manual and the *User Guide*. Depending on your version of Multisim, you may also receive a printed version of the manuals.

Getting Started and Tutorial

The *Getting Started and Tutorial* manual introduces you to the Multisim interface. It also offers an introductory tutorial that takes you through the stages of circuit design, simulation, analysis and reporting.

User Guide

The *User Guide* describes Multisim and its many functions in detail. The manual is organized based on the stages of circuit design, and explains all aspects of using Multisim, in detail.

On-Line Help

Multisim offers a full help file system to support your use of the product. Choose **Help/Multisim Manual** to display the help file that explains the Multisim program in detail, or choose **Help/Multisim Help** to display the help file that contains reference material (from the printed appendices) such as details on all the components families provided with Multisim. Both are standard Windows help files, offering a table of contents and index.

In addition, you can display context-sensitive help by pressing F1 from any command or window, or by clicking the **Help** button on any screen that offers it.

Adobe PDF Files

Both the *Getting Started and Tutorial* manual and the full *User Guide* are provided on the Multisim CD as Adobe PDF files and are accessible from the Multisim program folder on the Windows Start menu.

License Agreement

Please read this license carefully before installing and using the software contained in this package. By installing and using the software, you are agreeing to be bound by the terms of this license. If you do not agree to the terms of this license, simply return the unused software within thirty days to the place where you obtained it and your money will be refunded.

1. Copyright. The software in this package is copyright Interactive Image Technologies Ltd. (IIT). The software is licensed for use only on the terms set forth herein. You may use this software on any computer for which it is designed as long it is run in only one place at a time. You must pay for additional copies of the software if more than one copy will be running at the same time on one or more computers. You may not rent, sell, lease, sub-license, time-share or loan the software to others. You may not transfer this license without the written permission of IIT. Failure to comply will result in the automatic termination of this license.

In addition to the above restrictions, the following conditions apply if you have paid for license(s) of a Multi-Station Version or Student Edition.

Multi-Station Version Only. This multi-station license allows you to install this software on a specified number of stations. Each station shall consist of only one computer with only one user. If your multi-station version is to be used in a network environment, each computer connected to that network from which this software will be run shall be considered one station. If your multi-station version is to be used in an environment where each computer is stand-alone, each computer shall be considered one station. The total number of stations your particular multi-station license is restricted to is _____.

2. Limited Warranty. IIT warrants that under normal use for a period of thirty (30) days from the date of delivery that: (a) the media on which the software is furnished will be free from defects in the material and workmanship; and (b) the software will operate substantially as described in the User's Guide (documentation). In order to make a claim under this warranty you must call IIT for authorization to return any defective item during the warranty period. If you return merchandise to IIT, you must insure the defective item being returned because IIT does not assume the risk of loss or damage while in transit.

Upon return of a defective item, IIT shall, upon verification of the defect or error, at IIT's option, either repair or replace the defective copy or refund the amount paid for the license. If IIT elects to provide a refund, upon the date you receive notice of such election, this license shall terminate and you must comply with the provisions set out below.

You must assume full responsibility for the selection of this software to achieve your intended purposes, for the proper installation and use of the software and for verifying the results obtained from use of the software. IIT does not warrant in any way that the functions contained in the software will meet your requirements, that the software is fit for any particular purpose or that the operations of the software will be uninterrupted and error-free.

3. Term. The License granted in this agreement is effective until termination. Your misuse shall automatically terminate this license if you breach any of its terms and conditions. Upon termination, you shall return all media containing the software and all documentation to IIT and destroy any copies of the software or any portions of it which have not been returned to IIT, including copies resident in computer memory.

4. Copies, Modification or Merger. You shall not copy or modify all or any portion of the software or documentation or merge it into another software program. Copies shall include, without limitation, any complete or partial duplications on any media, adaptations, translations, compilations, partial copies within modifications, mergers with other material from whatever source, and updated works. You shall use your best efforts to prevent any unauthorized copying of the software.

You shall not make any change or modification to any of the executable files, nor shall you reverse engineer, de-compile or disassemble the software or any portion of it, or otherwise attempt to determine the underlying source code of the software or permit any such actions.

5. Disclaimer/Limitation of Liability. IIT expressly disclaims all other warranties, whether oral or written, express or implied, including without limitation warranties of merchantability or fitness for a particular purpose. All warranties shall terminate thirty days from date of delivery of the software to you.

Your exclusive remedy and IIT's entire liability arising from or in connection with the software, the software documentation, and/or this license (including without limitation for breach of war-

ranty) shall be, at IIT's option, the repair or replacement of software diskettes or refund of license fee.

In no event shall IIT's total liability for any damages, direct or indirect, in connection with the software, the software documentation, and/or this license exceed the license fees paid for your right to use this copy of the software, whether such liability arises from any claim based upon contract, warrants, tort or otherwise. In no event shall IIT or its partners be liable for any loss of profit or any other commercial damage, including but not limited to special, incidental, consequential or other damages, resulting from or in any way connected with the use of this software and including but not limited to any damages resulting from the use of the software for any special or high-risk applications such as those relating to or involving nuclear designs, medical devices and other critical or potentially dangerous applications.

IIT specifically disclaims any other warranties, expressed or implied, including but not limited to the implied warranties of merchantability and fitness for a particular purpose.

Some jurisdictions do not allow the exclusion of implied warranties, so the above exclusion may not apply to you. In that event, any implied warranties are limited in duration to ninety (90) days from the date of delivery of the software. This warranty gives you specific legal rights. You may have other rights, which vary from jurisdiction to jurisdiction.

No action for any breach of warranty shall be commenced more than one year following the expiration of such warranty.

6. General. You acknowledge that you have read this agreement, understand it and agree to be bound by its terms and conditions. You further agree that it is the complete and exclusive statement of the agreement between you and IIT and supersedes any proposal or prior agreement or any other communications between IIT and you relating to the use of the software.

If any provision of this Agreement is unenforceable, all others shall remain in effect. This Agreement shall be governed by the internal laws of the Province of Ontario and Canada, including Canadian copyright laws. The exclusive venue in the event of any suit, proceeding or claim brought by you, and at our option, any suit, proceeding or claim brought by IIT, shall be in the Courts located in Metropolitan Toronto, Ontario. If you have any questions regarding this Agreement, you may contact IIT by writing to us at the address set out below:

INTERACTIVE IMAGE TECHNOLOGIES LTD.

111 Peter Street, Suite 801 Toronto, Ontario M5V 2H1

Tel: (416) 977-5550 **Fax:** (416) 977-1818

e-mail: ewb@electronicsworkbench.com

Internet: www.electronicworkbench.com

Acknowledgements

multiSIM™ and Electronics Workbench™ copyright © 1989, 1992-1999 Interactive Image Technologies Ltd. All rights reserved.

Portions of this product are provided under license from:

- Green Mountain Computing Systems
- Metamor, Inc.

Table of Contents

Introduction

1.1	About this Chapter	1-1
1.2	About this Manual.	1-1
1.3	What is Multisim?	1-1
1.4	Installing Multisim.	1-2
1.5	How to Reach Us	1-4
1.6	Introduction to the Multisim Interface	1-5
1.7	Customizing the Multisim Interface	1-7

Building a Circuit

2.1	About this Chapter	2-1
2.2	Introduction.	2-1
2.3	Starting your Circuit File	2-2
2.4	Placing Components on the Circuit Window	2-3
2.5	Changing the Label and Color of Individual Components and Nodes	2-10
2.6	Wiring Components	2-10
2.7	Adding Text to the Circuit.	2-14
2.8	Conclusion	2-15

Editing a Component

3.1	About this Chapter	3-1
3.2	Introduction to the Component Editor.	3-1
3.3	Accessing the Component Editor	3-2
3.4	Beginning to Edit a Component	3-2
3.5	Conclusion	3-3

Adding Instruments to Your Circuit

4.1	About this Chapter	4-1
4.2	Introduction.	4-1
4.3	Adding and Connecting Instruments	4-2
4.4	Configuring Instrument Settings	4-3
4.5	Conclusion	4-5

Simulating Your Circuit

5.1	About this Chapter	5-1
5.2	Simulating your Circuit	5-1
5.3	Viewing Simulation Results	5-1
5.4	Conclusion	5-2

Performing Analyses on Your Circuit

6.1	About this Chapter	6-1
6.2	The Analyses	6-1
6.3	About Transient Analysis	6-1
6.4	Running the Analysis	6-2
6.5	Conclusion	6-4

Using an HDL

7.1	About this Chapter	7-1
7.2	About HDLs in Multisim	7-1
7.3	Using a VHDL-Modeled Component	7-2
7.4	Simulating the Circuit	7-4
7.5	Interfacing to Programmable Logic Synthesis	7-4
7.6	Conclusion	7-5

Generating a Report

8.1	About this Chapter	9-1
8.2	Introduction.	9-1
8.3	Creating and Printing a BOM	9-1
8.4	Conclusion	9-2

Chapter 1

Introduction

1.1 About this Chapter

This chapter introduces you to this manual and to Multisim itself. It also explains how to install the full product, or feature codes for add-in modules.

1.2 About this Manual

This manual is written for all Multisim users. It provides an overview of some of the main Multisim functions, using a tutorial approach where you are led through the steps of building, simulating, analyzing and reporting on a basic circuit. The majority of the functions described in this manual are available to all Multisim users, regardless of their version of the product. Where the functionality described is available only in some versions of Multisim, this is indicated by the presence of this symbol in the margin:



This manual assumes that you are familiar with Windows applications and know how, for example, to choose a menu from a command, use the mouse to select an item, and enable/disable an option. If you are new to Windows, see your Windows documentation for help.

1.3 What is Multisim?

Multisim is a complete system design tool that offers a very large component database, schematic entry, full analog/digital SPICE simulation, VHDL/Verilog design entry and simulation, FPGA/CPLD synthesis, RF capabilities, postprocessing features and seamless transfer to PCB layout packages such as Ultiboard, from Electronics Workbench. It offers a single, easy-to-use graphical interface for all your design needs.

Multisim provides all the advanced functionality you need to take designs from specification to production. And because the program tightly integrates schematic capture, simulation, and

programmable logic, you can design with confidence, knowing that you are free from the integration issues that often arise when exchanging data between applications from different vendors.

1.4 Installing Multisim

1.4.1 Single User Installation

The CD-ROM you received in your Multisim package is self-starting. Follow the directions below and on the screen during the installation process.

Note To successfully install Multisim, you may need up to 250 MB of hard disk space, depending on which version you have purchased. For example, the Personal Edition requires about 100 MB.

➤ To install Multisim:

1. If your version of Multisim has been issued with a dongle (hardware key) insert it into the parallel port (typically LPT1) in the back of your computer now. If you did not receive a dongle, this step is not required.
2. Exit **all** Windows applications prior to continuing with this installation.
3. Insert the Multisim CD into your CD-ROM drive. When the “Welcome” screen appears, read it. Click **Next** to continue.
4. Read the Multisim License Agreement, which can also be found in the front of this manual. To accept the terms of the agreement, and to continue to the next screen, click **Yes**. If you do not accept the terms of the agreement, click **No** and the Multisim installation will be terminated.
5. Read the system update screen that appears. The system windows files require updating at this time. Click **Next** to allow your windows system files to be updated.
6. You are once again reminded to close all Windows files. Click **Next** to re-boot your computer. This step is required to allow the setup program to use the updated Windows files.

Note Leave the Multisim CD in the CD-ROM drive. Once you have re-booted, Multisim will automatically continue the installation process. You will see the “Welcome” and “License Agreement” screens again; just click **Next** and **Yes**, respectively, to continue.

7. Enter your name, company and the 20 digit serial number (2 letters and 18 numbers) provided to you with Multisim. You will find this serial number on the back of the CD-ROM package. Click **Next** to continue.

8. If you purchased optional add-in modules with Multisim, you will also have received an 18 digit feature code for each module. Enter the first feature code now. If you did not receive a feature code with Multisim, this step is not required. Click **Next** to continue. If you enter a feature code and click **Next**, a new blank field appears. Continue entering the provided feature codes and clicking **Next** until you have entered all the feature codes you need. Leave the last field empty and click **Next**.

Note The feature code is *not* the same as your serial number. Only users who have purchased additional features with Multisim receive a feature code.

9. Select the location in which you want to install Multisim. Choose the default folder or click **Browse** to select a different location or to enter your own folder name. Click **Next** to continue.

10. Setup will now create the program folder you choose in step 9. Click **Next** to continue.

Multisim will now finish being installed. If for any reason you wish to stop the installation, click **Cancel** and the installation of Multisim will be terminated.

After Multisim is installed, you have the option of installing Adobe Acrobat Reader Version 4. This software is required to read the electronic manuals. If you already have Acrobat 4 or do not want to install it, click **Cancel**. If you do not have Acrobat Reader 4 and wish to install it, click **Next** and follow the instructions provided during the installation.

1.4.2 Installing Feature Codes

If you already have Multisim installed because you purchased it earlier, and have now received feature code(s) for optional add-in features which you purchased subsequently, you will need to re-run the initial installation. This provides you with the opportunity to enter your feature codes into Multisim, thereby unlocking the feature. When installing feature codes, you do not have to un-install Multisim.

- To install a feature code with a previous installation of Multisim:
 1. Re-run the Multisim installation program, as described in the previous section.
 2. When you are prompted, enter the feature code in the blank field, and click **Next**. A new blank field appears.
 3. If you have purchased more than one feature, enter the next feature code, and click **Next**.
 4. Continue to entering feature codes and clicking **Next** until you have entered all the feature codes you need.
 5. When you are finished entering feature codes, leave the last field empty and click **Next**.

1.5 How to Reach Us

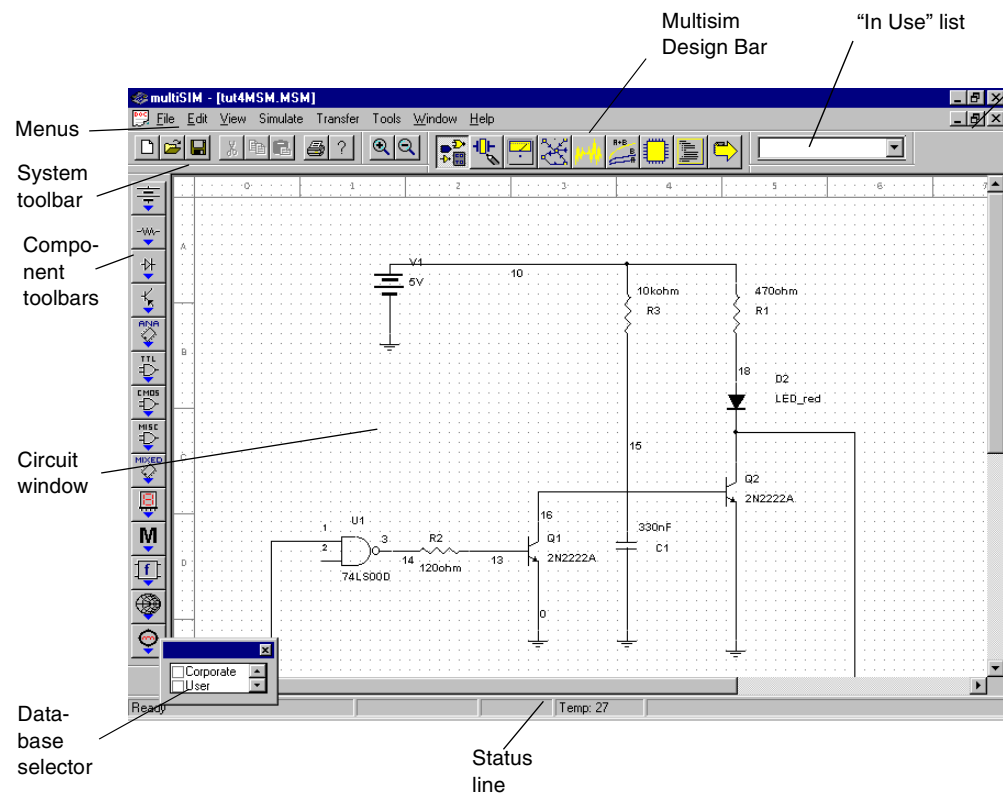
Electronics Workbench provides you with multiple ways of contacting us:

- If you have Internet access, send an email message to support@electronicsworkbench.com.
- Fax information to our Technical Support department at (416) 977-1818.
- Visit our web page at www.electronicsworkbench.com.
- Call us at (416) 977-5550 and ask for sales or technical support.

1.6 Introduction to the Multisim Interface

1.6.1 Basic Elements

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



Note Your circuit window may, by default, have a black background; however, for the purposes of this document, we show a white background. To change the background color, see “Controlling Current Circuit Display” on page 1-7.

Menus are, as in all Windows applications, where you find commands for all functions.

The **system toolbar** contains buttons for commonly-performed functions.

The **Multisim Design Bar** is an integral part of Multisim, and is explained in more detail below.

The “**In Use**” list lists all the components used in the current circuit, for easy re-use.

The **component toolbars** contain Parts Bin buttons that let you open component family toolbars (which, in turn, contain buttons for each family of components in the Parts Bin).

The **circuit window** is where you build your circuit designs.

The **database selector** allows you to choose which database levels are to be visible as component toolbars.

The **status line** displays useful information about the current operation and a description of the item the cursor is currently pointing to.

1.6.2 Design Bar

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



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



The **Component Editor** Design Bar button lets you modify the components in Multisim, or add components.



The **Instruments** Design Bar button lets you attach instruments to your circuit and see the results of your simulation on those instruments.



The **Simulate** Design Bar button lets you start, stop or pause the simulation of your circuit design.



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



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



The **VHDL/Verilog** Design Bar button allows you to work with VHDL modeling (not available in all versions).



The **Reports** Design Bar button lets you print reports about your circuits (Bill of Materials, list of components, component details).



The **Transfer** Design Bar button lets you communicate with and export to other programs, such as Ultiboard, also from Electronics Workbench. You can also export simulation results to programs such as MathCAD and Excel.

This manual explains the use of those Design Bar buttons that are necessary to create and simulate the circuit described in the manual. For details on all the Design Bar buttons, see the *Multisim User Guide*.

1.7 Customizing the Multisim Interface

You can customize virtually any aspect of the Multisim interface, including the toolbars, colors in your circuit, page size, zoom factor, time for autosave, symbol set (ANSI or DIN) and printer setup. Your customization settings are saved individually with each circuit file you use, so you could, for example, have one color scheme for one circuit and another for a different circuit. You can also override the settings for individual instances (for example, change one particular component from red to orange) or for the entire circuit.

To change settings for the *current* circuit, you typically right-click on the circuit window and choose from the pop-up menu options available.

Your user preferences (set using **Edit/User Preferences**) form the default settings to be used for all *subsequent* circuits, but do not (generally) affect the current circuit. Any newly created circuit uses, by default, the user preferences of the current circuit. For example, if your current circuit shows component labels, when you choose **File/New** and create a new circuit, that circuit will be set to show component labels as well.

1.7.1 Controlling Current Circuit Display

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

- To control your current circuit's display, right-click on the circuit window and choose, from the pop-up menu that appears, one of the following:
 - **Grid Visible** (toggles on and off)
 - **Show Title Block and Border** (toggles on and off)
 - **Color** (lets you choose the colors used for different elements placed on the circuit window)
 - **Show** (lets you choose how much detail is to be shown for components and related elements).

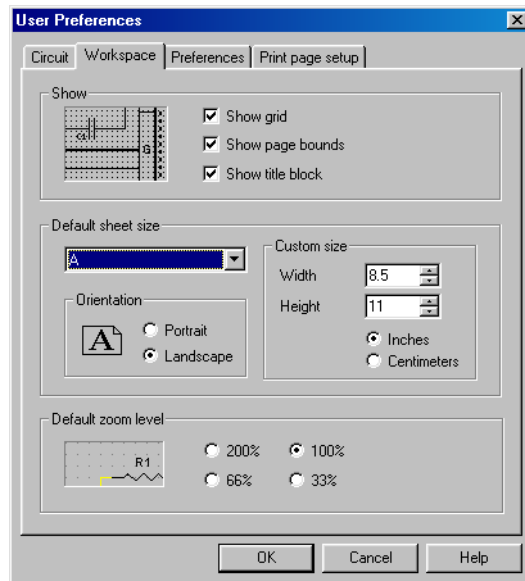
Try experimenting with these options if you wish.

1.7.2 Setting Default User Preferences

You can set up a default configuration that controls how Multisim's circuit windows look when you first create them. You do this using user preferences, which affect all subsequent circuits, but not the current circuit.

Introduction

- To set your default user preferences, choose **Edit/User Preferences**. The User Preferences screen appears:



In this window, choose the desired tab. For example, to set which component labels and colors are used, click the Circuit tab. To define whether grid, page bounds and titles blocks are shown, use the Workspace tab. Experiment with these options as you wish. Remember, you will not see the results of your choices until you create a new circuit file.

1.7.3 Other Customization Options

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

- system toolbar
- zoom toolbar
- Design Bar
- “in use” list
- database selector.

These changes apply to all circuits you are working with. Moved or resized items will return to that location and size when next opened.

Finally, you can use the **View** menu to display or hide various elements.

Chapter 2

Building a Circuit

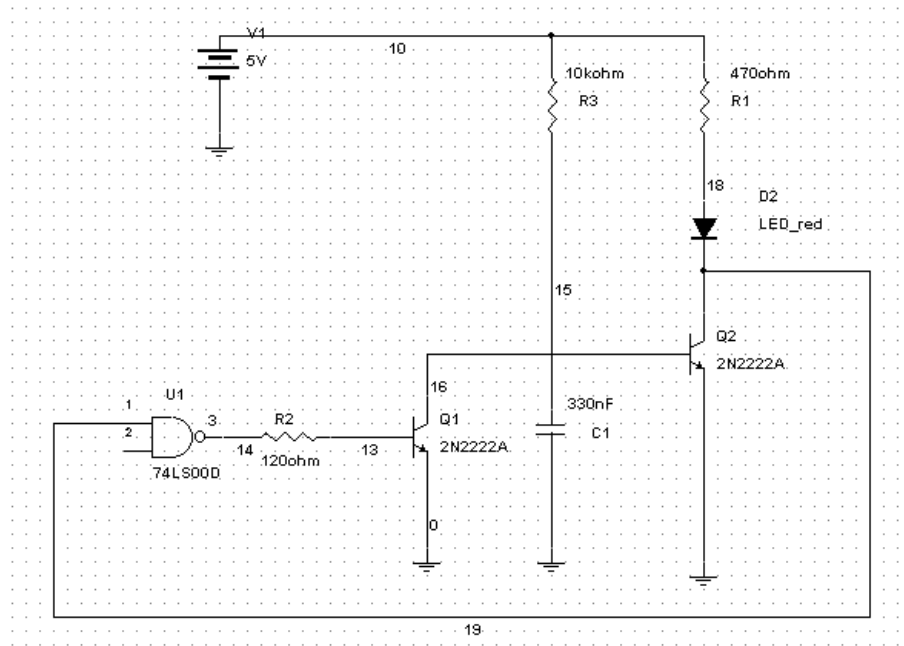
2.1 About this Chapter

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

2.2 Introduction

This manual takes you through the process of building and simulating a simple circuit. The first stage is to choose the components you want to use, place them on the circuit window in the desired position and orientation, wire them together, and otherwise prepare your design.

The circuit you will build in this chapter is a simple circuit that flashes an LED on and off. When you have completed the steps in this chapter, you will have a circuit that looks like this:



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

2.3 Starting your Circuit File

To start building your circuit file, just run Multisim. Multisim automatically opens a blank circuit file with which you can work. The circuit window you see when you run Multisim has its color, size, and display options based on the previously set user preferences. You can change these options to suit your needs by using the pop-up menu, as described on page 1-7, or by consulting the *Multisim User Guide*.

2.4 Placing Components on the Circuit Window



You are now ready to start placing the components for your circuit. As explained in the *Multisim User Guide*, Multisim offers up to three levels of component databases (“Multisim master”, “user” and, for some Multisim versions, “corporate/project (corp/proj)”). For the purposes of this tutorial, we are concerned only with the “Multisim” level, which is the component level shipped with Multisim. For more about the other database levels, see the *Multisim User Guide*.

2.4.1 About the Component Toolbars

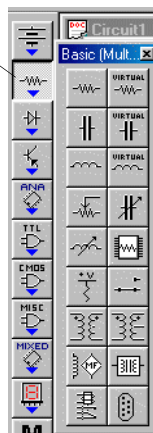


By default, the **Component** Design Bar button is selected and the component toolbar is visible. If it is not visible, click the **Component** button from the Design Bar.

The components you need to use to create a circuit are grouped into logical divisions or Parts Bins, each Parts Bin represented by a button on the component toolbar. Placing your cursor on one of these Parts Bin buttons displays another toolbar, the component family toolbar, containing buttons representing the component families contained in that Parts Bin.

For example:

Placing the cursor on this component toolbar Parts Bin...



...reveals this component family toolbar.

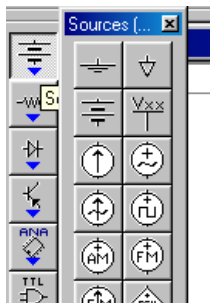
2.4.2 Placing Components

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

2.4.2.1 Placing the First Component

Step 1: Place a Battery

- To place the first component (a 5 volt battery):
 1. Place the cursor on the Sources Parts Bin button (or click it). The contents of the Sources family toolbar appear:



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



2. Click the DC Voltage Source button. Your cursor changes to indicate a part is ready to be placed.

Arrow points to the upper left pin so you can easily line up your component on the desired grid point.



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

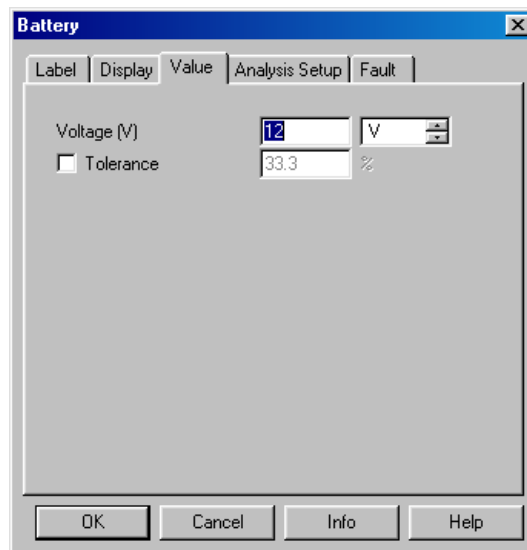


Note The descriptive text that appears around the various components can, if you wish, be hidden. To hide it, use the options from the Show screen, which appears when you right-click on the circuit window and choose **Show** from the pop-up menu.

Step 2: Change the Battery's Value

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

- To change the battery's value:
 1. Double-click on the battery. The battery's properties screen appears, with the Value tab displayed.



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

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

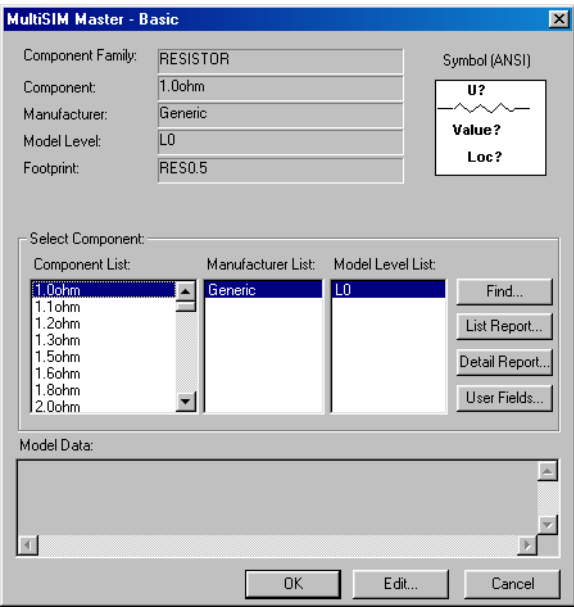
This ability to change the value of a part exists only for “virtual” components. Virtual components are not real; that is, you could not buy them from a supplier. Virtual parts include all sources, virtual resistor/capacitor/inductor parts, and numerous other “ideal” devices intended to provide theoretical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

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

2.4.2.2 Placing the Next Components

Step 1: Place a Resistor

- To place the first resistor:
 1. Place your cursor on the Basic Parts Bin button and, from the toolbar that appears, click the Resistor button. The resistor’s Browser screen appears:



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

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

2. Scroll through the **Component List** to find the 470ohm resistor we need for our circuit.

Tip To make your scroll through the Browser’s **Components List** faster, simply type the first few characters of the component’s name. For example, type “470” to move to that area of the list.

3. Select the 470ohm resistor and click **OK**. The cursor reappears on the circuit window.

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

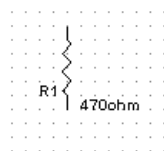
Notice that the resistor looks different from the battery you just placed—the resistor appears in a different color to remind you it is a real component (and will be exported to PCB Layout software, if you choose to perform that step).

Step 2: Rotate the Resistor

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

➤ To rotate the resistor:

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



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

Step 3: Add Other Resistors

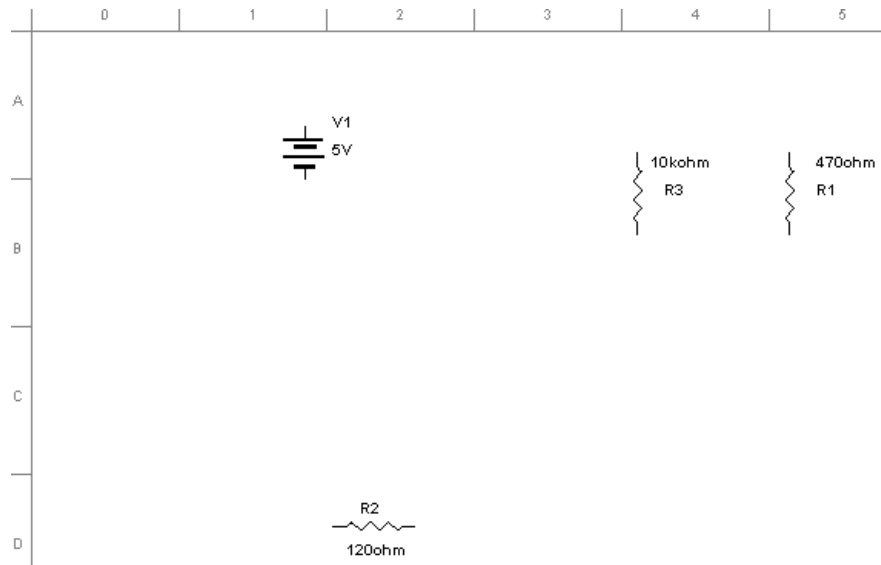
The circuit calls for two other resistors, this time a 120ohm and another 470ohm.

To add the other resistors:

1. Following the instructions above, add a 120ohm resistor at roughly the intersection of row D and column 2. Notice how this second resistor is given the reference ID “R2”, to indicate it is the second resistor placed.
2. Place the third resistor, a 470ohm one (you could use the “In Use” list for this if you wish), at roughly the intersection of 4B, and rotate it.

Take a moment to look at the “In Use” list, just to the right of the Design Bar. It lists all the components you have placed so far. You can easily re-use a component from this list by clicking on it.

The results should look like this:]



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

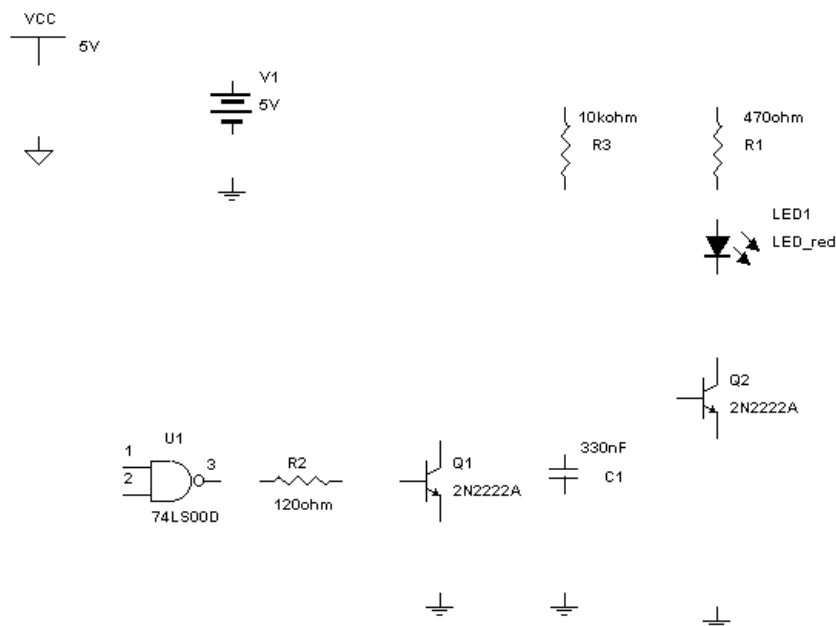
Step 4: Save Your changes

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

2.4.2.3 Placing Other Components

- Following the procedures above, place the following components in the following locations as shown in the beginning of section 2.2:
 - a red LED (from the Diodes family) just below R1
 - a 74LS00D (from the TTL group), roughly at D1 (Since this component consists of four separate gates, you are prompted to choose which one you will use. All are identical, so you can choose any one.)
 - a 2N2222A BJT_NPN (from the Transistor group), to the right of R2
 - another 2N2222A BJT_NPN below the LED (simply copy the previous one and paste the copy at the new location)
 - a 330nf capacitor (from the Basic group), to the right of your first BJT_NPN, and rotate it clockwise (move the labels after rotating, if desired)
 - a ground (from the Sources group), below V1, Q1, Q2 and C1 — you may use any number of grounds in your circuit, as in this tutorial, or just one with multiple parts connected to it)
 - a 5Volt VCC (from the Sources group), in the top left corner of the circuit window, and a digital ground (from the Sources group) just below it.

The results should look something like this:



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

2. Save your changes using **File/Save**.

2.5 Changing the Label and Color of Individual Components and Nodes

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

- To change the label of any individual component you have placed:
 1. Double-click on the component. The component's properties screen appears.
 2. Click the Label tab and enter or modify the label (which must be composed of letters or numbers only—no special characters or spaces).
 3. To cancel your changes, click **Cancel**. To save your changes, click **OK**.
- To change the color of any individual component, right-click on it and choose **Color** from the pop-up menu. Choose the desired color from the screen that appears.

Note Changing the color of an individual component is different from changing the color scheme of the current circuit, or of the defaults set in user preferences, as explained earlier in this manual.

2.6 Wiring Components

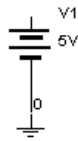
Now that you have the components placed on the circuit window, you need to wire them together. All components have pins that you use to wire them to other components. In Multisim, you can choose to wire components either automatically or manually. Automatic wiring, a feature unique to Multisim, means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

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

2.6.1 Using Automatic Wiring

We'll begin with wiring V1 and its ground.

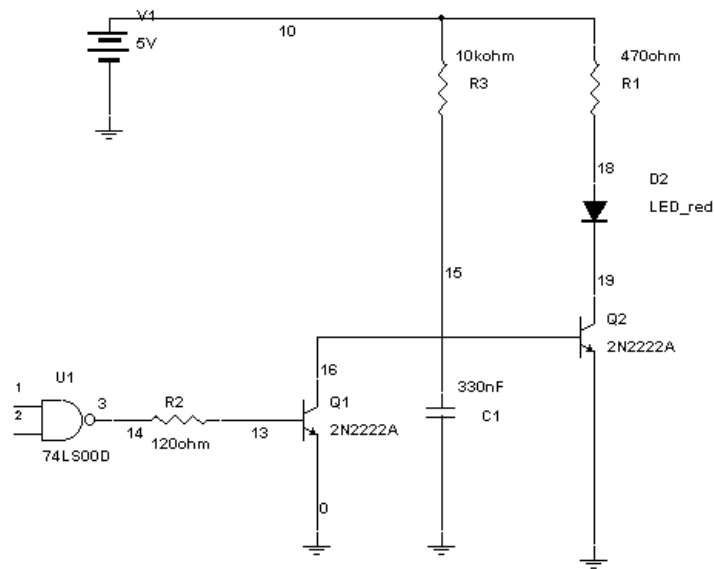
- To start the automatic wiring process:
 1. Click the pin coming out of the bottom of V1.
 2. Click the pin on the top of the ground. The two components automatically become wired together and look like this:



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

3. Using automatic wiring, make the following connections:
 - wire V1 to R1
 - wire R1 to the LED
 - wire the LED to the collector of Q2
 - wire the emitters of Q2 and Q1
 - wire C1 to its ground
 - wire base of Q1 to R2
 - wire R2 to pin3 (output) of U3
 - wire R3 to C1
 - wire pin1 of U1 to pin 2 of U1
 - wire R3 to the wire running from V1 to R1 (node 1) — click on the pin of R3 and then click the wire to automatically place a junction where the wires meet
 - wire base of Q2 to collector of Q1.

The results should look like this:



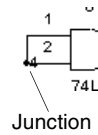
- To stop the wiring process, press ESC.
- To delete a wire, right-click on it and choose **Delete** from the pop-up menu, or press DELETE.

2.6.2 Using Manual Wiring

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

- To add a junction:
 1. Choose **Edit/Place Junction**. The cursor changes to indicate a junction is ready to be placed.
 2. Click on the wire between the inputs of U1, where you want to place the junction.
 3. When prompted, leave the node properties at their defaults and click **OK**.

4. The junction appears on the wire:



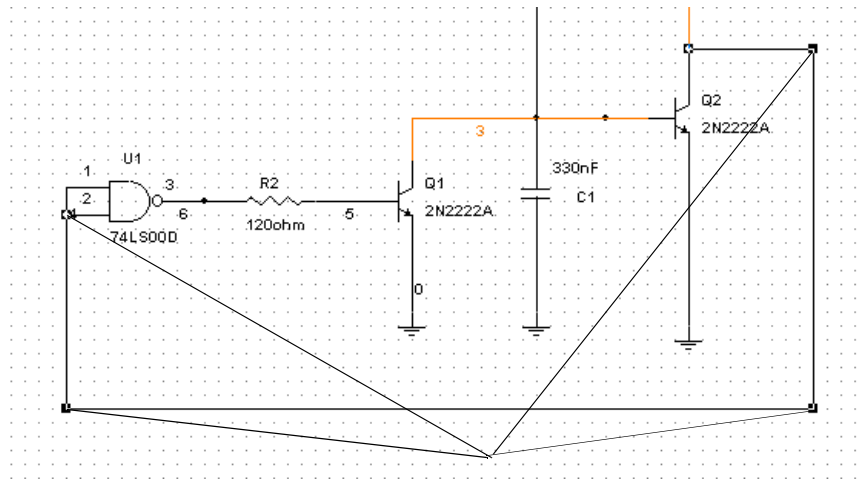
We now need to use manual wiring to ensure that the wire follows the path we want. To help position the wire appropriately, display the grid (if it is not visible).

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

We are now ready to wire the connection manually.

- To perform a manual connection:
 1. Click the junction you just placed at the inputs of U1.
 2. Drag the wire down to below the other components and click. This “fixes” the wire in its place.
 3. Drag the wire across the width of the circuit to a few grid spaces past the last component. Click again.
 4. Drag the wire up to a point across from the wire between LED1 and Q2. Click again.
 5. Drag the wire to a place on the wire between LED1 and Q2 and click again.

The results look like this:



The mouse
was clicked at
these points.

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

While the wire is selected, you can add additional drag points by using the CTRL key and clicking the wire where you want to add more points.

- To remove a drag point, press the CTRL key and click on the point.

2.7 Adding Text to the Circuit

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

- To add a title block, choose **Edit/Set Title Block**. Enter the desired title block text in the field and click OK. The title block appears at the bottom right of your circuit window.
- To add text:
 1. Choose **Edit/Place Text**.
 2. Click anywhere on the circuit window. A text box appears.
 3. Type the text—for example, type “My tutorial circuit”.
 4. Click on the location on the circuit window where you want the text placed.

- To delete text, right-click on the text box and choose **Delete** from the pop-up menu or press DELETE.
- To change the color of text, right-click on the text box, choose **Color** from the pop-up menu, and choose the desired color.
- To edit text, double-click on the text box and make your changes. Click any location out of the text box to stop editing text.
- To move text, click on the text box and drag it to a new location.

2.8 Conclusion

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

Chapter 3 Editing a Component

3.1 About this Chapter

This chapter gives you a brief introduction to the kinds of functions you can perform with the Component Editor. It shows you how to access the Component Editor and move through its tabs. However, because the Component Editor is, of necessity, both powerful and complex, an explanation of using the Component Editor for specific tasks is beyond the scope of this manual. For details on using the Component Editor to edit components, see the *Multisim User Guide*.

3.2 Introduction to the Component Editor

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

You can also use the Component Editor to create your own component (and place it into the database), load a component from another source, or remove a component from the database.

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

- general information (such as name, description, manufacturer, icon, family and electrical characteristics)
- symbol (pictorial representation of the component for schematic capture)
- model (information used to represent the actual operation/behavior of the component during simulation) — necessary only for a component that will be simulated
- footprint (the Package information that Multisim uses when exporting a schematic diagram containing this component to a PCB Layout package such as Ultiboard, also from Electronics Workbench).

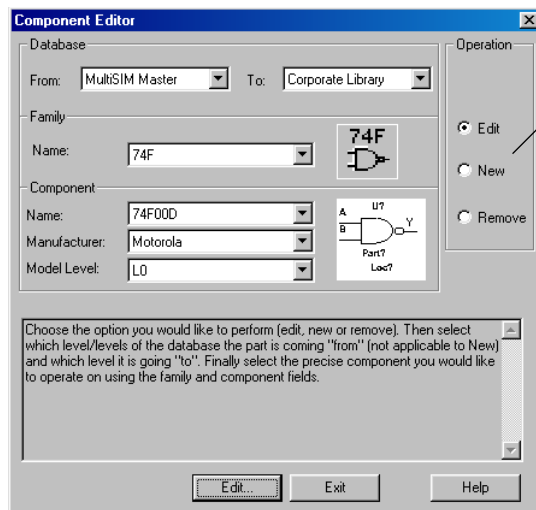
3.3 Accessing the Component Editor

- To access the Component Editor, do one of the following:



- Click the **Component Editor** button in the Design Bar.
- Choose **Tools/Component Editor**. The Component Editor screen appears.

Note Typically, it is much simpler to edit an existing component to generate the component you want, rather than to create the component from scratch.



Choose which operation you want to perform—edit a component, create a component, or remove a component. The top fields of the screen change depending on your selection.

3.4 Beginning to Edit a Component

The first step in using the Component Editor is to select the component you want to edit.

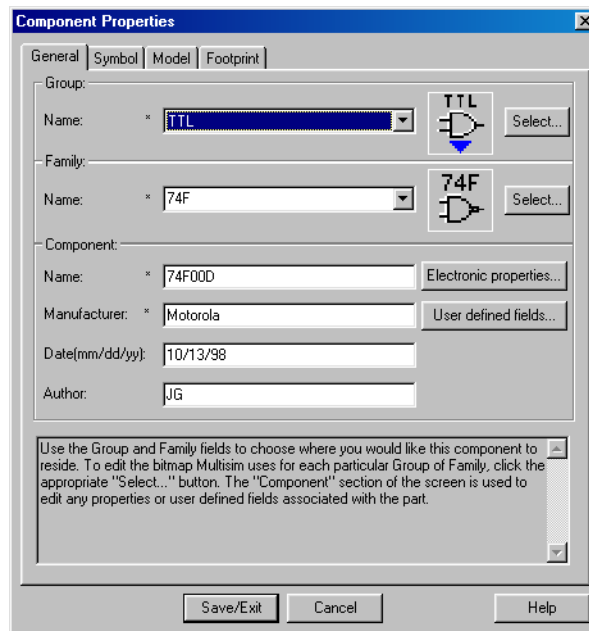
- To edit an existing component:
1. Under the **Operation** options, enable **Edit**.
 2. From the **From** list, choose the database level containing the component you want to edit. Typically, this will be the “Multisim master” level.
 3. From the **To** list, choose the database level in which you want the edited component stored. You will note that the **To** list does not include the “Multisim master” database level, since that level cannot be modified.
 4. From the **Name** list in the **Family** area, choose the component family containing the component you want to edit. The **Name** list in the **Component** area changes to show a list of components in that family.

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

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

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

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



These tabs correspond to the type of information you can edit for a component. To see the Component Editor in action, we would need to actually modify a symbol, a model, or a footprint. This is a process that is, of necessity, somewhat involved and is therefore beyond the scope of this manual. For more about using these tabs, see the *Multisim User Guide*.

3.5 Conclusion

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

Chapter 4

Adding Instruments to Your Circuit

4.1 About this Chapter

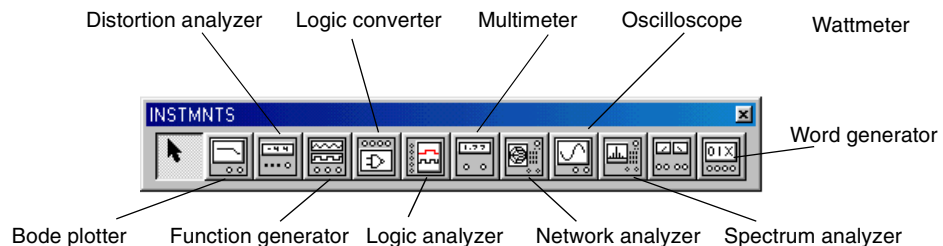
This chapter gives you a brief overview of Multisim's instrument functions and takes you through the steps of adding an instrument to your circuit. Multisim offers a variety of virtual instruments. This chapter describes the oscilloscope—for other instruments, please see the *Multisim User Guide*.

4.2 Introduction

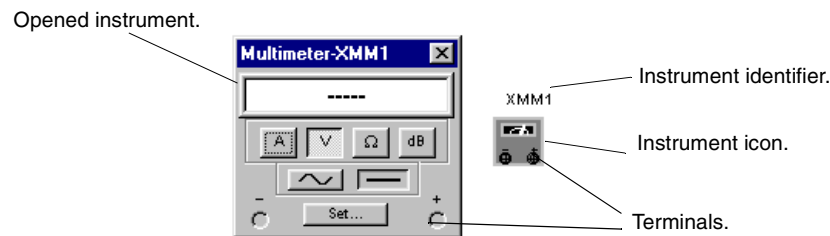
Multisim provides a number of virtual instruments. You use these instruments to measure the behavior of your circuits. These instruments are set, used and read just like their real-world equivalents. They look and feel just like the instruments you've seen and used in a lab. Virtual instruments are one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



Instruments are accessed through the **Instruments** button on the Design Bar. When you click this button, the Instruments toolbar appears. It includes one button for each instrument.



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



4.3 Adding and Connecting Instruments

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

Step 1: Add an oscilloscope

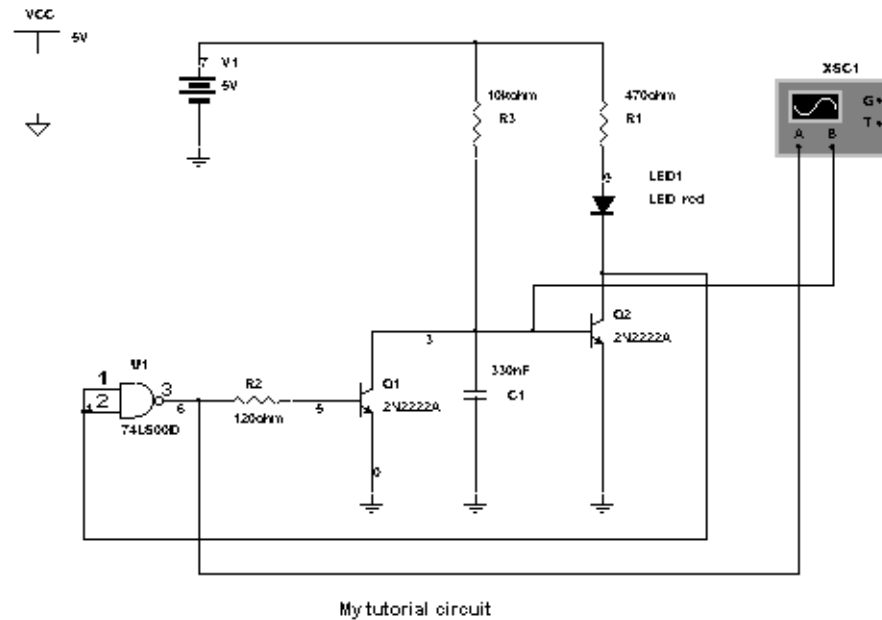


- To add the oscilloscope:
 1. Click the **Instruments** button in the Design Bar. The Instruments toolbar appears.
 2. Click the oscilloscope button. Your cursor changes, to indicate it is ready to place an instrument.
 3. Move the cursor to the right side of your circuit and click.
 4. The oscilloscope icon appears on the circuit window.
 5. You now need to wire the oscilloscope into your circuit.

Step 2: Wire the oscilloscope to the circuit

- To wire the oscilloscope into your circuit:
 1. Click the A terminal on the oscilloscope icon and drag a wire to the junction between U1 and R2.
 2. Click the B terminal on the oscilloscope icon and drag a wire to between Q2 and C1.

Your circuit should look like this:



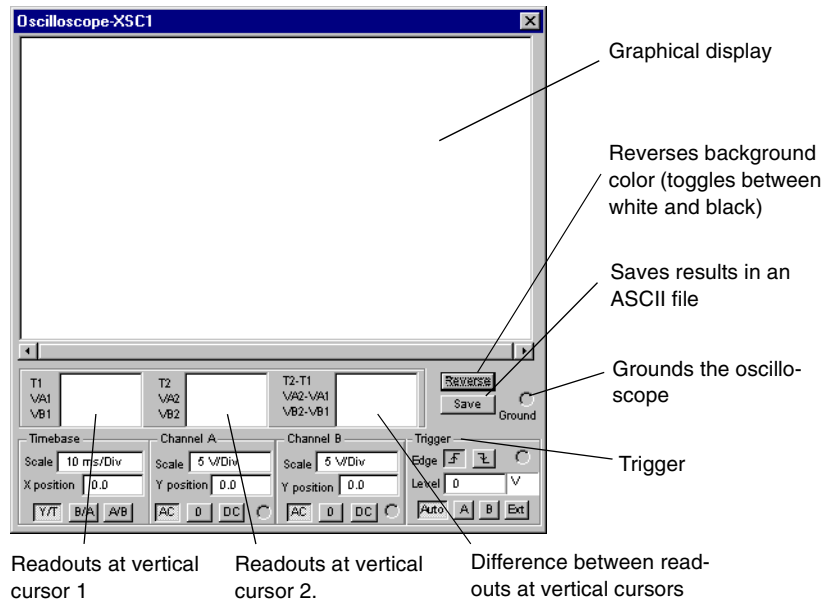
You'll see the results of your work in the next chapter when you simulate the circuit.

4.4 Configuring Instrument Settings

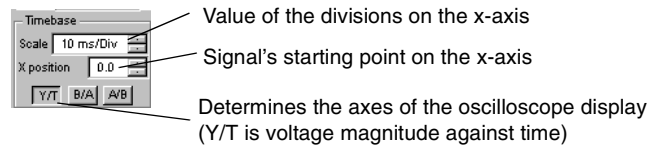
Each virtual instrument provided by Multisim includes its own series of optional settings that control its display.

Adding Instruments to Your Circuit

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



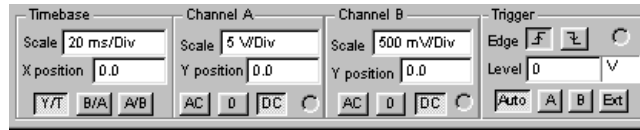
The time base section of the scope's panel controls the scale of the oscilloscope's horizontal or x-axis when comparing magnitude against time (Y/T)



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

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

The results should look like this:



4.5 Conclusion

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

Chapter 5

Simulating Your Circuit

5.1 About this Chapter

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

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

5.2 Simulating your Circuit

You are now ready to simulate your circuit.

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



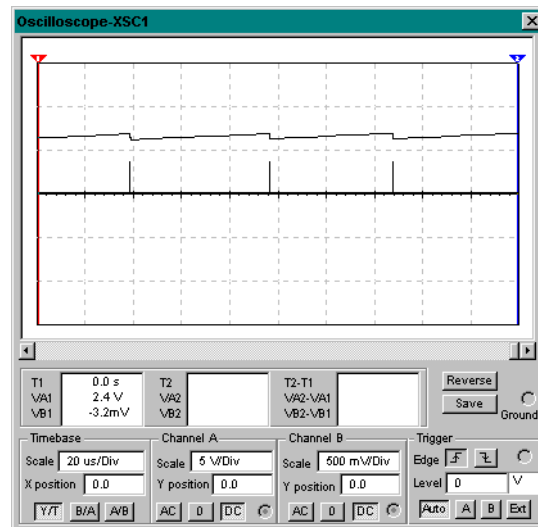
- To simulate the circuit, click the **Simulate** button in the Design Bar and, from the pop-up menu, choose **Run/Stop**.

5.3 Viewing Simulation Results

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

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

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



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

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

Note If the results you get do not appear to be the same as indicated on the oscilloscope face above, it may be due to the sampling rate of the instrument. To fix this, choose **Simulate/Default Instrument Settings**. Click **Maximum Time Step (TMAX)** and enter $1e-4$ in the space provided. Click **Accept**.

5.4 Conclusion

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

Chapter 6

Performing Analyses on Your Circuit

6.1 About this Chapter

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

6.2 The Analyses

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

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



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

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

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

6.3 About Transient Analysis

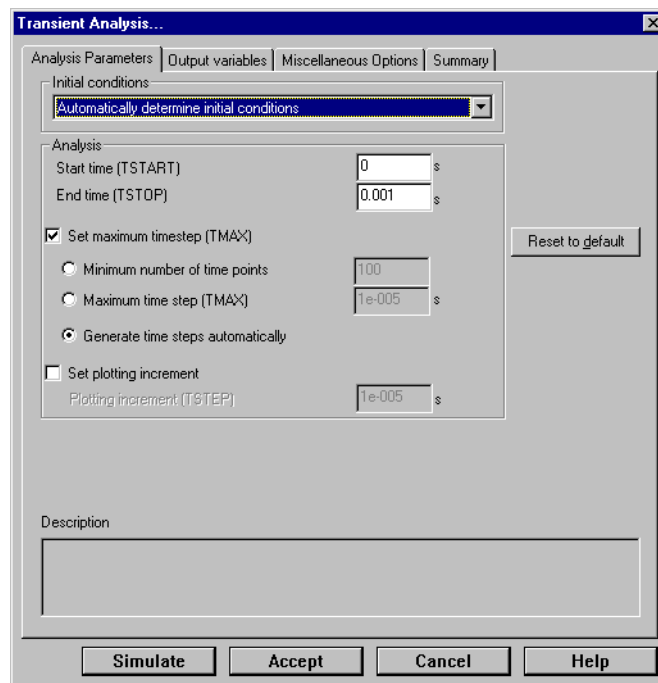
In the *Multisim User Guide*, each analysis is described in detail, including the theory behind what Multisim does to perform the analysis. It is not necessary that you understand this information, which is provided primarily for the sake of completeness.

In transient analysis, also called time-domain transient analysis, Multisim computes the circuit's response as a function of time. Each input cycle is divided into intervals, and a DC analysis is performed for each time point in the cycle. The solution for the voltage waveform at a node is determined by the value of that voltage at each time point over one complete cycle.

6.4 Running the Analysis



- To initiate the analysis, click the **Analysis** button from the Design Bar and choose **Transient Analysis** from the pop-up menu. The Transient Analysis screen appears, offering four tabs.



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

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

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

Step 1: Choose Output Variables

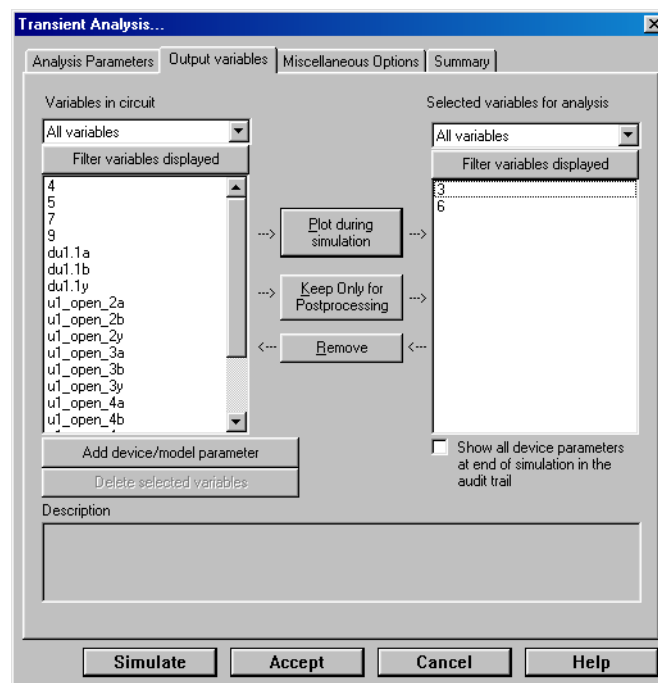
We are going to want to perform the Transient Analysis on nodes 3 (between Q1 and Q2) and 6 (from pin 3 of the gate). Choose those nodes from the available nodes on the Output Variables tab.

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

➤ To select the nodes:

1. Select 3 from the **Filter variables displayed** and click **Plot during simulation**.
2. Select 6 from the **Filter variables displayed** and click **Plot during simulation**.

The results look like this:

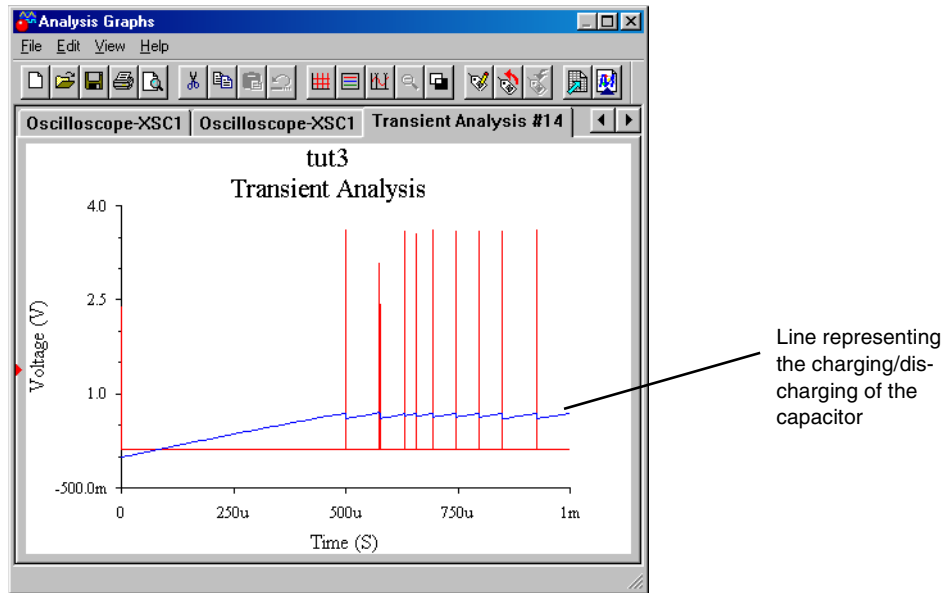


Step 2: Set the Analysis Parameters

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

Step 3: View the Results

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



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



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

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

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

6.5 Conclusion

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

Chapter 7 Using an HDL

7.1 About this Chapter

This chapter gives you a very brief introduction to HDLs and provides a simple example of co-simulation of SPICE with VHDL. Multisim supports simulation using SPICE, VHDL, Verilog, and any combination of these, but this chapter uses only SPICE and VHDL for demonstration purposes.



HDLs are not available in all versions of Multisim. If you will not be using HDLs at all, you may want to skip this chapter.

7.2 About HDLs in Multisim

HDLs are designed specially to describe the behavior of complex digital devices. For this reason they are referred to as “behavioral level” languages. They use behavioral level models (instead of transistor/gate level, like SPICE) to describe the behavior of these devices. Using HDLs avoids the unwieldy task of describing such devices at the gate level, greatly simplifying the design process.

Designers typically choose from two different HDLs: VHDL and Verilog. Multisim supports both of these languages.

HDLs are commonly used for two purposes: for modeling complex digital ICs that could not easily be modeled in SPICE, or for designing circuits in programmable logic. Multisim supports both of these applications of HDLs.

For the second of these two uses, designing programmable logic devices such as FPGAs and CPLDs using VHDL/Verilog, Multisim is ideally suited. However, explaining this process is beyond the scope of this manual. The *Multisim User Guide* gives complete details.

It is the first of these two applications, modelling of complex digital devices, that we use for demonstration purposes in this chapter. In order to keep the explanation on an introductory level, however, we don’t actually use a “complex” device, but a simple NAND gate instead.

Using an HDL

Of course, you would not normally use a VHDL model for a NAND gate, since SPICE does this perfectly well, but this allows us to focus on the process of using a VHDL-modelled part and not on the code itself which makes up the model.

Furthermore, Multisim allows you to use co-simulation (for example, SPICE and VHDL), either by making use of an already available VHDL model or by having you write the VHDL code for the model yourself. We make use of the first of these two options by utilizing an already available model—in our case a part from Multisim's database—although you could use models obtained from anywhere (public domain, colleagues, device vendors, etc.)

Because we are using a part for which a VHDL simulation model already exists, there is no need for you to be familiar with VHDL code to follow along and use this process yourself.

Providing an example in which we write and debug VHDL code is beyond the scope of this manual but is covered in detail in the *Multisim User Guide*.

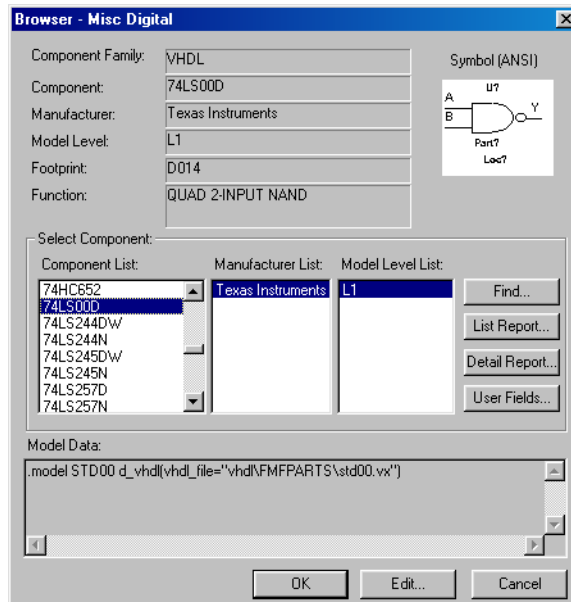
7.3 Using a VHDL-Modeled Component

To see VHDL in action, we need to use a part in our circuit whose simulation model uses VHDL, not SPICE. Our NAND gate can serve this purpose, since it is available as a VHDL-modelled part.

- To select the VHDL-modeled 74LS00D:
 1. From the Misc. Digital Parts Bin in the component toolbar, choose the VHDL family.



The Browser screen appears:.



2. Scroll and select the 74LS00D.
3. Click **OK** and place the new component on your circuit.

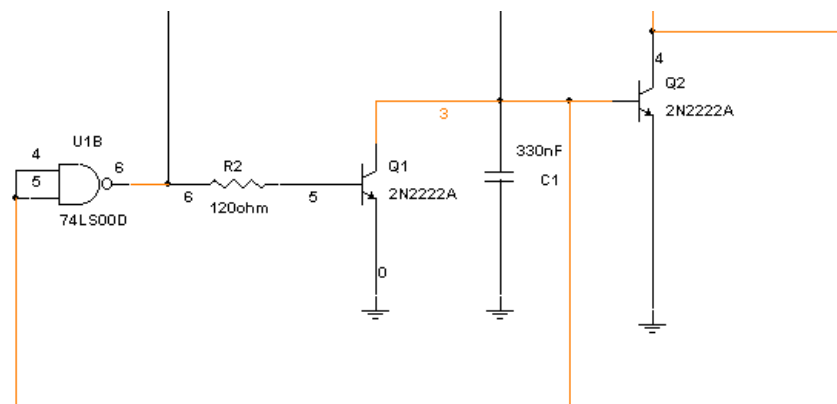
Because we already had a SPICE-modelled 74LS00D in our circuit, we need to delete it, to make room for the new VHDL-modelled part (we don't need two NAND gates).

- To delete our old SPICE-modelled part:
 1. Take note of the current wiring of the old 74LS00D (the wires will disappear when you delete the component).
 2. Select the old 74LS00D, and click **Delete**.

Now we need to connect our new VHDL-modelled 74LS00D.

- To connect our new VHDL-modelled part:
 1. On the circuit window, place the part in the position previously occupied by the old 74LS00D you just deleted.
 2. Wire it the same way as the old 74LS00D you just deleted.

When you are done the results should look like this:



7.4 Simulating the Circuit

Now simulate the circuit again, as described in the previous chapter—the method of performing co-simulation is exactly the same as for SPICE-only simulation. If necessary, open the oscilloscope. You will notice that the results are identical to those for the SPICE-modelled part we simulated in Chapter 5.

In the background, Multisim has performed co-simulation for you by simulating most components in your circuit with SPICE and the NAND gate with VHDL. It knows which simulation engine to use for which part, based on the part's model, and then combines all the simulation results together for display and analysis.

Note that for simulating VHDL or Verilog code on its own (code that you are writing and debugging), simulation is invoked differently because unique tools are needed. This is explained in the *Multisim User Guide*.

7.5 Interfacing to Programmable Logic Synthesis

Multisim also provides complete VHDL and Verilog design, simulate and debug capabilities for designing your circuits for implementation in FPGAs and CPLDs. This requires the optional capability of synthesis, which is also available from Electronics Workbench and is explained in the *Multisim User Guide*.

7.6 Conclusion

This chapter has briefly introduced you to Multisim's VHDL simulation capabilities. The final stage in our circuit design is to generate a Bill of Materials report, as explained in the next chapter.

Using an HDL

Chapter 8 Generating a Report

8.1 About this Chapter

This chapter explains how to create a Bill of Materials (BOM).

8.2 Introduction

Multisim allows you to generate a number of reports: Bill of Materials, Database Family List, and Component Detail Report. This chapter uses the BOM as an example. The other reports are explained in the *Multisim User Guide*.

8.3 Creating and Printing a BOM

A bill of materials lists the components used in your design and therefore provides a summary of the components needed to manufacture the circuit board. Information provided in the Bill of Materials includes:

- quantity of each component needed
- description, including the type of part (example: resistor) and value (examples: 5.1 Kohm)
- reference ID of each component
- package or footprint of each component
- if you have purchased the Team/Project Design module optionally available with the Professional Edition and included in the Power Professional edition, all user fields and their values (for example, price, availability, supplier, etc.) for each component that has such fields. (For more on user fields, see the *Multisim User Guide*.)

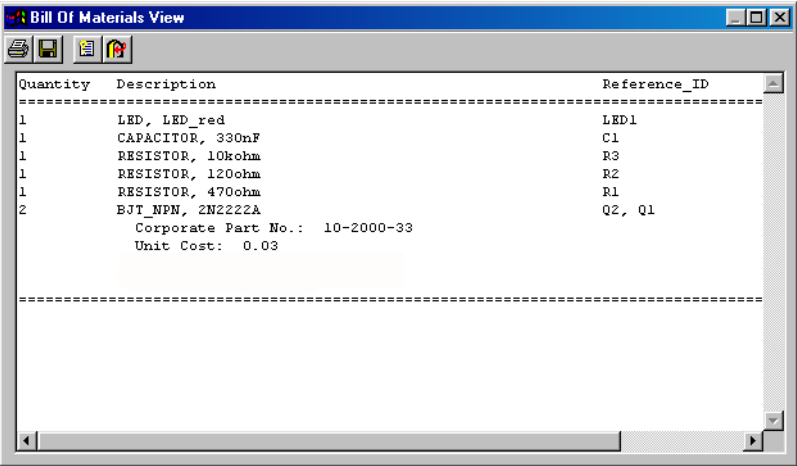


➤ To create a BOM for your circuit:



1. Click the **Reports** button from the Design Bar and choose **Bill of Materials** from the menu that appears.

2. The report appears, looking similar to this:



Quantity	Description	Reference_ID
1	LED, LED_red	LED1
1	CAPACITOR, 330nF	C1
1	RESISTOR, 10kohm	R3
1	RESISTOR, 120ohm	R2
1	RESISTOR, 470ohm	R1
2	BJT_NPN, 2N2222A	Q2, Q1

Corporate Part No.: 10-2000-33
Unit Cost: 0.03

- To print the Bill of Materials, click the Print button. A standard Windows print screen appears, allowing you to choose the printer, number of copies, and so on.
- To save the Bill of Materials to a file, click the Save button. A standard Windows file save screen appears, allowing you to specify the path and file name.

Because the Bill of Materials is primarily intended to assist in procurement and manufacturing, it includes only “real” parts. That is, it excludes parts that are not real or able to be purchased, such as sources or virtual components.
- To see a list of components in your circuit that are not “real” components, click the Others button. A separate window appears, showing these components only.

8.4 Conclusion

Multisim provides a number of report generation capabilities, including the BOM described in this chapter.

You have now been through the basic steps in creating and simulation a circuit. The last button on the Design Bar, for transferring and communicating your designs, is described in the *Multisim User Guide*.