Electronics Workbench™

Ultiboard 9 PCB Layout

User Guide

NATIONAL INSTRUMENTS™

Worldwide Technical Support and Product Information

`ni.com`

National Instruments Corporate Headquarters

11500 North Mopac Expressway   Austin, Texas 78759-3504   USA   Tel: 512 683 0100

Worldwide Offices

Australia 1800 300 800, Austria 43 0 662 45 79 90 0, Belgium 32 0 2 757 00 20, Brazil 55 11 3262 3599,
Canada 800 433 3488, China 86 21 6555 7838, Czech Republic 420 224 235 774, Denmark 45 45 76 26 00,
Finland 385 0 9 725 725 11, France 33 0 1 48 14 24 24, Germany 49 0 89 741 31 30, India 91 80 41190000,
Israel 972 0 3 6393737, Italy 39 02 413091, Japan 81 3 5472 2970, Korea 82 02 3451 3400,
Lebanon 961 0 1 33 28 28, Malaysia 1800 887710, Mexico 01 800 010 0793, Netherlands 31 0 348 433 466,
New Zealand 0800 553 322, Norway 47 0 66 90 76 60, Poland 48 22 3390150, Portugal 351 210 311 210,
Russia 7 095 783 68 51, Singapore 1800 226 5886, Slovenia 386 3 425 4200, South Africa 27 0 11 805 8197,
Spain 34 91 640 0085, Sweden 46 0 8 587 895 00, Switzerland 41 56 200 51 51, Taiwan 886 02 2377 2222,
Thailand 662 278 6777, United Kingdom 44 0 1635 523545

For further support information, refer to the *Technical Support Resources and Professional Services* page. To comment
on National Instruments documentation, refer to the National Instruments Web site at `ni.com/info` and enter the
info code `feedback`.

Important Information

Warranty

The media on which you receive National Instruments software are warranted not to fail to execute programming instructions, due to defects in materials and workmanship, for a period of 90 days from date of shipment, as evidenced by receipts or other documentation. National Instruments will, at its option, repair or replace software media that do not execute programming instructions if National Instruments receives notice of such defects during the warranty period. National Instruments does not warrant that the operation of the software shall be uninterrupted or error free.

A Return Material Authorization (RMA) number must be obtained from the factory and clearly marked on the outside of the package before any equipment will be accepted for warranty work. National Instruments will pay the shipping costs of returning to the owner parts which are covered by warranty.

National Instruments believes that the information in this document is accurate. The document has been carefully reviewed for technical accuracy. In the event that technical or typographical errors exist, National Instruments reserves the right to make changes to subsequent editions of this document without prior notice to holders of this edition. The reader should consult National Instruments if errors are suspected. In no event shall National Instruments be liable for any damages arising out of or related to this document or the information contained in it.

EXCEPT AS SPECIFIED HEREIN, NATIONAL INSTRUMENTS MAKES NO WARRANTIES, EXPRESS OR IMPLIED, AND SPECIFICALLY DISCLAIMS ANY WARRANTY OF MERCHANTABILITY OR FITNESS FOR A PARTICULAR PURPOSE. CUSTOMER'S RIGHT TO RECOVER DAMAGES CAUSED BY FAULT OR NEGLIGENCE ON THE PART OF NATIONAL INSTRUMENTS SHALL BE LIMITED TO THE AMOUNT THERETOFORE PAID BY THE CUSTOMER. NATIONAL INSTRUMENTS WILL NOT BE LIABLE FOR DAMAGES RESULTING FROM LOSS OF DATA, PROFITS, USE OF PRODUCTS, OR INCIDENTAL OR CONSEQUENTIAL DAMAGES, EVEN IF ADVISED OF THE POSSIBILITY THEREOF. This limitation of the liability of National Instruments will apply regardless of the form of action, whether in contract or tort, including negligence. Any action against National Instruments must be brought within one year after the cause of action accrues. National Instruments shall not be liable for any delay in performance due to causes beyond its reasonable control. The warranty provided herein does not cover damages, defects, malfunctions, or service failures caused by owner's failure to follow the National Instruments installation, operation, or maintenance instructions; owner's modification of the product; owner's abuse, misuse, or negligent acts; and power failure or surges, fire, flood, accident, actions of third parties, or other events outside reasonable control.

Copyright

Under the copyright laws, this publication may not be reproduced or transmitted in any form, electronic or mechanical, including photocopying, recording, storing in an information retrieval system, or translating, in whole or in part, without the prior written consent of National Instruments Corporation.

National Instruments respects the intellectual property of others, and we ask our users to do the same. NI software is protected by copyright and other intellectual property laws. Where NI software may be used to reproduce software or other materials belonging to others, you may use NI software only to reproduce materials that you may reproduce in accordance with the terms of any applicable license or other legal restriction.

Trademarks

National Instruments, NI, ni.com, and LabVIEW are trademarks of National Instruments Corporation. Refer to the *Terms of Use* section on `ni.com/legal` for more information about National Instruments trademarks.

Other product and company names mentioned herein are trademarks or trade names of their respective companies.

Members of the National Instruments Alliance Partner Program are business entities independent from National Instruments and have no agency, partnership, or joint-venture relationship with National Instruments.

Patents

For patents covering National Instruments products, refer to the appropriate location: **Help»Patents** in your software, the `patents.txt` file on your CD, or `ni.com/patents`.

WARNING REGARDING USE OF NATIONAL INSTRUMENTS PRODUCTS

(1) NATIONAL INSTRUMENTS PRODUCTS ARE NOT DESIGNED WITH COMPONENTS AND TESTING FOR A LEVEL OF RELIABILITY SUITABLE FOR USE IN OR IN CONNECTION WITH SURGICAL IMPLANTS OR AS CRITICAL COMPONENTS IN ANY LIFE SUPPORT SYSTEMS WHOSE FAILURE TO PERFORM CAN REASONABLY BE EXPECTED TO CAUSE SIGNIFICANT INJURY TO A HUMAN.

(2) IN ANY APPLICATION, INCLUDING THE ABOVE, RELIABILITY OF OPERATION OF THE SOFTWARE PRODUCTS CAN BE IMPAIRED BY ADVERSE FACTORS, INCLUDING BUT NOT LIMITED TO FLUCTUATIONS IN ELECTRICAL POWER SUPPLY, COMPUTER HARDWARE MALFUNCTIONS, COMPUTER OPERATING SYSTEM SOFTWARE FITNESS, FITNESS OF COMPILERS AND DEVELOPMENT SOFTWARE USED TO DEVELOP AN APPLICATION, INSTALLATION ERRORS, SOFTWARE AND HARDWARE COMPATIBILITY PROBLEMS, MALFUNCTIONS OR FAILURES OF ELECTRONIC MONITORING OR CONTROL DEVICES, TRANSIENT FAILURES OF ELECTRONIC SYSTEMS (HARDWARE AND/OR SOFTWARE), UNANTICIPATED USES OR MISUSES, OR ERRORS ON THE PART OF THE USER OR APPLICATIONS DESIGNER (ADVERSE FACTORS SUCH AS THESE ARE HEREAFTER COLLECTIVELY TERMED "SYSTEM FAILURES"). ANY APPLICATION WHERE A SYSTEM FAILURE WOULD CREATE A RISK OF HARM TO PROPERTY OR PERSONS (INCLUDING THE RISK OF BODILY INJURY AND DEATH) SHOULD NOT BE RELIANT SOLELY UPON ONE FORM OF ELECTRONIC SYSTEM DUE TO THE RISK OF SYSTEM FAILURE. TO AVOID DAMAGE, INJURY, OR DEATH, THE USER OR APPLICATION DESIGNER MUST TAKE REASONABLY PRUDENT STEPS TO PROTECT AGAINST SYSTEM FAILURES, INCLUDING BUT NOT LIMITED TO BACK-UP OR SHUT DOWN MECHANISMS. BECAUSE EACH END-USER SYSTEM IS CUSTOMIZED AND DIFFERS FROM NATIONAL INSTRUMENTS' TESTING PLATFORMS AND BECAUSE A USER OR APPLICATION DESIGNER MAY USE NATIONAL INSTRUMENTS PRODUCTS IN COMBINATION WITH OTHER PRODUCTS IN A MANNER NOT EVALUATED OR CONTEMPLATED BY NATIONAL INSTRUMENTS, THE USER OR APPLICATION DESIGNER IS ULTIMATELY RESPONSIBLE FOR VERIFYING AND VALIDATING THE SUITABILITY OF NATIONAL INSTRUMENTS PRODUCTS WHENEVER NATIONAL INSTRUMENTS PRODUCTS ARE INCORPORATED IN A SYSTEM OR APPLICATION, INCLUDING, WITHOUT LIMITATION, THE APPROPRIATE DESIGN, PROCESS AND SAFETY LEVEL OF SUCH SYSTEM OR APPLICATION.

Preface

Congratulations on choosing Ultiboard 9 from Electronics Workbench. We are confident that it will deliver years of increased productivity and superior board 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 Ultiboard 9, and any other Electronics Workbench products you may select.

About this User Guide

This user guide contains a general introduction to Ultiboard 9, an overview of the interface, and a series of sections that explain common functions. It also contains installation and configuration procedures and an introductory tutorial.

Online help is also available — use the **Help** menu or press F1 from a dialog box for information on that specific dialog box.

This user guide applies to all versions of Ultiboard 9. Functions that are available only in some versions are clearly marked with an icon in the left margin.

User Guide Conventions

This user guide uses the convention **Menu/Item** to indicate menu commands. For example, **File/Open** means choose the **Open** command from the **File** menu.

When this user guide refers to a toolbar button, an image of the button appears in the left column.

When this user guide says "click", it means to single-click the left mouse button. Right-clicks are clearly identified.

When this user guide refers to a "context menu", it means the menu that pops up when you right-click on an element on the screen.

When this user guide says "drag", it means to click and hold the left mouse button as you move the mouse.

License Agreement

Please read the license agreement found at www.electronicsworkbench.com 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 ten days to the place where you obtained it and your money will be refunded.

Table of Contents

1. Software Installation

2. Ultiboard Tutorial

3. User Interface

4. Beginning a Design

5. Setting Up a Design

6. Working with Parts

7. Working with Traces and Copper

8. PCB Calculators

9. Internal Router

10. Preparing for Manufacturing/Assembly

11. Viewing Designs in 3D

12. Using Mechanical CAD

Appendix A

Glossary

Chapter   1
Software Installation

This chapter contains Ultiboard installation instructions. It also includes documentation on the **Support and Upgrade Utility** (SUU) and the **Network License Server** (NLS).

Some of the features described in this chapter may not be available in your edition of Ultiboard 9. Such features have an icon in the column next to their description. Refer to the release notes for a list of the features in your edition.

The following are described in this chapter.

1.1   Installation Requirements

To successfully install Ultiboard 9, you may need up to 150 MB of hard disk space, depending on which edition you have purchased. Your system also requires the following:

Minimum System Requirements	Recommended System Requirements
Windows 2000/XP	Windows XP Professional
Pentium III Processor	Pentium 4 Processor
128 MB RAM	256 MB RAM
CD-ROM	CD-ROM
800 x 600 screen resolution	1024 x 768

1.2   Installation Overview

Ultiboard 9 has three types of installation: Single User Edition, Network Edition, and Multi-Station Standalone Edition.

Single User Edition – The Single User Edition is only licensed on the computer on which you install it. If for any reason you wish to move the software to a different computer, you must first uninstall it from the initial computer, and then re-install it onto the new computer. In this case, you must contact Electronics Workbench to receive a new Release Code.

For details on the installation process see "1.3.2.1 Installing the Single User Edition" on page 1-4".

Network Edition – Either fixed or floating licenses are available from Electronics Workbench for the Network Edition.

A floating license allows any computer to run Ultiboard 9 on a first-come, first-served basis. Maximum concurrent uses are limited to the number of seats you have purchased. A fixed license allows only a set number of specific computers to run the software. You may select and change the specific computers allowed access using the NLS Utility, up to the number of fixed licenses you have purchased (see "1.4.2 Administering Fixed Seat Licenses" on page 1-11).

See also, "1.3.4 Network Version" on page 1-5.

Multi-Station Standalone Edition – If you are installing Ultiboard 9 on various computers in a non-networked environment, then you are installing a Multi-Station Standalone Edition. This type of installation is identical to that of the Single User Edition, except that you may install a single copy of Ultiboard 9 per computer on as many computers as specified by the License Agreement. For details on the installation process see "1.3.2.1 Installing the Single User Edition" on page 1-4.

1.3   Installing Ultiboard 9

1.3.1   Before Installing Ultiboard 9

If you are upgrading to Ultiboard 9 from a previous version and wish to import your corporate or user database, you will first need to make a back-up copy of the database.

➢ To back-up your database:

1. Browse to the directory location where you have your previous version of Ultiboard (for example, C:\Program Files\Electronics Workbench\EWB9\database).

2. Copy the entire \database folder to a location that you will remember later (for example, C:\Temp).

3. Your database is now safely backed-up.

Note Please note that you will require Administrator privileges to install Electronics Workbench software and to enter release codes.

1.3.2 Single User Edition

If you purchased the Single User Edition, you have been provided with a serial number that you will be required to enter at the time of installation. The Single User Edition is only licensed on the computer on which you install it. If for any reason you wish to move the software to a different computer, you must first uninstall it from the initial computer, and then re-install it onto the new computer. In this case, you must contact Electronics Workbench to receive a new Release Code.

1.3.2.1 Installing the Single User Edition

The Ultiboard 9 CD you received will autostart when inserted in the CD-ROM drive. Follow the instructions below and on the screen during the installation process.

➢ To install Ultiboard 9:

1. Copy the Serial Number you have received with your Ultiboard 9 package.

2. Exit all Windows applications prior to continuing with the installation.

3. Insert the Ultiboard 9 CD into your CD-ROM drive. When the splashscreen appears, click on Ultiboard 9 to begin the installation.

4. Follow the on-screen prompts to complete the installation.

1.3.2.2 Requesting a Release Code for the Single User Version

Ultiboard 9 requires you to enter a Release Code within five days of the date of installation. After the five day grace period has expired, Ultiboard 9 will not run until a Release Code is entered.

To obtain your Release Code, you must provide us with your Serial Number and Signature number, as displayed on the splash screen. Contact Electronics Workbench via our website (preferred method) at www.electronicsworkbench.com and select the Product Registration link, or call Customer Service at 1.800.263.5552. Customers outside North America should contact their local distributor.

Electronics Workbench recommends that you obtain your Release Code as soon as possible after you have installed Ultiboard 9.

➢ To enter the Release Code:

1. Click on the **Enter Release Code** button at the start-up splash screen

2. If you have received your Release Code via email there are a few ways to easily enter it without the need to type each number or character one at a time. Select one of the following methods:

   • Highlight the Release Code. Drag and drop it on one of the text boxes.
   • Highlight the Release Code, right-click on it and select Copy. Click on the **Paste Release Code** button.
   • Highlight the Release Code, right-click on it and select Copy. Right-click on one of the text boxes and click on Paste from the pop-up menu.

3. If you have received your Release Code over the phone, you must type it in the Release Code fields 5 characters at a time.

4. Click **Accept** to continue.

1.3.3   Multi-Station Standalone Edition

If you purchased the Multi-Station Standalone edition, you have been provided with one serial number that you will use for each computer. You must then go to each computer on which Ultiboard is installed, and separately request a Release Code for each, which enables the software on that computer. Because of this inconvenience, and because of the fact that the software is tied to particular computers, the Network Edition is the preferred alternative in networked settings.

The Multi-Station Standalone Edition and Single User Edition installations are identical except that you will re-use the same serial number multiple times.

1.3.4   Network Version

For network versions, the client software (Ultiboard 9) may be installed:

1. On a central file server that serves the software to other networked computers (called "workstations").

2. Locally on each workstation computer. This option gives the best performance, as the software need not be accessed across a network. However, it takes up the most disk space on the workstation computers and requires updates to be installed separately on each workstation. This option can be combined with option 1, where some workstations may

have the client software installed locally, and other workstations may have the software served from a shared file system.

3. On several file servers, each one of which serves the software to a subset of the networked computers, and, optionally, also on selected workstation computers. This option is intermediate between options 1 and 2.

If options 1 or 3 are selected, the network must be set up to allow access from the workstations to the shared file systems on which the client software is installed. For all of the options above (1, 2 or 3), you must also install the Electronics Workbench Network License Server (NLS) on any one computer on the network. This computer does not need to share a file system with the client software. However, it requires TCP/IP access from the workstation computers. This computer will run a Windows service that keeps track of and limits the number of licenses currently in use. From that computer, administrators may run the NLS utility that shows information on licenses available, the users currently utilizing the licenses, and enables users to be remotely logged off.

As with the single user edition, you have been provided with a serial number that you will require during each installation of Ultiboard 9 and for the lone installation of NLS. The serial number identifies the product as being a Network Edition. Using NLS, you may access the hardware signature of the NLS server computer. You must send the NLS server's hardware signature and your serial number to Electronics Workbench. You will receive a Release Code that you will need to enter into NLS to enable the network edition to function. The Release Code encodes the number of seats and any term limits on your license.

An Ultiboard workstation will access either a fixed or a floating license depending upon the serial number entered at the time of install of the Ultiboard 9 client software that the workstation uses. In either the fixed or the floating case, the Ultiboard 9 client software may be installed either directly on the workstation computer, or onto a shared file server. Different serial numbers are provided to you for each of these cases.

Note NLS is only licensed on the computer on which you install it. If for any reason you wish to move the NLS service to a different computer, you must first uninstall it from the initial computer, and then re-install it onto the new computer. In this case, you must contact Electronics Workbench to receive a new Release Code.

1.3.4.1 Installing the Network Edition

The Ultiboard 9 CD you received will autostart when inserted in the CD-ROM drive. Follow the instructions below and on the screen during the installation process.

➢ To install Ultiboard 9:

1. Copy the Serial Number you have received with your Ultiboard 9 package.

2. Exit all Windows applications prior to continuing with the installation.

3. Insert the Ultiboard 9 CD into your CD-ROM drive. When the splashscreen appears, click on Ultiboard 9 to begin the installation.

4. Follow the on-screen prompts to complete the installation.

1.3.4.2 Entering the Release Code for the Network Edition

After installing NLS, you will be required to enter a Release Code to enable the workstations to run Ultiboard 9. (NLS is installed at the same time as the Ultiboard 9 Network Edition).

To perform this step, you will need to use the NLS Utility that you have previously installed on your license server. See "1.4 Network License Server" on page 1-10 for further information.

Note To obtain the Release Code, contact Electronics Workbench via our website (preferred method) at www.electronicsworkbench.com and select the Product Registration link, or call Customer Service at 1.800.263.5552. Customers outside North America should contact their local distributor.

1.3.4.3 Workstation Setup

If for your Network Installation you choose to serve Ultiboard 9 from a file server to the workstation computers, you will need to perform an extra setup step on each workstation computer as follows.

1. Log on to each workstation with administrator privileges.

2. Using Windows Explorer, navigate to the **<install-root>\EWB9\UBSetup** folder on the computer drive where Ultiboard 9 was installed on the network.

3. Double-click on setup.exe to install the shortcuts to the software on the computer. The setup routine will configure the workstation.

Access to Ultiboard 9 directories should be restricted by placing certain user permissions on various directories. Follow the instructions below to place the appropriate permissions.

1.3.4.4 Setting User Permissions

If you are installing Ultiboard 9 Network Edition, you will need to set certain restrictions on the folder where Ultiboard is installed in order to prevent non-administrative users from modifying, writing or deleting program files that otherwise would make the software unusable.

Note that there are many different ways to set permissions in a networked environment. The following settings are one way of configuring a Windows XP Professional-based computer that is not part of a network domain (i.e., using workgroup sharing). It is recommended that

you have advanced knowledge of NTFS and share permissions. Contact your administrator for help.

Ultiboard 9 Permissions on a Windows XP Professional-based Computer

1. Browse to **<install-root>\EWB9,** where the Ultiboard 9 Network Edition is installed on the server.

2. Right-click on the **\EWB9** folder and click on "**Sharing and Security**" from the pop-up menu.

3. Click on the "**Sharing**" tab on the EWB9 Properties dialog and select "Share this folder".

4. Click on the "**Permissions**" button. The "Permissions for EWB9" dialog will pop-up.

5. Under "Groups or user names", select the security group that requires access to Ultiboard 9. (If everybody on the network is allowed access, select "Everyone").

6. Set the following permissions:

Permission	Allow	Deny
Full Control	X	
Change	X	
Read	X	

All other options shown on the dialog should be unchecked unless specified on the above table. Click **OK** to accept the permissions.

7. Click on the **Security** tab and select the appropriate group under "Group or user names".

8. Click on the "**Advanced**" button. The "Advanced Security Settings for EWB9" dialog will pop-up.

9. On the **Permissions** tab, click on the "Edit" button. The "Permissions Entry for EWB9" dialog will pop-up.

10. Set the following permissions:

Permission	Allow	Deny
Read & Execute	X	
List Folder Contents	X	
Read	X	

All other options shown on the dialog should be unchecked unless specified on the above table.

Under the "Advanced" tab, make sure that the option "**Inherit from parent the permission entries that apply to child objects. Include these with entries explicitly defined here**" is unchecked.

Place a check mark on "**Replace permission entries on all child objects with entries shown here that apply to child objects**". Click **OK** to accept the permissions.

11. Click **OK** on the "EWB9 Properties" dialog. You have now set up the EWB8 folder directory and all sub-directories and files with "Read & Execute" permissions.

Certain files and sub folders need to also be given "Write" permissions for Ultiboard 9 to function properly on a networked environment.

Please follow the instructions below:

1. Browse to **<install-root>\EWB9\database** folder.

2. Right-click on the **\EWB9\database** folder and click on "**Sharing and Security**" from the pop-up menu.

3. Click on the **Security** tab and select the appropriate group under "Group or user names".

4. Under "Permissions" add a check mark for "Modify" and "Write" under the "Allow" column. Click **OK** to accept the permissions on that folder.

5. Repeat Step 2-4 for the "**Ultiboard users**" and "**Ultiboard users**" folder.

1.3.5 Changing Server Name and/or Port Number After Client Installation

If after having installed Ultiboard 9 on workstations and/or file servers it becomes necessary to change the server name or port number, it is possible to do this without re-installing the client software. On each Ultiboard 9 installation, navigate to the Settings sub-directory (by default located in **'C:\Program Files\Electronics Workbench\EWB9\Settings'**) and use a text editor (such as Notepad) to edit the file **Ultiboard.ini.** Change the entries next to Port and Server.

1.4 Network License Server

The **Network License Server** (NLS) is used to administer network installations. If you have installed a network version of any Electronics Workbench software, the **Network License Server** will automatically run in the background whenever the computer on which it is installed is operating.

1.4.1 Administering the Network License Server

➤ To administer the **Network License Server**:

1. Click Start > All Programs > Electronics Workbench > Network License Server > Network License Server.

   Or

   Double-click on the short-cut icon that was placed on your desktop during installation.

2. Click **New** to add a product. The **Add Product** dialog box appears.

3. Enter a valid serial number and click **OK**.

   Tip If you have the serial number recorded electronically (for instance, in an email), you can drag-and-drop it into the field in the **Add Product** dialog box.

4. Right-click on the Release Code line and select Edit Release Code from the pop-up.

5. You can either copy the release code and click **Paste Release Code**, drag-and-drop the release code into any of the **Release Code** fields, or type it in manually.

   Note For instructions on obtaining the release code, see "1.3.4.2 Entering the Release Code for the Network Edition" on page 1-7.

6. Click **Save** to return to the main dialog box. The release code has been added to the Release Code line.

7. When network seats are being used, the dialog will appear similar to the following:

Right-click to edit release codes, edit serial numbers or remove the product

Right-click to edit serial number

Right-click to edit release code

Right-click to log-off all users

Right-click on individual users to log them off

EWB Network License Server

Electronics Workbench Network License Server
Computer Hardware Signature: 32610-60900
IP: 10.111.11.73
Name: JOHNXP
* Port: 1289
Multisim 8 Power Pro Network - 1 of 4 in use
Product name: Multisim 8
Tier: Power Pro Network
* Serial: PP-Net-4011-7000-5410-0707-90
* Release Code: c7fuc-xxtc4-8sufw-emud4-d8wsd-4cxja-3udy8-k4k3w-310hd-w0j43-nxdpa-y8ucc
Expiry Date: 01 March 2008
* Seats: 1 of 4 currently in use
John - JOHNXP - 10.111.11.73 - 0.0 hours - 3:31:49 PM

CLOSE   HELP   RESTART   NEW

Click to restart the Network License Server software.   Click to add a new product.

1.4.2 Administering Fixed Seat Licenses

This section gives additional information needed for the administration of Fixed Seat Licenses.

➢ To add (authorize) a seat signature:

1. Go to the computer that contains the client software that you wish to add (for example, Multisim) and launch the software. As this software is not yet authorized, a message displays indicating that the computer's signature was not recognized by the server. Copy down the signature indicated in the message.

2. From NLS, right-click on **Authorized Seat Signatures** and select **Authorize a new computer** from the pop-up that displays.

3. When prompted, enter the signature that you copied down from the client computer and click **OK**.

Authorized and Available Seats

Number of seats logged in

Number of Authorized Seats

* Fixed seats: 0 of 3 currently in use
☐ * Authorized Seat Signatures - 3 of 4   Maximum Seats Available
    31572-53417
    55373-25957
    67639-01722
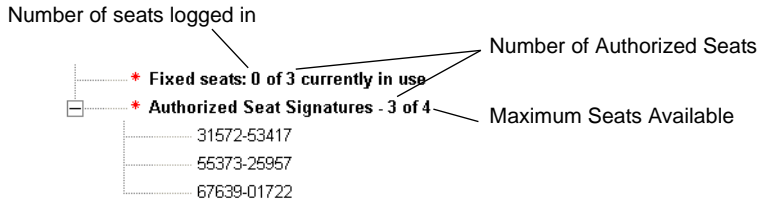
1.4.3   Reviewing License Server Events

The **Network License Server** records all client connections or attempted connections in the system event log. To access the event log, go to the **Windows Control Panel**, select **Administrative Tools**, and then select **Event Viewer**.

In the left-hand pane of the **Event Viewer**, choose **Application Log**. All events with the source **EWBNLSS** are from the **Network License Server**. Double-click on any event to see more detail.

1.4.4   Troubleshooting

The following contains solutions to situations that may be encountered with the **Network License Server**.

• **Network License Server gives "Permission Denied" message.**

For security reasons, the **Network License Server** may only be run from an account with administrator privileges. Attempting to control the license server from a "user" or "power user" account will cause a permission denied error.

• **Client application gives "Connection to the license server failed" message.**

Make sure that the server address and port number on the client **Network License Server** dialog have been set correctly. If you have entered a machine name as the server address, try using the numeric IP address instead. In some networks, this IP address may change dynamically. Please contact your network administrator for assistance if this occurs frequently.

If either the server machine or client machine has a firewall installed (including the Windows XP service pack 2 internal firewall), make sure that either:

a) the required port is allowed to be open (in TCP protocol) or

b) the server program (EWBNLSS) or client application is granted an exception.

Please see your firewall's documentation for more information.

- **Client application gives "This product is not registered on the license server" message.**

  Make sure that you have used the same product serial number on both the client and the server.

  Make sure that the registration code has been accepted by the license server. No client application will be licensed until the product has been successfully registered.

- **Client application gives "No more instances of the program can be run on the network" message.**

  This message is returned when all available license seats have been used up. When a user shuts down the client application normally, their seat should be released immediately; however, when a client application terminates unexpectedly (the computer is turned off while running or there is a fatal error), the license server may take up to five minutes to relinquish this license for reuse. If you believe that there should be seats available and clients are being denied licenses, waiting for five minutes to log on is usually sufficient to allow these seats to be freed.

  The administrator may also wish to check the **Network License Server** control screen and try to identify unused but unreleased licenses by their computer name or IP address subdomain. If the administrator logs off these users, their licenses will be immediately available for new clients. The administrator may also check for users running multiple clients on a single machine.

  In extreme cases, the administrator may click the **Restart** button on the **Network License Server**. This will immediately release all licenses and then reissue them silently to active authorized users only. For most client users, this will be an invisible process. This option should however be used with caution: if the number of clients requesting licenses is indeed over the seat limit, the license server cannot guarantee that exactly the same set of clients will be granted licenses as had them before the reset operation. Therefore, some authorized clients may be given the "no more instances of the program can be run on the network" message in the middle of their active session and be prohibited from continuing their work (however, they will be given the opportunity to save their work).

1.5   Support and Upgrade

The Support and Upgrade program lets you call Electronics Workbench with your technical questions. The program also entitles you to no-charge upgrades to the software as new versions are released.

Between major upgrade releases, Electronics Workbench puts out a series of patches that may add incremental functionality, add new parts to the database, or quickly address any issues found in the field.

If you allow it to (a choice you initially make upon install and can later change), Multisim or Ultiboard will quickly and silently check the Electronics Workbench website for the availability of new patches and upgrades when you start the application. If any are available, it will open the Support and Upgrade Utility (SUU) program that will allow you to download and install all the necessary patches to get you to the most current version of your product.

1.5.1   Checking for Updates

Unless you selected **I will check for updates and messages manually** during the installation of your software (e.g., Multisim, Ultiboard), SUU checks for updates when you launch that software. You can also check for updates at any time by following the procedure below.

➢ To check for updates:

1. Select **Help/Check for Updates**. SUU launches and checks for updates.

2. If updates are available, it is indicated in the **Support and Upgrade Utility** window. You can click on the **Release Notes** link for a description of the upgrade.

Note   A message advising that your software is up-to-date appears if there are no updates available.

3. Proceed to "1.5.2 Installing Updates" on page 1-14.

1.5.2   Installing Updates

Use the procedure below to install updates. To determine if updates are available, see "1.5.1 Checking for Updates" on page 1-14.

Most users will install all available upgrades. You can also upgrade to a specific version. This option should only be used to ensure parallel versions of software are being run within an organization or institution.

Installing all Updates

➢ To install all available updates:

1. From the **Support and Upgrade Utility**, click on **Upgrade to Latest Version**.

   Patch information appears as the download progresses. When all downloads have been made the install process begins.

2. Follow the onscreen prompts (if available) to complete the upgrade.

Note   Some patches may be configured to install without user input.

Upgrading to a Specific Version

➤ To upgrade to a specific version:

1. From the **Support and Upgrade Utility**, click on **Advanced Options**.

2. Click on a button in the **Upgrade to Specific Version** column.

3. Click **Yes** when prompted to confirm the upgrade.

   Patch information appears as the download progresses.When all downloads have been made the install process begins.

4. Follow the onscreen prompts to complete the upgrade.

1.5.3   Viewing Messages

➤ To view messages:

1. Select **Help/Check for Updates**. SUU launches and checks for updates.

   (For instructions on what to do if there are available updates, see "1.5.2 Installing Updates" on page 1-14).

2. Click on **more info...** in the **Messages** area, beside the message of interest. The full message appears.

3. You can also click on **Message History** to display a history of received messages.

1.5.4   Changing Settings

The initial settings for SUU are done during Ultiboard's installation procedure. If you would like to change these settings, follow the procedure outlined below.

➤ To change the Support and Upgrade Utility's settings:

1. Select **Help/Check for Updates**. SUU launches and checks for updates.

   (For instructions on what to do if there are available updates, see "1.5.1 Checking for Updates" on page 1-14).

2. Click on the **Settings** button.

   Note   The **Language** field applies to messages only. The sofware's language is not affected by this setting.

3. Select the desired options and click **OK**.

1.6   Uninstalling Ultiboard 9

1.6.1   Uninstalling the Single User Version

1. Ensure you have recorded the serial number before uninstalling Ultiboard.
2. Click the Windows **Start** button.
3. Click **Control Panel**.
4. Click **Add or Remove Programs**. The **Add or Remove Programs** dialog appears.
5. From the list, select **Ultiboard 9** and select **Remove**. Ultiboard 9 will be removed from your computer.

1.7   Uninstalling SUU

➢ To uninstall SUU:
1. Click the Windows **Start** button.
2. Click **Control Panel**.
3. Click **Add or Remove Programs**. The **Add or Remove Programs** dialog appears.
4. From the list, select **EWB Support and Upgrade Utility** and select **Remove**. SUU will be removed from the standalone workstation.

1.8   Uninstalling a Site Version

1.8.1   Uninstalling Standalone Multi-Station Installation

At each standalone workstation, perform the following procedure:
1. Ensure you have recorded the serial number before uninstalling Ultiboard.
2. Click the Windows **Start** button.
3. Click **Control Panel**.
4. Click **Add or Remove Programs**. The **Add or Remove Programs** dialog appears.
5. From the list, select **Ultiboard 9** and select **Remove**. Ultiboard 9 will be removed from the standalone workstation.

1.8.2   Uninstalling Network Installation

Only the workstation that was used to install Ultiboard 9 to the network directory (host computer) can be used to remove it. Performing the steps below on other workstations (client computer) will only remove the Ultiboard 9 folder.

➢ To uninstall Ultiboard 9 from the network directory or the Ultiboard 9 folder:

1. Ensure you have recorded the serial number before uninstalling Ultiboard.

2. Click the Windows **Start** button.

3. Click **Control Panel**.

4. Click **Add or Remove Programs**. The **Add or Remove Programs** dialog appears.

5. From the list, select Ultiboard 9 and select **Remove**. If the workstation is the host computer, Ultiboard 9 is removed from the network directory. If the workstation is only a client computer, the Ultiboard 9 folder is removed.

1.8.3   Uninstalling Combination Standalone Multi-Station and Network Installations

Follow the instructions in "1.8.1 Uninstalling Standalone Multi-Station Installation" on page 1-16 and "1.8.2 Uninstalling Network Installation" on page 1-17, as necessary.

1.9   Uninstalling NLS

➢ To uninstall NLS:

1. Click the Windows **Start** button.

2. Click **Control Panel**.

3. Click **Add or Remove Programs**. The **Add or Remove Programs** dialog appears.

4. From the list, select **EWB Network License Server** and select **Remove**. SUU will be removed from the standalone workstation.

Chapter   2
Ultiboard Tutorial

The tutorial in this chapter places the components and traces for the tutorial circuit that is discussed in the *Multisim 9 User Guide*.

Useful tips are indicated by an icon in the left column.

The following are described in this chapter.

2.1   The Electronics Workbench Suite

Electronics Workbench provides a suite of EDA (Electronics Design Automation) tools that assist you in carrying out the major steps in the circuit design flow. The suite consists of the following major components:
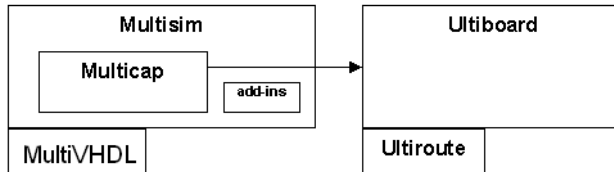
Multicap is a schematic capture program suitable for pure schematic entry, driving simulation, and feeding to downstage steps, such as PCB layout. *Multisim* includes all of Multicap and adds mixed analog/digital simulation capability. *MultiVHDL* adds HDL model creation and co-simulation to Multisim. Depending on your tier of Multisim, optional add-ons may be available, such as for RF simulation and extended parts libraries.

Ultiboard, fed from Multicap or Multisim, is used to design printed circuit boards, perform certain basic mechanical CAD operations, and prepare them for manufacturing. *Ultiroute* is an add-on to Ultiboard that provides automated parts placement and layout beyond what is available in Ultiboard stand-alone.

All of the products in the Electronics Workbench suite are available in multiple tiers depending upon budget and needs.

2.2   Opening the Tutorial

➢ To open the tutorial file:

1. Launch Ultiboard and select **File/Open**.

2. Navigate to the `tutorial` folder found in your Ultiboard installation (for example, `C:\Program Files\Electronics Workbench\EWB9\samples\tutorial`), select Tutorial.EWPrj and click **Open**. The project file is loaded into Ultiboard.
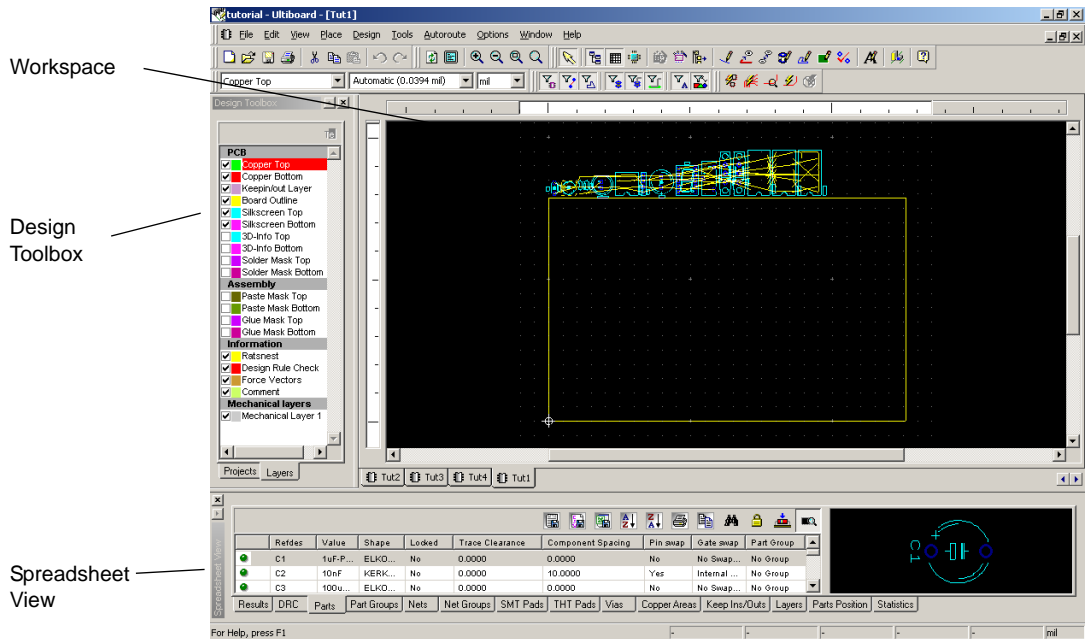
3. To select a design (for example, Tut1) either click on its tab, or click on its name in the **Projects** tab of the **Design Toolbox**.

2.3   Creating a Board Outline

You can create a board outline in one of the following ways:

- draw a board outline using the drawing tools (you can press the * key on the numeric keypad to enter the specific coordinates for the outline)
- import a DXF file
- use the **Board Wizard**.

➢ To experiment with the **Board Wizard**:

1. Double-click on **Board Outline** in the **Layers** tab.

2. Click on the existing board outline in the Tut1 design and press DELETE.

3. Choose **Tools/Board Wizard**.
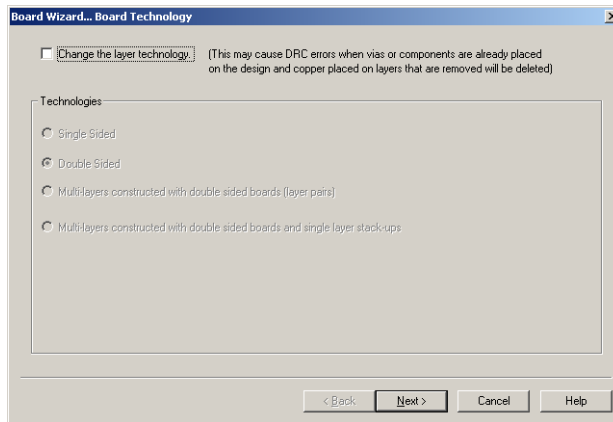
4. Enable the **Change the layer technology** option to make the other options available.

5. Choose **Multi-layers constructed with double sided boards and single layer stack ups**, and click **Next**.

6. The next dialog box allows you to define the **Lamination Settings** for the board. (For this tutorial, the settings will not be changed) .
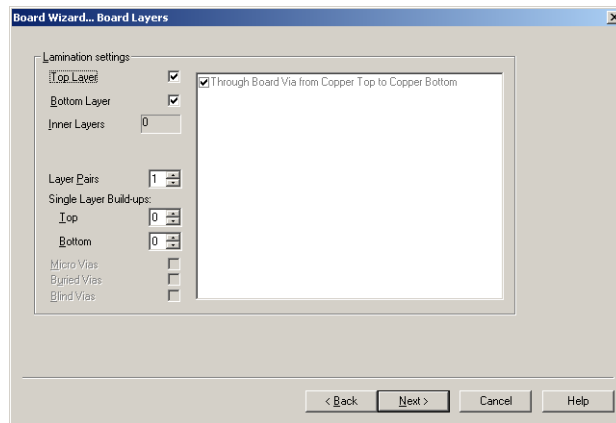
7. Click **Next**.

   In the **Shape of Board** dialog box:

   - make sure the **Reference Point** is set to **Left-Bottom** for **Alignment**.
   - make sure the **Rectangular** option is selected
   - set the **Width** to 2500 and the **Height** to 2000
   - set the **Clearance** to 5.00000. This is the distance from the edge of the board that is to be kept free of any other elements.
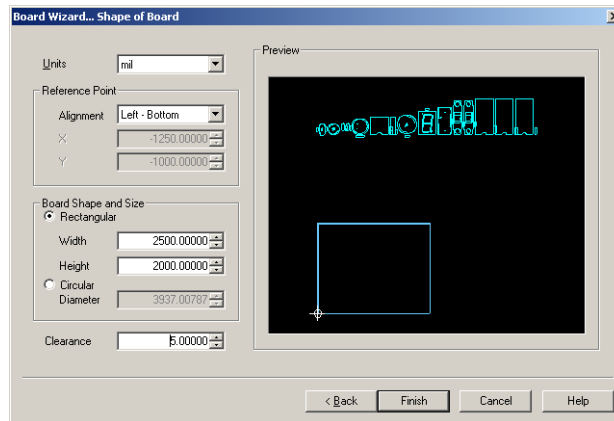
8. Click **Finish**. The board outline is placed on your design.

Note For complete details on the **Board Wizard**, see "5.2.4 Using the Board Wizard" on page 5-7.

➢ To move the board outline:

1. Double-click on **Board Outline** in the **Layers** tab.

2. Click anywhere on the board outline in the workspace and drag the board to a location just below the row of components.

➢ To change the reference point:

1. Select **Options/Set Reference Point**. The reference point is attached to your cursor.

2. Move the cursor the the lower-left corner of the board outline and click to place it.

New reference point.   New board outline.
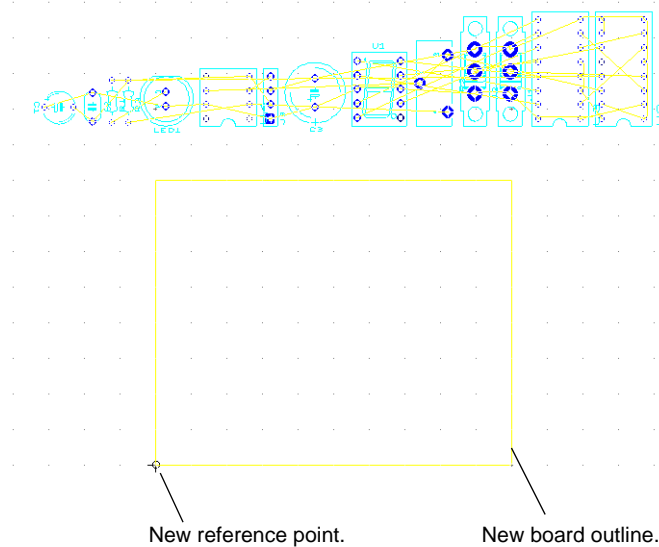
2.4   Placing Components

You can place components on your Tut1 design file in several different ways:

• select one or more components from outside the board outline and drag them into place
• use the **Parts** tab in the **Spreadsheet View** to locate components and place them
• select parts from the database.

You can use the **Place/Unplace Components** command to quickly remove all non-locked components from the PCB and experiment with a different placement technique.

2.4.1 Dragging Components from Outside the Board Outline

By default, components are placed outside the board outline when you open a netlist from Multisim or another schematic capture program. Before you begin, double-click the **Copper Top** layer in the **Design Toolbox** to make it the active layer.

➢ To drag U1 from outside the board outline:

1. Find U1 in the collection of components outside the board outline. To make this easier, zoom in (press F8) until you can see U1.

You can also search for a part with the **Edit/Find** command. While this command works much like a Find function in other applications, it also allows you to search for a part by name, number, shape, value, or by all variables. For more information, see "4.12 Searching for Design Elements" on page 4-11.

2. Click on U1 (the 7-segment display) and drag it to the center of the board.

Force Vector. For details, see "6.1.3.2 Working with Force Vectors" on page 6-6.

Ratsnest. For details, see "6.1.3.1 Working with Ratsnests" on page 6-5.
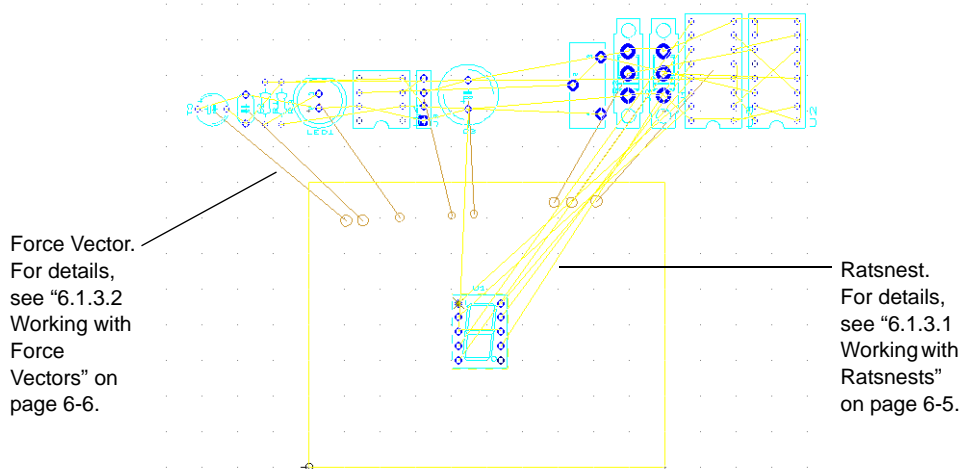
U1 remains selected. This is an important point for Ultiboard that holds throughout the application — you need to explicitly end any particular action. In this case, simply clicking somewhere else de-selects the component. Right-clicking also ends the current action.

3. Go to the **Parts** tab in the **Spreadsheet View** and scroll to U1. You will notice that the green light beside the component is slightly brighter — this indicates that the component has been placed.

2.4.2 Dragging Components from the Parts Tab

➢ To drag components from the **Parts** tab:

1. In the **Parts** tab, scroll down until you see J3.

	Refdes	Value	Shape	Locked	Trace Clearance	Component Spacing	Pin swap	Gate swap	Part Group ▲
●	J1	SPDT	SW_S3P	No	10.0000	0.0000	No	No Swap...	No Group
●	J2	SPDT	SW_S3P	No	10.0000	0.0000	No	No Swap...	No Group
●	J3	HDR1X4	HDR1X4	No	0.0000	0.0000	No	No Swap...	No Group
●	LED1	LED_blue	LED9R2...	No	0.0000	0.0000	No	No Swap...	No Group
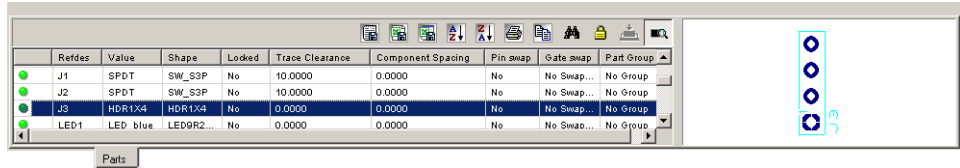
Parts

2. Click on J3 and drag it from the **Parts** tab onto the workspace. J3 is attached to your mouse pointer.

3. Drop J3 on the left side of the board, roughly in the middle. As before, in the **Parts** tab J3's green light is slightly brighter, indicating that the component has been placed.

For even more rapid placement of parts, in the **Parts** tab select an unplaced part (its green light is dim) and click the **Start Placing the Unpositioned Parts** button. Ultiboard systematically goes through the list of components in the **Parts** tab, selecting each one and attaching it to your mouse pointer so you can place it, then selecting the next one on the list.

2.4.3 Placing the Tutorial Components

Using any method or combinations of methods, make your layout look like the illustration below. You can also, if you prefer, simply open the next design file in the Tutorial project, Tut2, which has already been set up this way. To open Tut2, click the **Projects** tab in the **Design Toolbox** and click "Tut2".

Your design should look like this:
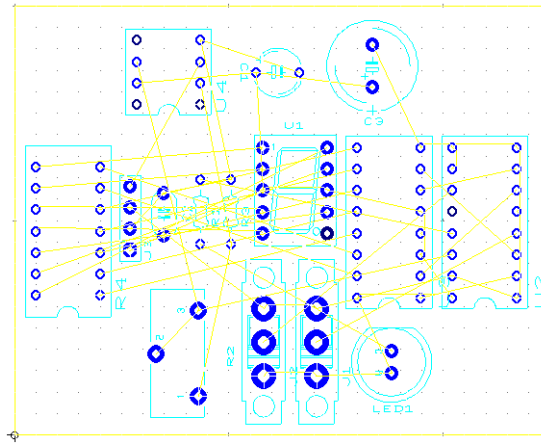
2.4.4   Placing Parts from the Database

In addition to placing parts imported as part of your design file, you can place parts directly from the database. We will use this procedure to place the mounting holes.

➢ To place parts from the database:

1. Choose **Place/From Database**. The **Get a part from the Database** dialog box opens:

2. In the **Database** panel, expand the **Through Hole Technology Parts** category and navigate to the **Holes** category. The parts appear in the **Available Parts** panel.

3. In the **Available Parts** panel, select the Hole35 part. The part is previewed in the **Preview** panel.
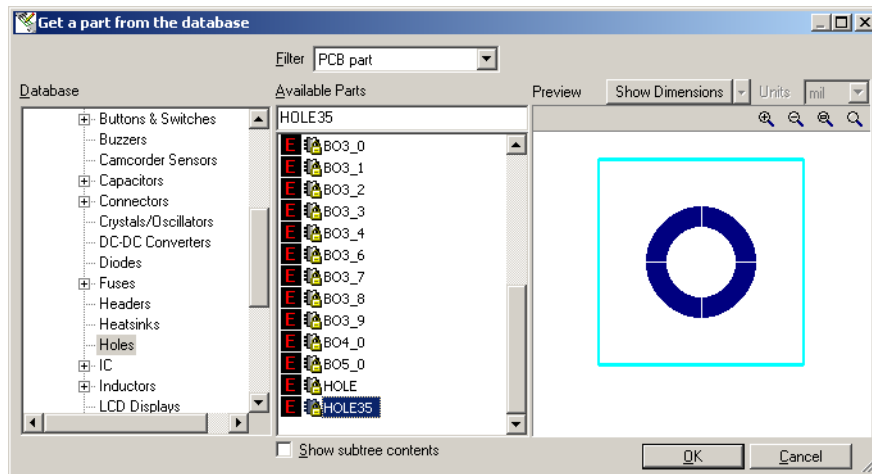
4. Click **OK**. The **Get a part from the database** dialog box disappears, and you are prompted to enter the **RefDes** and **Value**.
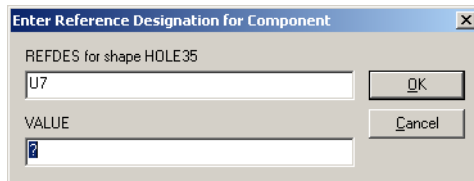
5. Enter the hole's reference designator (H1) and value (HOLE) and click **OK**.

6. Move the pointer over the board. The part is attached to the pointer.

7. When the hole is in position in the top left corner, click to drop it on the board.

8. The **Enter Reference Designation for Component** dialog box reappears, with the reference designator automatically incremented to H2. Click **OK** to place the next mounting hole in the top right corner, and repeat to place H3 in the bottom right corner, and H4 in the

bottom left corner. Click **Cancel** to stop, and click **Cancel** again to close the **Get a part from the database** dialog box.
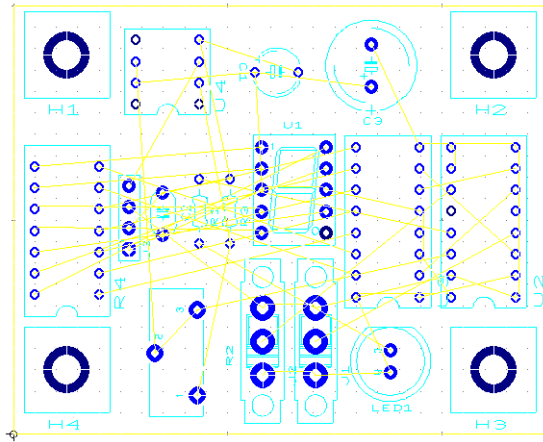
2.4.5   Moving Components

You can use the same methods for moving components as you do for placing them. To select a component already on the board, simply click on it. To specify the X/Y coordinates to which the part is to move, press the * key on the numeric keypad. Alternatively, in the **Parts** tab, select a placed component (indicated by a brighter green light beside it) and drag it to a new location.

The component's label is a separate element from its footprint. When selecting a component on the board, be sure to select the whole component, not just the label.

Once a component is selected, you can also move it around on the board by pressing the arrow keys on your keyboard.

When you move a component that has connected traces, its connections will be maintained — this is called **Rubber Banding**. (The optional Ultiroute application must be installed). For details, see " Rubber Banding" on page 6-7.

You can also select a group of components and move them together. To do this, you can do one of the following:

• hold down the SHIFT key and click on more than one component
• drag a box around several components to be selected.

All the selected components will move together when you drag the cursor.

These are temporary groups — once you select another component, the group connection is lost. To make a group that remains until you remove it, you can use **Tools/Group Editor**. For details, see "5.5 Working with the Group Editor" on page 5-14.

Component shoving allows you to move one component and have Ultiboard automatically push other components on the board out of the way to create enough available space for the component. Turn shoving on by choosing **Options/Part Shoving**.

Another option for moving components is to use the **Edit/Align** commands to align the edges of selected components or to space them relative to each other. Use the **Edit/Align** commands to align the mounting holes you just placed.

1. Select H1 and hold down the SHIFT key to select H2.

2. Choose **Edit/Align/Align Top**. If H2 was originally not placed exactly in line with H1, you will see it move.

3. Click on an empty space on the board, then select H2 and H3.

4. Choose **Edit/Align/Align Right**.

5. Continue in this manner to align the bottoms of H3 and H4, and the left sides of H1 and H4.

2.5   Placing Traces

This section discusses a number of options for trace placement.

2.5.1   About Component Connections

A net is a collection of connected pins. It has an assigned name and carries connectivity information on all the pins to which it connects. Ultiboard uses netlist information to:

• guide you as to which pins need to be connected with traces
• warn you when you connect a trace between pins that are not part of the same net
• inform the autorouter of where it needs to place traces when autorouting.

2.5.2   Options for Placing Traces

You have the following options for placing traces:

• manual trace
• follow-me trace
• connection machine.

A manual trace is placed exactly as you specify, even running through a component if that is the path you set out. A follow-me trace automatically draws a legal trace between the pins you select with your mouse movements—you can move from pin to pin, leaving a legal trace. A connection machine trace automatically joins two pins by the most efficient route, though you have the option of changing it.

As you place a trace, and before you click to fix it in place, you can always remove a segment by backing up over it. Each time you click while placing a manual trace, or each time a follow-me trace or connection machine trace changes direction, a separate segment of that trace is created. When performing operations on traces, be sure to select either the appropriate segment or, if you wish, the whole trace.

2.5.3　Placing a Manual Trace

You can continue with the design you have been working on, or open Tut3. Be sure you are on the **Copper Top** layer before beginning.

If necessary, press F7 to show the whole design.

➢ To place a trace manually:

1. Choose **Place**/**Line**.

The **Line** command is used to create a line on any layer. The results differ depending on the layer selected. For example, if the selected layer is silkscreen, you will create a line on the silkscreen layer of the PCB. If the selected layer is a copper layer, then the "line" is actually a trace.

2. Locate J3, at the left-hand part of the board. Find the bottom pin, as shown below:

Part J3

Start with
this pin.
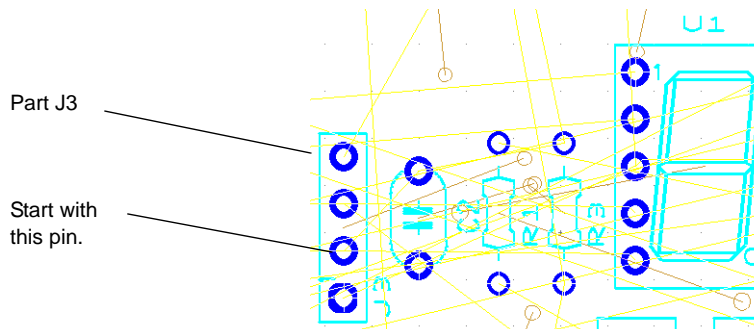
If you have trouble locating the component, use the **Find** function of the **Parts** tab. Select the component in the **Parts** tab, then click the **Find and select the part** button. The component is shown in the workspace. If necessary, zoom in further using F8.

3. Click on the pin specified in the above step. Ultiboard highlights all the pins that are part of the same net as the pin you clicked on. (The color of the highlighting can be changed in the **Colors** tab of the **Preferences** dialog box). This is how you know where to connect pins to match the connectivity from your schematic.

Pins that are part of the same net are indicated with an X.
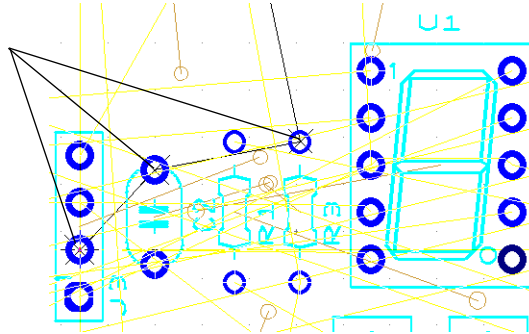
4. Move the cursor in any direction. A green line (the trace) is attached to the selected pin. Each time you click you anchor the trace segment.

5. Click on the destination pin.

Destination pin

Click to fix trace before changing direction
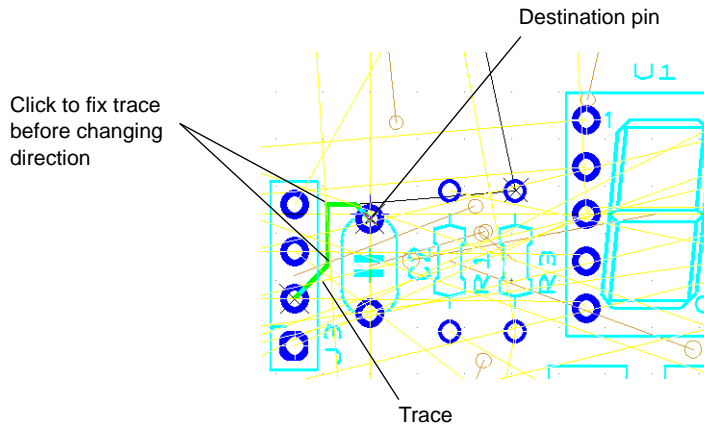
Trace

6. Right-click to stop placing traces.

2.5.4 Placing a Follow-me Trace

➢ To place a follow-me trace:

1. Choose **Place/Follow-me**.

2. Click on the top pin of J3.

3. Click on the second pin in the right column of U4.

4. Ultiboard draws the connection for you.
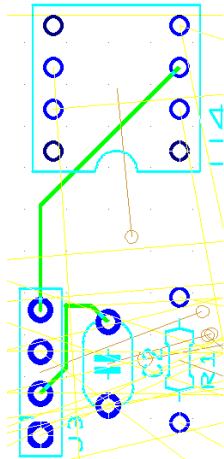
You do not need to click exactly on a pin — you can also start by clicking on a ratsnest line.

2.5.5 Placing a Connection Machine Trace

➢ To place a **Connection Machine** trace:

1. Choose **Place/Connection Machine**.

2. Click on the segment of the ratsnest indicated below.
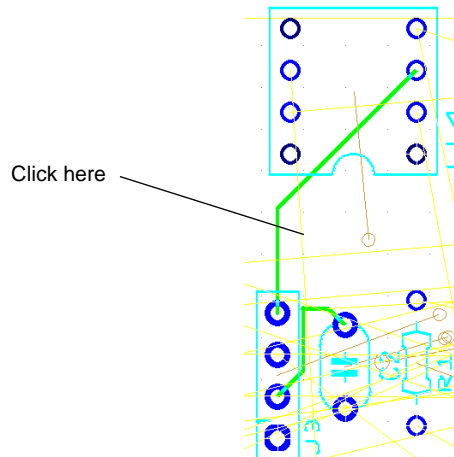
3. As you move your cursor, Ultiboard suggests various traces routed around obstacles.

4. When you see the route you want, click to fix the trace. You don't have to click on the ratsnest or the destination pin.

New trace segments appear between these pins
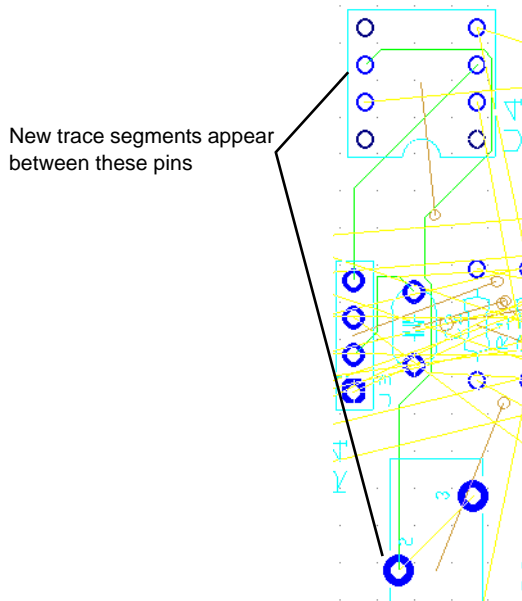
5. Right-click to end trace placement.

2.5.6   Net Bridges

Use the net bridge functionality to make connections between different nets (e.g., digital and analog grounds) without losing the properties of either net.

For details, see "7.6.4 Net Bridges" on page 7-36.

2.5.7   PCB Transmission Line Calculator

You can use the **PCB Transmission Line Calculator** to calculate the following parameters for typical printed circuit board trace geometries:

• Characteristic Impedance (Zo)
• Per unit length Capacitance (Co)
• Per unit length Inductance (Lo)
• Propogation Delay (tpd)

For details, see "8.1 PCB Transmission Line Calculator" on page 8-1.

2.5.8   PCB Differential Impedance Calculator

The **PCB Differential Impedance Calculator** performs calculations for two traces that carry signals that are exactly equal and opposite (a differential pair).

You can use the **PCB Differential Impedance Calculator** to calculate the following parameters for differential pairs:

- Characteristic Impedance (Zo)
- Per unit length Capacitance (Co)
- Per unit length Inductance (Lo)
- Propogation Delay (tpd)
- Differential Impedance (Zdiff)

For details, see "8.2 PCB Differential Impedance Calculator" on page 8-12.

2.6   Preparing for Manufacturing/Assembly

This section explains the functions performed to output your board for production and documentation purposes. Ultiboard 9 can produce many different output formats to support your production and manufacturing needs.

2.6.1   Cleaning up the Board

Before sending the board for manufacturing, you should clean up any open trace ends (trace segments that do not have any terminating connections in the design) and unused vias that have been left on the board.

➢ To delete open trace ends, make sure the Tut4 design is open and choose **Edit/Copper Delete/ Open Trace Ends.** This deletes all open trace ends in the design.

➢ To delete any unused vias, make sure the design is open and choose **Design/Clean Unused Vias** to delete all vias that do not have any trace segments or copper areas connected to them.

2.6.2   Adding Comments

Adding a comment permits "redlining", which can be used to show engineering change orders, to facilitate collaborative work among team members, or to allow background information to be attached to a design.

You can "pin" a comment to the workspace, or directly to a component. When a component with an attached comment is moved, the comment also moves.

For details, see "10.3 Placing a Comment" on page 10-5.

2.6.3 Exporting a File

Exporting a file refers to producing an output from Ultiboard in a format that can be understood by the board manufacturer. An exported file contains complete information describing how a finished board is to be manufactured. Files that can be exported include Gerber RS-274X and RS-274D files.

For complete details, see "10.9 Exporting a File" on page 10-10.

2.7 Viewing Designs in 3D

Ultiboard lets you see what the board looks like in three dimensions at any time during the design. For complete details, see " Viewing Designs in 3D" on page 11-1.

You can use the **Exploded View** to look between the layers of a multi-layer PCB. For details, see "11.2.3 Exploded View" on page 11-5.

Chapter 3
User Interface

This chapter explains the basic components of the Ultiboard Graphical User Interface (GUI) and shows how to set up preferences and PCB properties.

The following are described in this chapter.

3.1 Introduction to the Ultiboard Interface

Ultiboard's user interface is made up of several elements.
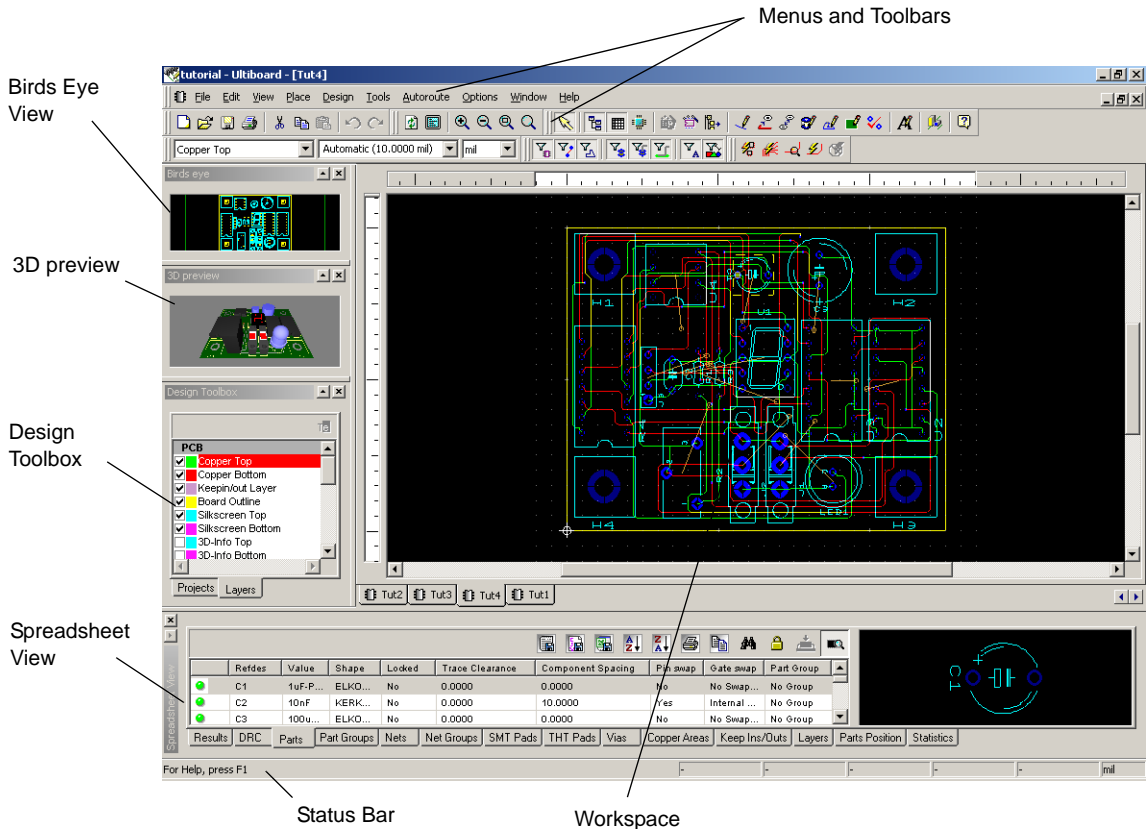
The **Birds Eye View** shows you the design at a glance and lets you easily navigate around the workspace.

The **3D Preview** shows you a three-dimensional preview of the board.

The **Design Toolbox** lets you show, hide, or dim elements of your design.

The **Spreadsheet View** allows fast advanced viewing and editing of parameters including component details such as footprints, Reference Designators, attributes and design constraints.

The **Status Bar** displays useful and important information.

The **Workspace** is where you build your design.

The **Menus** and **Toolbars** give you access to the Ultiboard commands.

3.2   Toolbars

The toolbars provide a quick, convenient way for you to access the most common Ultiboard 9 functions. You can toggle toolbars on and off using the **View/Toolbars** menu.

3.2.1   The Standard Toolbar

The **Standard** toolbar contains buttons for basic editing functions, and appears by default when you run Ultiboard 9.

The **Standard** toolbar buttons are described in the table below:

	New file	Creates a new project (if none are currently open) or a new design if a project is currently open. For details, see "4.2 Creating a Project" on page 4-2 or "4.3 Creating a Design" on page 4-3.
	Open file	Opens an existing project. For details, see "4.6 Opening an Existing File" on page 4-6.
	Save file	Saves the active design to its current name and directory. For details, see "4.7 Saving and Closing" on page 4-7.
	Print	Displays the Print dialog. For details, see "10.10 Printing your Design" on page 10-20.
	Cut	Cuts the selected element from the design and places it on Windows Clipboard.
	Copy	Copies the selected element from the design and places it on Windows Clipboard.
	Paste	Pastes the element from Windows Clipboard to the design.
	Undo	Undoes the last action.
	Redo	Redoes the last action (used after undoing).

3.2.2   The View Toolbar

The **View** toolbar contains buttons for modifying the way the screen is displayed, and appears by default when you run Ultiboard 9.

The **View** toolbar buttons are explained in the table below:

	Redraw the screen	Redraws the currently active design workspace. For details, see "4.13.3 Refreshing the Design" on page 4-13.
	Toggle Full Screen	Adjusts the size of the workspace so it displays the entire design. For details, see "4.13.1 Using the Full Screen" on page 4-12.
	Zoom In	Zooms in on the design, providing a closer view. For details, see "4.13.2 Magnifying and Shrinking the View" on page 4-13.
	Zoom Out	Zooms out on the design, providing a broader view.
	Zoom Window	Magnifies a selected part of the design. For details, see "4.13.2 Magnifying and Shrinking the View" on page 4-13.
	Zoom Bounds	Shows the entire design, including objects that are outside of the board outline.

3.2.3   The Main Toolbar

The **Main** toolbar contains buttons for common board design functions.

The **Main** toolbar buttons are described in the table below:

	Select	De-activates any selected mode (for example, for placing traces) and allows you to select an element on the board.
	Show or Hide Design Toolbox	Shows or or Hides the Design Toolbox. For details, see "3.5 Design Toolbox" on page 3-31.
	Show or Hide Spreadsheet Bar	Shows or hides the Spreadsheet View. For details, see "3.6 Spreadsheet View" on page 3-32.
	Database Manager	Displays the Database Manager. For details, see "6.9 Managing the Database" on page 6-58.

	Board Wizard	Launches the Board Wizard. For details, see "5.2.4 Using the Board Wizard" on page 5-7.
	Start the Component Wizard	Starts the Component Wizard. For details, see "6.8.2 Using the Component Wizard to Create a Part" on page 6-54.
	Place Part from Database	Allows you to browse the database for a part to place. For details, see "6.4 Placing Parts from the Database" on page 6-39.
	Place Line	Places a straight line on the design (or places a trace, if the active layer is a copper one). For details, see "6.3.3 Placing Shapes and Graphics" on page 6-32 or "7.1.2 Placing a Trace: Manual Method" on page 7-3.
	Follow-me	Places a follow-me trace. For details, see "7.1.3 Placing a Trace: "Follow Me" Method" on page 7-4.
	Place a via	Places a via on the design. For details, see "7.4 Working with Vias" on page 7-15.
	Place copper area	Places a copper area on the design.
	Create a powerplane	Places a powerplane on the design. For details, see "7.2.2 Placing Powerplanes" on page 7-11.
	Update the netlist and check for errors	Runs the design rule check and places results in the Results tab.
	Place Text	Places text on the design. For details, see "10.1 Placing and Editing Text" on page 10-2.
	Show 3D	Displays the current design in three dimensions. For details, see Chapter 11, "Viewing Designs in 3D".
	Help Contents	Displays the helpfile contents page.

3.2.4 The Select Toolbar

The **Select** toolbar contains the functions used to control selection filters, and appears by default when you run Ultiboard 9. Please note that all filters are not available in all versions of Ultiboard. .

The **Select** toolbar buttons are explained in the table below:

	Enable Selecting Parts	Allow or prevent selection of components.
	Enable Selecting Traces	Allow or prevent selection of traces.
	Enable Selecting Copper Areas	Allow or prevent selection of copper areas.
	Enable Selecting Vias	Allow or prevent selection of vias.
	Enable Selecting Pads	Allow or prevent selection of pads.
	Enable Selecting SMD Pads	Allow or prevent selection of Surface Mount Device pads.
	Enable Selecting Attributes	Allow or prevent selection of attributes.
	Enable Selecting other objects	Allow or prevent selection of other attributes on PCB.

3.2.5   The Settings Toolbar

The **Settings** toolbar contains buttons for functions that control the appearance of lines and shapes drawn on any layer, except a copper layer.

The **Settings** toolbar buttons are explained in the table below:

	Line Color	Sets the color of the layer's line.
	Line Type	Sets the lines style, for example, solid, dashed.
	Fill Color	Sets the color of the layer's fill.
	Fill Style	Sets the fill style, either transparent or solid.

3.2.6   The Draw Settings Toolbar

The **Draw Settings** toolbar lets you select the layer, thickness and unit of measure of a line or object that is being drawn.

The **Draw Settings** toolbar elements are explained below:

Copper Top	Selects the layer for the line or object being drawn.
mil	Sets the thickness and unit of measure of the line being drawn, or of an object's border. The maximum number of values that can be stored here are set in the Line Width Cache Size field of the General Settings tab of the Preferences dialog box.

3.2.7   The Edit Toolbar

The **Edit** toolbar contains the functions used for editing specific elements, including in-place editing and orientation.

The **Edit** toolbar buttons are explained in the table below:

	Toggle "In-Place" Edit Text or Attribute	Allows you to edit the selected text.
	Toggle "In-Place" PCB Part Edit	Activates In-Place Edit for Placed PCB components. For details, see "6.5.1 Editing a Placed Part (In-Place Edit)" on page 6-40.
	Rotate Clockwise	Rotates selected item clockwise. For details, see "6.1.3.6 Orienting Components" on page 6-11.
	Rotate Counter Clockwise	Rotates selected item counter-clockwise. For details, see "6.1.3.6 Orienting Components" on page 6-11.
	Swap Layer	Places a component on mirror layer. For details, see "6.1.3.6 Orienting Components" on page 6-11.

3.2.8   The Align Toolbar

The **Align** toolbar contains the functions used to align elements with each other. For more on the Aligning commands, see "6.1.3.7 Aligning Components" on page 6-12. For more on the Spacing commands, see "6.1.3.8 Spacing Components" on page 6-12.

The **Align** toolbar buttons are explained in the table below:

	Align Left	Aligns the left sides of the selected components.
	Align Right	Aligns the right sides of the selected components.
	Align Top	Aligns the top edges of the selected components.
	Align Bottom	Aligns the bottom edges of the selected components.
	Align Center Horizontal	Shifts the selected elements horizontally so their centers are aligned.

	Align Center Vertical	Shifts the selected elements vertically so their centers are aligned.
	Space Across	Spaces three or more objects beside each other evenly.
	Space Across Plus	Increases horizontal space between two or more objects.
	Space Across Min	Decreases horizontal space between two or more objects.
	Space Down	Spaces three or more objects above each other evenly.
	Space Down Plus	Increases vertical space between two or more objects.
	Space Down Min	Decreases vertical space between two or more objects.

3.2.9   The Place Toolbar

The **Place** toolbar contains the functions used to place different elements such as traces, lines and polygons on the design.

The **Place** toolbar buttons are explained in the table below:

	Place Comment	Places a comment on the design. For details, see "10.3 Placing a Comment" on page 10-5.
	Capture Screen Area	Captures a section of the screen and places it on the system clipboard.
	Select	De-activates any selected mode (for example, for placing traces) and allows you to select an element on the board.
	Line	Places a line on the design (or place a trace, when used on a copper layer). For details, see "6.3.3 Placing Shapes and Graphics" on page 6-32.
	Arc	Places an arc on the design. For details, see "6.3.3 Placing Shapes and Graphics" on page 6-32.
	Bezier	Places a bezier on the design. For details, see "6.3.3 Placing Shapes and Graphics" on page 6-32.
	Circle	Places a circle on the design. For details, see "6.3.3 Placing Shapes and Graphics" on page 6-32.
	Ellipse	Places an ellipse on the design. For details, see "6.3.3 Placing Shapes and Graphics" on page 6-32.

	Pie	Places a pie-shape on the design. For details, see "6.3.3 Placing Shapes and Graphics" on page 6-32.
	Rounded Rectangle	Places a rectangle with rounded corners. For details, see "6.3.3 Placing Shapes and Graphics" on page 6-32.
	Rectangle	Places a rectangle on the design. For details, see "6.3.3 Placing Shapes and Graphics" on page 6-32.
	Polygon	Places a polygon on the design. For details, see "6.3.3 Placing Shapes and Graphics" on page 6-32.
	Place Copper Area	Places a polygon copper area on the design. For details, see "7.2.1 Placing Copper Areas" on page 7-10.
	Follow-me	Places a follow me trace, a trace that automatically draws a legal trace between two selected points. For more details, see "7.1.3 Placing a Trace: "Follow Me" Method" on page 7-4.
	Place Multiple Traces as a Bus	Use to connect multiple traces between multi-pinned devices such as ICs. For details, see "7.1.5 Placing a Bus" on page 7-5.
	Place Group Array Box	A group array box is used to place components in an array. For details, see "6.1.3.9 Placing a Group Array Box" on page 6-13.
	Place text	Places text on the design. Useful for annotation purposes. For more details, see "10.1 Placing and Editing Text" on page 10-2.
	Place a Standard Dimension (All Angles)	Places a dimension between any two selected points. For details, see "6.3.6 Working with Dimensions" on page 6-36.
	Place a Horizontal Dimension	Places a horizontally-oriented dimension between two selected points. For details, see "6.3.6 Working with Dimensions" on page 6-36.
	Place a Vertical Dimension	Places a vertically-oriented dimension between two selected points. For details, see "6.3.6 Working with Dimensions" on page 6-36.
	Place a Net Bridge	Places a net bridge connection. For details, see "7.6.4 Net Bridges" on page 7-36.
	Place a Hole	Places a hole on your PCB.
	Place a via	Places a via on the design. For details, see "7.4 Working with Vias" on page 7-15.
	Add one or more pins to your drawing	Places pins on the design when editing or creating a part in In-place Edit mode or Footprint Edit mode. For details, see "6.5.1 Editing a Placed Part (In-Place Edit)" on page 6-40 or "6.8.1 Using the Database Manager to Create a Part" on page 6-52.
	Polygon splitter	Splits copper areas and powerplanes. For details, see "7.2.3 Splitting Copper" on page 7-11.
	Remove copper islands	Removes copper islands. For details, see "7.2.1 Placing Copper Areas" on page 7-10.

3.2.10 The Wizard Toolbar

The **Wizard** toolbar contains the wizard functions supported by Ultiboard 9. The **Wizard**

toolbar buttons are explained in the table below:

	Board Wizard	Starts the Board Wizard. For details, see "5.2 Working with the Board Outline" on page 5-5.
	Component Wizard	Starts the Component Wizard. For details, see "6.8.2 Using the Component Wizard to Create a Part" on page 6-54.

3.2.11 The Autoroute Toolbar

This toolbar is only available when you have installed Ultiroute, Electronics Workbench's autoplacement and autorouting application.

The **Autoroute** toolbar contains the autorouting and placement functions supported by Ultiboard 9.

The **Autoroute** toolbar buttons are explained in the table below:

	Begin Autoplacing with Ultiroute	Starts automatically placing components using Ultiroute. For details, refer to the *Ultiroute 9 User Guide*.
	Start/Resume Autorouting with Ultiroute	Starts automatically placing traces using Ultiroute. For details, refer to the *Ultiroute 9 User Guide*.
	Ultiroute Selected Buses	Displays the Ultiroute Bus Routing dialog. For details, refer to the *Ultiroute 9 User Guide*.
	Start trace optimization with Ultiroute	Starts trace optimization using Ultiroute. For details, refer to the *Ultiroute 9 User Guide*.
	Stop/Pause Ultiroute	Stops an Ultiroute process that is running. For details, refer to the *Ultiroute 9 User Guide*.

3.3　Setting Preferences

This section explains general procedures for setting preferences. The following sections describe details of setting specific options.

➢ To set your user preferences:

1. Choose **Options/Global Preferences**. The **Preferences** dialog box appears, offering you the following tabs:

    • General Settings Tab
    • Paths Tab
    • Colors Tab
    • PCB Design Tab
    • Dimensions Tab
    • 3D Options Tab

2. Select the desired tab.

3. Set the desired customization options. The specific options and settings available in the tabs are described in the following sections of this chapter.

4. Click **OK** to save your changes.

3.3.1   General Settings Tab

➢ To view or change workspace options:

1. Choose **Options/Global Preferences**.

2. Select the **General Settings** tab:

Enable or disable the items you want shown.

Enable to automatically load last project when Ultiboard 9 starts up

Enable to have changes automatically saved (at the specified interval).

Enter size of control points on vertices and other objects.

The number of recently used line widths Ultiboard keeps in memory. Minimum cache size is 5
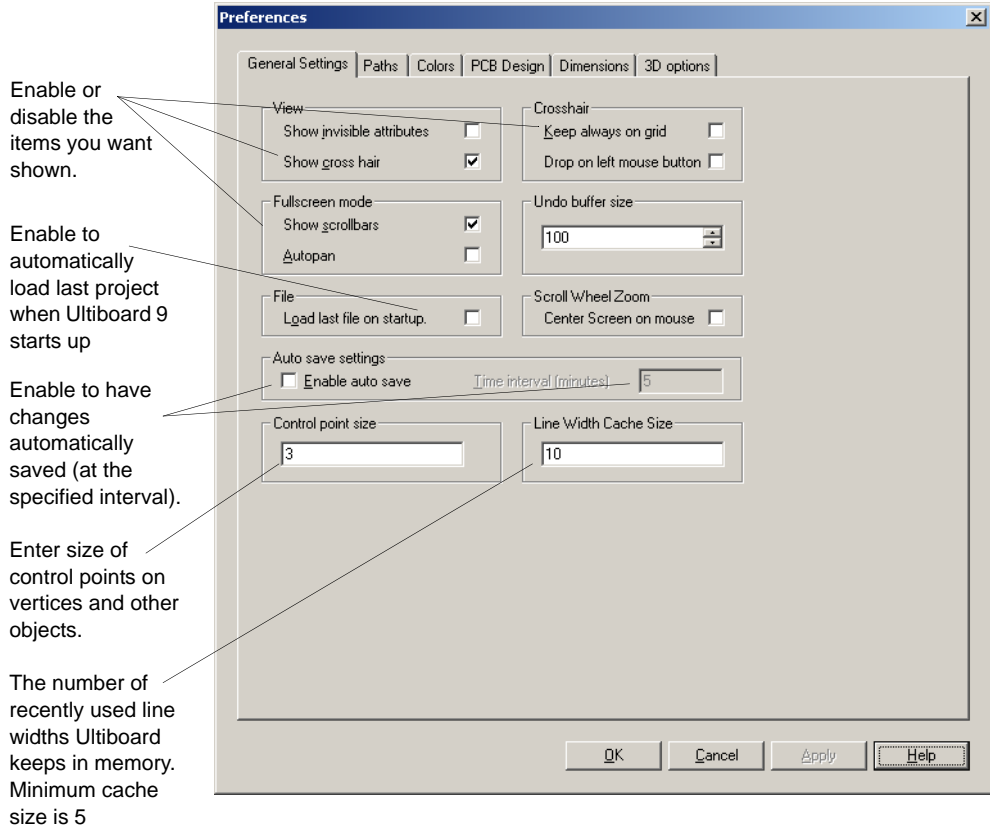
3. Set the options as detailed in the above diagram.

4. To apply your changes but leave the **Preferences** dialog box open, click **Apply**. To apply your changes and close the **Preferences** dialog box, click **OK**. To cancel your changes, click **Cancel**.

3.3.2   Paths Tab

The Ultiboard 9 installation puts specific files in specific locations. If necessary you can point Ultiboard 9 to a new location to find, for example, component libraries. You can also use this dialog box to create and specify user settings files that contain individuals' preferences for all options.

➢ To set up file locations:

1. Choose **Options/Global Preferences**.

2. Select the **Paths** tab and navigate to the appropriate locations for the different elements:
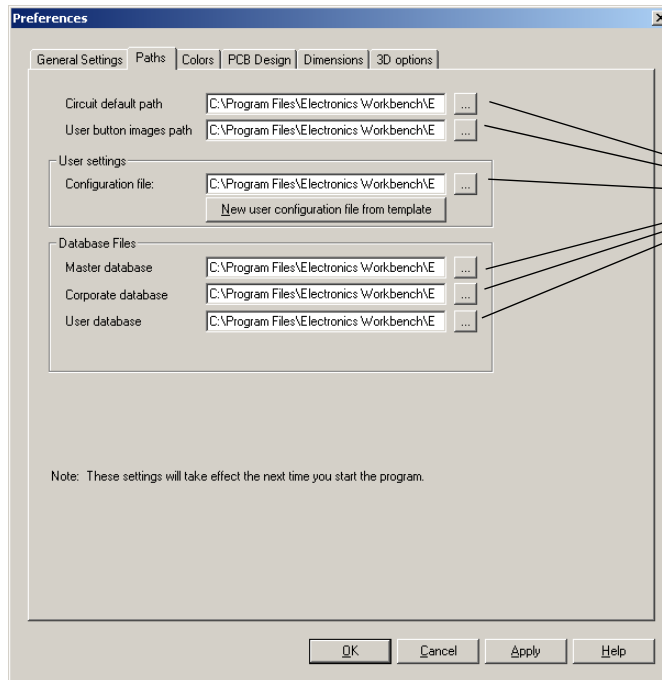
Click to navigate to a new location for the files

3. To use a different configuration file, navigate to the appropriate user settings file. To create a new user configuration file, click **New user configuration file from template**. You are prompted to select the configuration file to use as a template, then to enter a name for the new configuration file. All options changed in the **Preferences** dialog box are saved in the new configuration file.

4. To apply your changes but leave the **Preferences** dialog box open, click **Apply**. To apply your changes and close the **Preferences** dialog box, click **OK**. To cancel your changes, click **Cancel**.

3.3.3   Colors Tab

Ultiboard allows you to set up color schemes for the workspace background and other displayed elements.

➢ To set up color schemes:

1. Choose **Options/Global Preferences**. The **Preferences** dialog box appears.

2. Select the **Colors** tab:
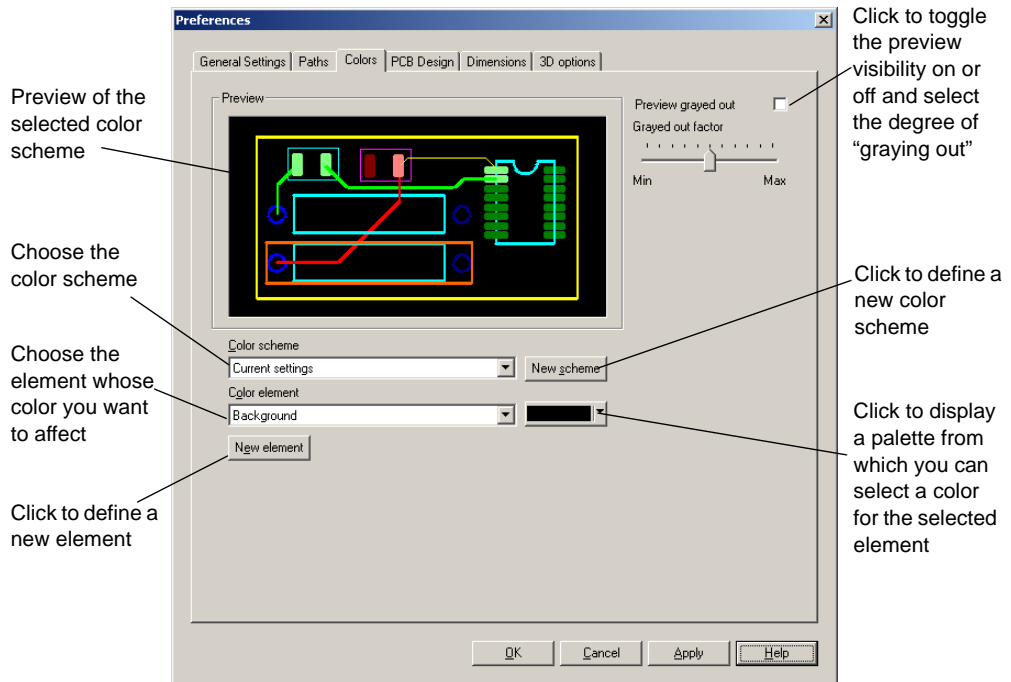
- To apply an existing color scheme, either the default or one you have created, choose from the **Color scheme** drop-down list.
- To modify an existing color scheme, choose it from the **Color scheme** drop-down list. From the **Color element** drop-down list, choose the element whose color you want to modify. Click the color button (unlabelled) and choose a new color for that element. Your change is stored with the color scheme when you click **Apply** or **OK**.
- To create a color scheme, click **New scheme** and provide a scheme name. The element colors remain at the value they had in the most recently loaded scheme. Modify the element colors as described above, and save your changes by clicking **Apply** or **OK**.
- To add a color element, click **New element** and enter a name. The element appears in the **Color element** list, and you can choose a color for the element.

3. To apply your changes but leave the **Preferences** dialog box open, click **Apply**. To apply your changes and close the **Preferences** dialog box, click **OK**. To cancel your changes, click **Cancel**.

3.3.4   PCB Design Tab

Use the **PCB Design** tab of the **Preferences** dialog box to define a variety of actions associated with the overall PCB design.

➢ To define the default actions:

1. Choose **Options/Global Preferences**. The **Preferences** dialog box appears.

2. Select the **PCB Design** tab.
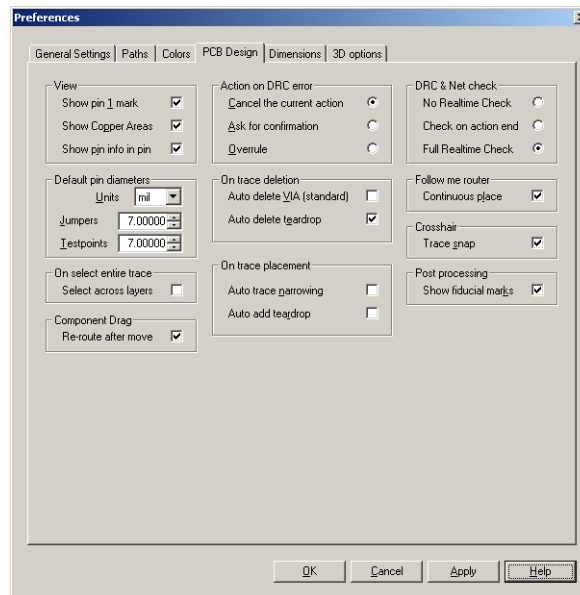
3. Set the viewing options in the **View** area:

- **Show pin 1 mark** — enable to display pin 1 of a device with a special marking.
- **Show Copper Areas** — enable to display copper areas. This applies to copper areas only, not regular polygon shapes on non-copper layers.
- **Show pin info in pin** — enable to display the pin number and hole size when zoomed in.

4. In the **Default pin diameters** area:

- **Units** — select unit of measurement from drop-down list.
- **Jumpers** — enter the default pin diameter for jumpers.
- **Testpoints** — enter the default pin diameter for testpoints.

5. In the **On select entire trace** area:

- **Select Across Layers** — enable so that when you choose **Edit/Select Entire Trace**, the trace will be selected across all layers, instead of just the active layer.

6. In the **Component Drag** area:

- **Re-route after move** — enable if you would like the traces that are attached to a component to be re-routed if you move the component.

7. In the **Action on DRC error** area, set the actions to be taken when Ultiboard encounters a Design Rule Error:

- **Cancel the current action** — enable to prevent the current operation from being completed. For example, placing a trace over a pad that is part of a different netlist will not be permitted.
- **Ask for Confirmation** — enable so that if an invalid operation is attempted a prompt is given asking if the operation should be completed even though a Design Rule will be violated.
- **Overrule** — enable to permit the operation to be completed.

8. In the **On trace deletion** area:

- **Auto delete Via (standard)** — enable to have vias automatically deleted when you delete the trace associated with that via.
- **Auto delete teardrop** — enable to have teardrops automatically deleted when you delete the trace associated with that teardrop.

9. In the **On trace placement** area:

- **Auto add teardrop** — enable to add a teardrop when a trace is placed.
- **Auto trace narrowing** — enable to allow traces to narrow as necessary during routing.

10. In the **DRC & Net check** area:

- **No Realtime Check** — enable to prevent a DRC and netcheck in "real time" as you place objects, for example, parts.
- **Check on action end** — enable to perform a DRC and netcheck after each action, for example, part placement.
- **Full Realtime Check** — enable to perform DRC checks and ratnest updates to your work in "real time" (as you place objects, for example, parts).

11. In the **Follow me router** area:

- **Continuous place** — enable to have the next trace begin from the finish point of the previous trace when placing a "Follow-me" trace. If this is not selected, you must click to start a new trace.

For more details about Follow-me traces, see "7.1.3 Placing a Trace: "Follow Me" Method" on page 7-4.

12. In the **Crosshair** area:

• **Trace snap** — enable to have the pointer snap to the nearest trace.

For details about using the trace snap, see "7.1.1 Working with Traces" on page 7-3.

13. In the **Post processing** area:

• **Show fiducial marks** — enable to show fiducial marks when you postprocess this design. Fiducial marks are the marks used to align layers with each other. Ultiboard has three different types of cocentric circles, which are placed on three of the four corners when the design is printed or exported to Gerber.

Fiducial Marks
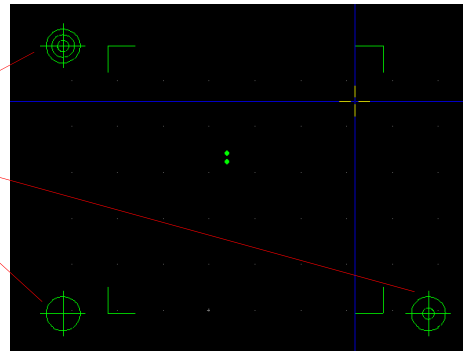
14. To apply your changes but leave the **Preferences** dialog box open, click **Apply**. To apply your changes and close the **Preferences** dialog box, click **OK**. To cancel your changes, click **Cancel**.

3.3.5 Dimensions Tab

Use the **Dimensions** tab of the **Preferences** dialog box to define the characteristics to be used for any dimensions placed in the board (for information on placing dimensions, see "6.3.6 Working with Dimensions" on page 6-36).

➤ To define the default dimension characteristics:

1. Choose **Options/Global Preferences**. The **Preferences** dialog box appears.

2. Click the **Dimensions** tab.
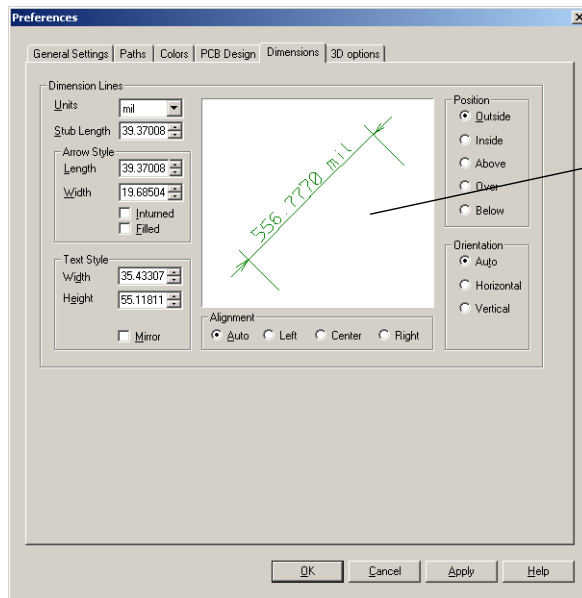
A preview of the results of your choices appears here.

3. Choose the units of measurement to be used and displayed on the dimension, the length of the dimensions' "stub" (the length of the lines defining the dimension) and style of its arrow and text, its alignment, position and orientation. The results of your choices are previewed on the tab.

4. To apply your changes without closing the dialog box, click **Apply**. Existing dimensions are not affected. To apply your changes and close the dialog box, click **OK**. To cancel your changes, click **Cancel**.

3.3.6   3D Options Tab

➤ To set 3D viewer options:

1. Choose **Options/Global Preferences** and select the **3D Options** tab:
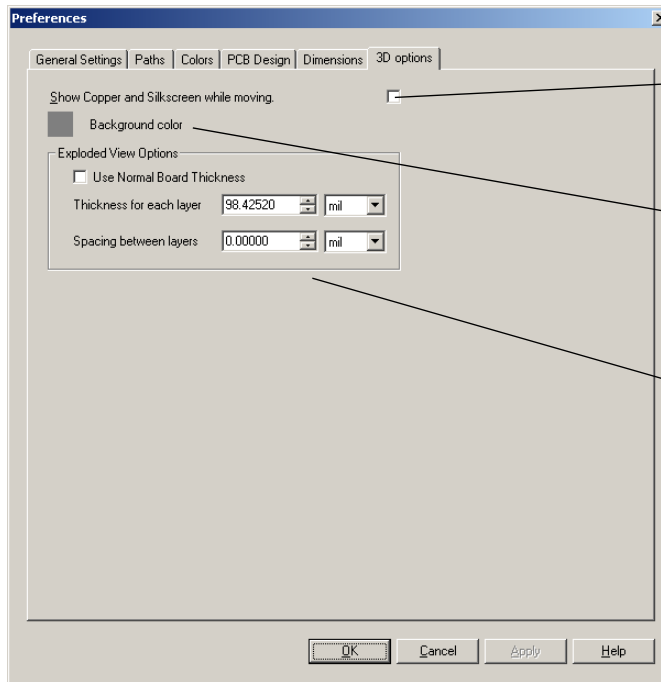
Select to show copper and silkscreen while moving the 3D view

Click to change background color in 3D viewer

Set Exploded View options here. The more space that you set between the layers, the easier it will be to view when you zoom in on the PCB in the 3D Viewer mode. For details, see "11.2.3 Exploded View" on page 11-5

2. To change the background color in the 3D viewer, click on **Background color**. The **Color** dialog box appears.

3. Select the desired color and click **OK** in the **Color** dialog box.

4. To apply your changes but leave the **Preferences** dialog box open, click **Apply**. To apply your changes and close the **Preferences** dialog box, click **OK**. To cancel your changes, click **Cancel**.

3.4   Setting PCB Properties

Many characteristics of your PCB design are controlled through the board's properties including number of layers, design rules and grid settings. These settings are saved with the design and will be in effect when the design is reopened.

> To display the **PCB Properties** dialog box for the board, do one of the following:

- With nothing selected on the board, right-click on an empty portion of the board and choose **Properties** from the context menu that appears.
- Click on an empty portion of the board and choose **Edit/Properties**.
- Double-click on an empty portion of the board.

3.4.1 Attributes Tab

All properties dialog boxes for all design elements have an **Attributes** tab, although a design's PCB typically does not have any attributes. To add an attribute, see "6.2.1 Attributes" on page 6-18.

3.4.2 Grid and Units Tab

Use the **Grid & Units** tab of the **PCB Properties** dialog box to control Ultiboard grids and set the unit of measurement for your design.

Preview area shows results of choices for visible grid.
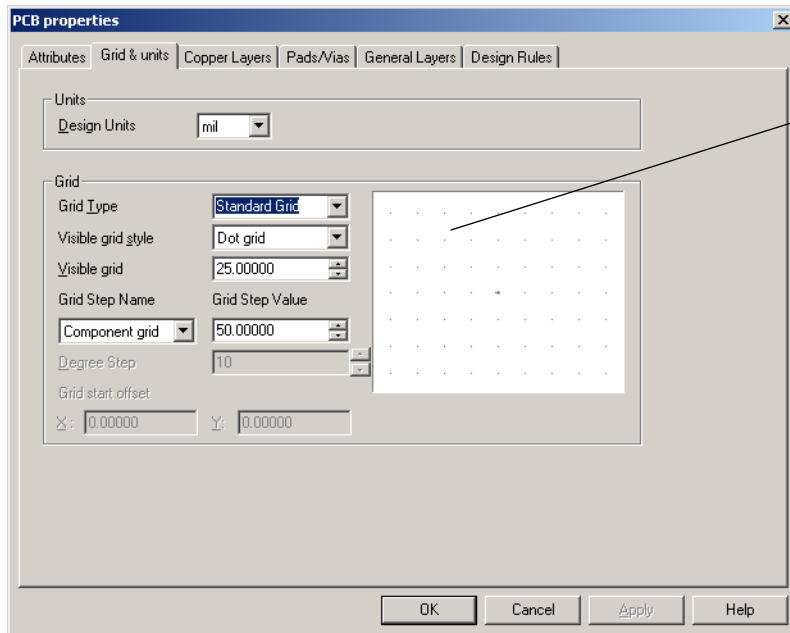
➤ To set up the grid properties:

1. To set the units of measurement to be used by default in Ultiboard, choose from the **Design Units** list.

2. In the **Grid** area, set the following elements as desired:

   • **Grid Type** — select Standard Grid for a rectangular grid, or Polar Grid, for a circular grid.
   • **Visible grid style** — this is where you set the style for the **Visible Grid**. (The **Visible Grid** provides a visual reference for you to visually align components and traces).
   • **Visible grid** — this is the distance between the grid's elements (dots, lines or crosses) that you set up in the **Visible grid style** field.
   • **Grid Step Name** — use this field to change the distance between a grid's elements. Select the desired grid: **Component Grid**; **Copper Grid**; **Via Grid**; **SMD Grid**, and then enter the desired increment in the **Grid Step Value** field.
   • **Degree Step** — this field becomes active when Polar Grid is selected in the **Grid Type** field. Enter the desired distance between the grid's elements (dots, lines or crosses).
   • **Grid start offset** — this field becomes active when Polar Grid is selected in the **Grid Type** field. Enter the distance to offset the center of the polar grid: **X** is horizonatal offset; **Y** is vertical offset.

3. Click **OK** to close the dialog.

➤ To show or hide the visible grid, use **View/Grid**.

3.4.3 Copper Layers Tab

Use the **Copper Layers** tab of the **PCB Properties** dialog box to determine the layer pairs that are acceptable for blind or buried vias. This is used when you try to place a via. Your choices will affect the cost of the board, and should be confirmed with the board manufacturer.

This tab also allows you to set the default clearance for the board — the distance from the edge of the board that is to be kept free of any other elements. Trying to run a trace through a clearance, or trying to place a part so that a pad is put within a clearance, for example, results in a design rule error.

If you used the **Board Wizard**, the layer information and clearance may already have been set. For more details, see "5.2.4 Using the Board Wizard" on page 5-7.
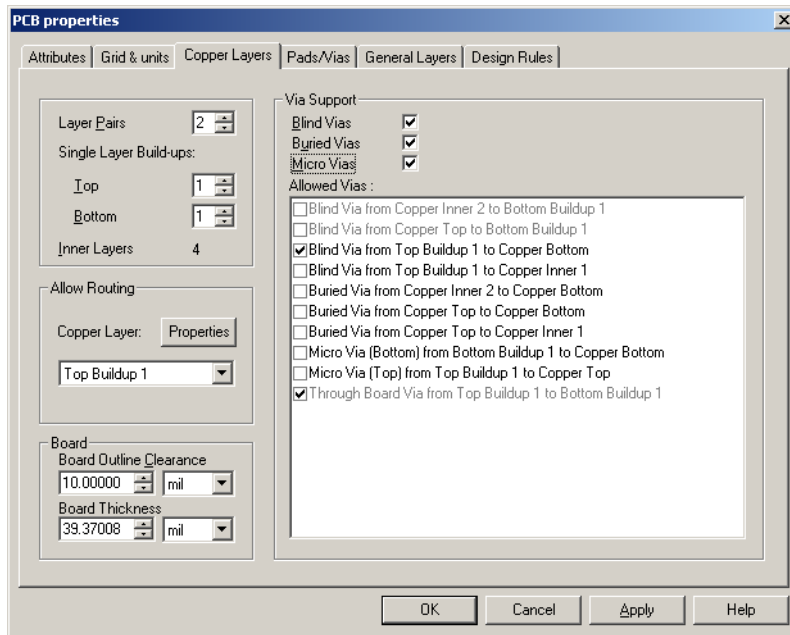
1. Set the number of layer pairs you intend to use by entering the value or using the up/down arrows in the **Layer Pairs** field. There should be at least one layered pair to act as a core.

2. Set the number of **Single Layer Build-ups** for both the top and bottom. There should be at least one layered pair to act as a core.

3. Select the **Micro Vias**, **Buried Vias**, or **Blind Vias** checkboxes to use these in your design.

4. As you make changes to the layer settings, the **Allowed Vias** window shows the acceptable layer combinations for blind and buried vias or microvias. Use the checkboxes to select the layer combinations you want to allow in your design.

5. In the **Allow Routing** area, from the **Copper Layer** drop-down list, select the copper layer for which you wish to assign routing properties and click **Properties** to display the **Copper Layer Properties** dialog box.
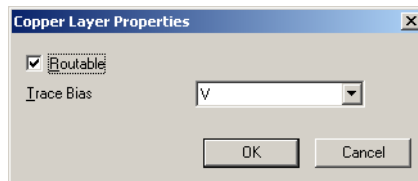
Enable the **Routable** checkbox to allow routing on the selected layer. In the **Trace Bias** drop-down list, select one of "H" for horizontal; "V" for vertical; or "None".

Click **OK** to close the **Copper Layer Properties** dialog box.

6. In the **Board** area, enter the desired **Board Outline Clearance** and **Board Thickness**.

7. Click **OK** to close the dialog box.

3.4.4   Pads/Vias Tab
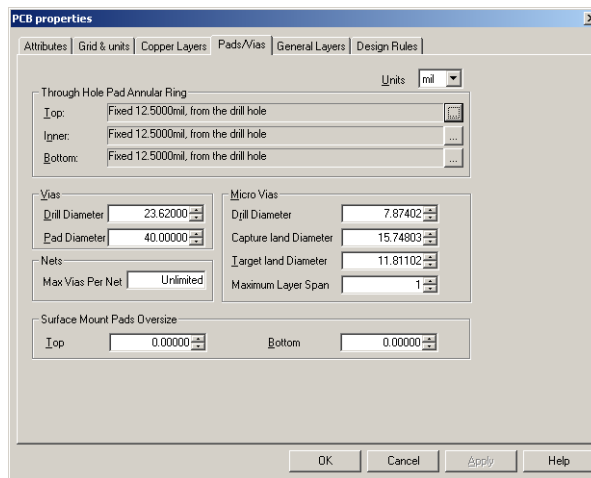
Use the **Pads/Vias** tab of the **PCB Properties** dialog box to set the following constraints:

- Through Hole Pad Annular Ring
- Vias
- Microvias
- Maximum Vias Per Net
- Surface Mount Pads Oversize

Through Hole Pad Annular Ring

Footprints in the database have been designed with pad sizes in accordance with the manufacturers' recommendations. To change these settings you can edit the footprint in the database or directly on the design using the **In-place Part Edit** command. Alternatively, you can apply a set of design rules to specific footprints or to the entire design.

The annular ring setting (the ring of copper around the drill hole of a pad or via) allows you to specify either an absolute value (e.g. 50 mils) or a relative value that depends on the radius of

the drill hole size. For example, a 50 mil drill hole (radius of 25 mils) with a relative value of .6 (60%) will create an annular ring of 15 mils (60% of 25 mils). When using relative settings, it is important to choose Min and Max values to ensure that the annular ring will never go beyond those bounds.

➢ To enter an annular ring setting:

1. Click the button beside the desired field (**Top**, **Inner** or **Bottom**) in the **Pad/Vias** tab. The **Through Hole Pad Properties** dialog box appears.
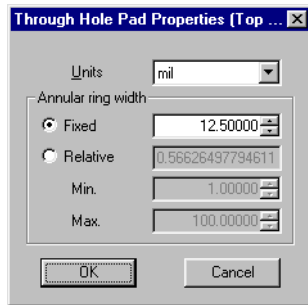
Choose **Fixed** and enter the fixed value.

Or

Choose **Relative** and enter the relative setting. Be sure to choose **Min** and **Max** values. For more on absolute and relative values, see earlier in this section.

➢ To apply the design rules to a footprint:

1. Select the footprint and choose **Edit/In-Place Part Edit**.

2. Select the pads to which the rule should apply.

3. Right-click and, from the context menu, choose **Properties**.

4. On the **Pad** tab of the properties dialog box, enable the **Use Design Rules** option.

Vias

The **Vias** setting sets the default via dimensions to be used when a via is placed during trace placement. This setting will also apply to vias that have already been placed in the design.

Microvias

The **Micro Vias** area is where you set the default via dimensions to be used when a microvia is placed during trace placement. This setting will also apply to microvias that have already been placed in the design.

The **Capture Land Diameter** field determines the land diameter where the microvia starts, while **Target Land Diameter** determines the diameter where the microvia ends. These terms

are in accordance with the IPC and JPCA joint standard IPC/JPCA-2315, Design Guide for High Density Interconnects (HDI) and Microvias. The **Maximum Layer Span** is either one or two layers.

Maximum Vias Per Net

The **Nets** area is where you set up the maximum number of vias allowed per net.

Surface Mount Pads Oversize

Depending on the manufacturing process you are using, you may wish to enlarge the size of all surface mount pads by a fixed amount. This can be set independently for the **Top** and **Bottom** layers. The setting refers to the amount that will be added to the pad. In the case of a circular pad this amount is added to the radius. In the case of a rectangular or square pad the amount will be added to the width and length.

3.4.5 General Layers Tab

Use the **General Layers** tab of the **PCB Properties** dialog box to control which layers are available on the board.
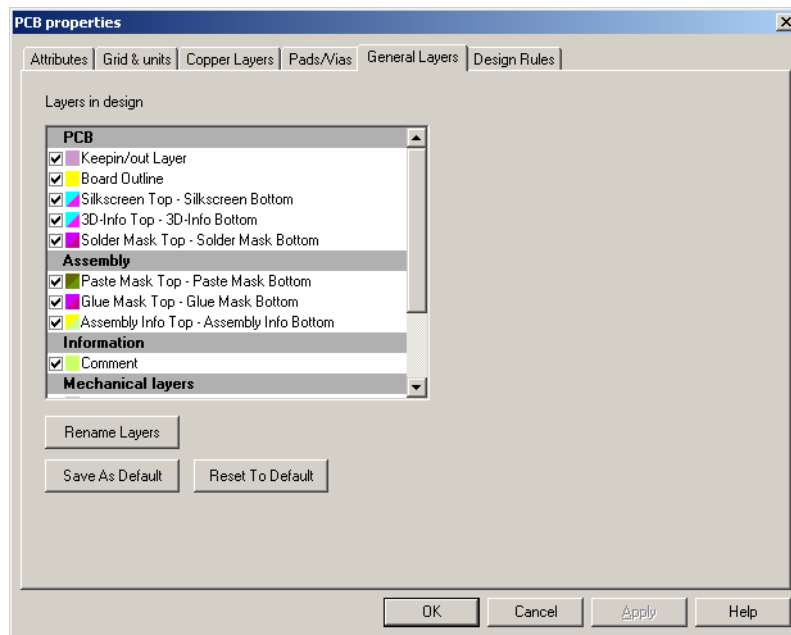
◈ If you are in a Mechanical CAD file, the **General Layers** tab appears as follows.
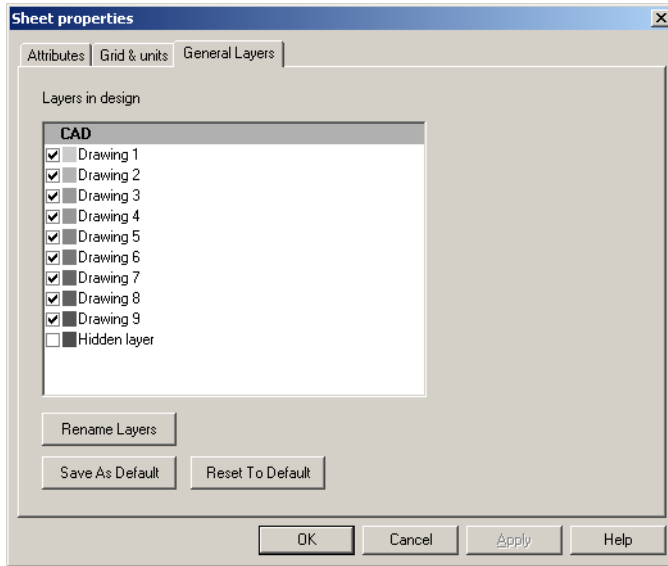
➢ To control the numbers of layers in the board, do the following:

- Add layers to the board by selecting the checkbox beside the layer name.
- Remove layers from the board by clearing the checkbox beside the layer name.

Note You can have a layer available on the board but temporarily dim or hide it. For details, see "5.1.2 Accessing Layers" on page 5-3.

Note There are up to ten mechanical CAD layers in PCB design. These layers allow you to provide information that is directly related to the PCB. You place elements on them to represent the mechanical elements of your design – for example, cabinet casing or mounting brackets.

> To rename layers:

   1. Click **Rename Layers**. The **Select Layer for Renaming** dialog box appears.
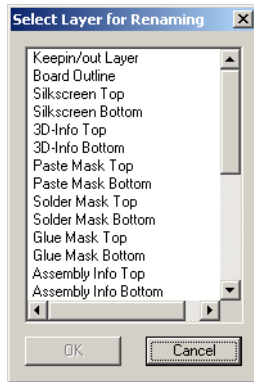
   2. Select the layer that you wish to rename and click **OK**. The **Layer** dialog box appears.
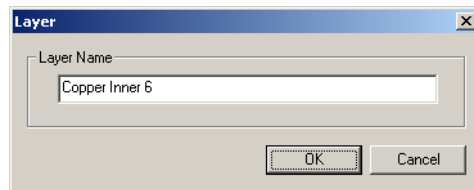
   3. Enter the new layer name and click **OK**.

3.4.6   Design Rules Tab

> To set the design rules for the PCB:

   1. Click on the **Design Rules** tab.
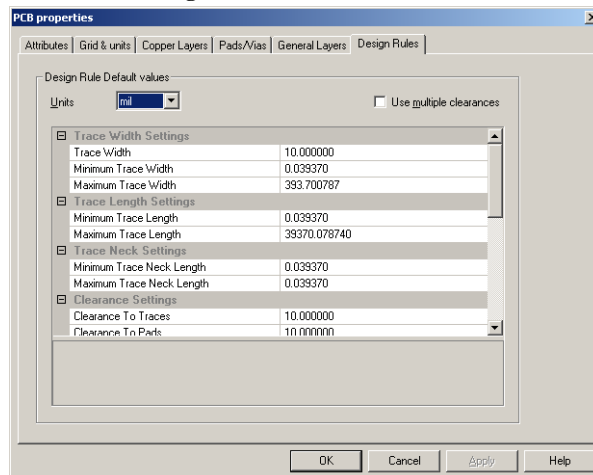
2. Set the following parameters as desired:

- **Trace Width Settings**—set trace width and minimum and maximum trace width.
- **Trace Length Settings**—set minimum and maximum trace length.
- **Trace Neck Settings**—set minimum and maximum trace neck length.
- **Clearance Settings**—set clearance to traces, pads, vias and copper areas. For all of these clearances to be active, and display in the **Spreadsheet View**, the **Use multiple clearances** checkbox must be selected. Otherwise, only clearance to traces will be operational.
- **Annular Ring Settings**—set minimum annular ring size.
- **Component Spacing Settings**—set minimum distance between components.
- **Pin & Gate Swapping Settings**—set parameters for pin and gate swapping.
- **Thermal Relief**—set the thermal relief shape.

3.4.7   Setting Favorite Layers

You can assign shortcut numbers for up to ten layers using the **Favorite Layers** tab.

➢ To assign shortcut numbers to layers:

1. Click on the Favorite Layers tab.

2. Select the desired layer from the drop-down list beside each layer number. For example, you may wish to assign **Layer 2** to the **Copper Bottom** layer.

3. Click **OK**.

➢ To make a layer active, press CTRL+ALT+<LAYER NUMBER> on your keyboard. The layer becomes active and is highlighted in the **Layers** tab of the **Design Toolbox**. In the above example, pressing CTRL+ALT+2 on your keyboard will make the **Copper Bottom** layer active.

3.5   Design Toolbox

The **Design Toolbox** is a vital part of the user interface. You will use it often to manage your design efforts by controlling major parts of Ultiboard's functionality. To toggle it on or off, use **View/Design Toolbox**.
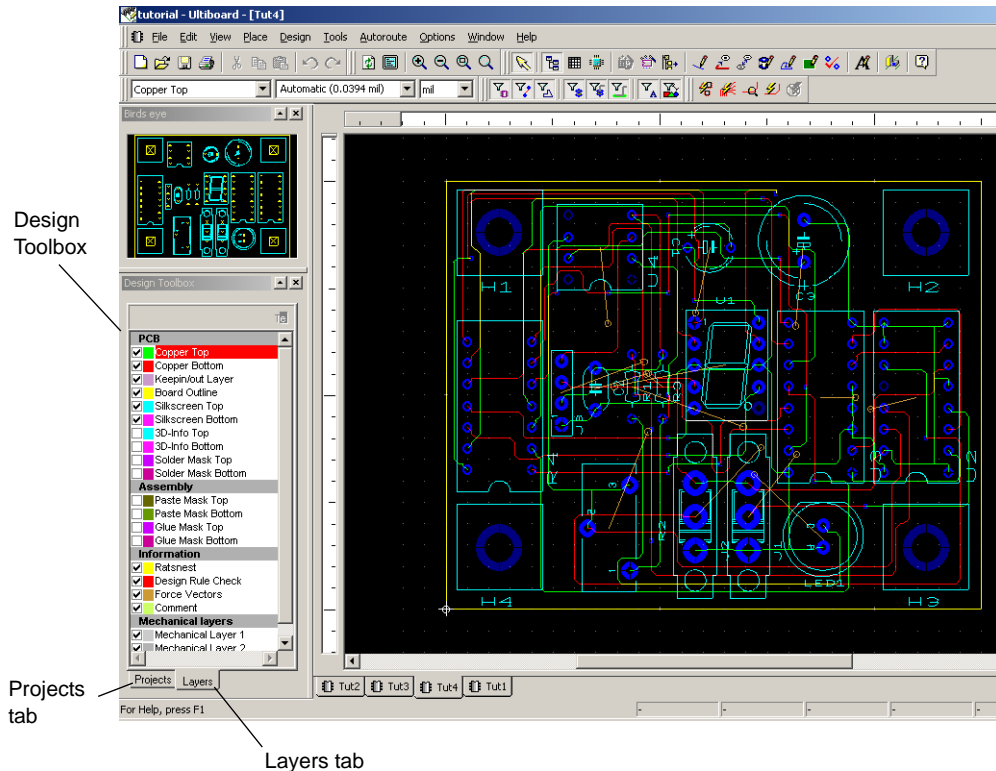
The **Design Toolbox** is made up of two tabs, which are used as follows:

- The **Projects** tab lets you view the projects that are currently open. Each project may contain one or more designs. Double-click to make a particular design the current view. You can also click on the desired tab below the workspace, for example, "Tut2" in the above diagram.
- The **Layers** tab lets you move between layers of your design, control the appearance of layers, and perform several other functions.

3.6   Spreadsheet View

◈ The **Spreadsheet View** allows fast advanced viewing and editing of parameters including component details such as footprints, Reference Designators, attributes and design constraints.

▦ By default, the **Spreadsheet View** does not appear until you have opened a project. To toggle the **Spreadsheet View** on and off, select **View/Spreadsheet View**.

The following buttons are available in the **Spreadsheet View**.

Note The buttons do not appear in all tabs.

Button	Description
▦	**Export to Textfile** button. Displays a standard Windows Save dialog where you save the selection as a textfile.
▦	**Export to CSV File** button. Displays a standard Windows Save dialog where you save the selection as a file with comma-separated values.
▦	**Export to Excel** button. Click to open a Microsoft® Excel spreadsheet with the selected data displayed. (You must have Excel installed to use this function).
A↓Z	**Sort Ascending** button. Sorts the selected column in ascending order.
Z↓A	**Sort Descending** button. Sorts the selected column in descending order.
🖨	**Print** button. Prints the data in the selected tab.
📋	**Copy** button. Copies the selection to the clipboard.
🔍	**Find and Select the Part** button. Zooms in on the selected part. For details, see "6.1.2.2 Using the Parts Tab for Other Functions" on page 6-4.
🔒	**Lock the Selected Part** button. Locks the selected unlocked parts or unlocks the selected locked parts. For details, see "6.1.2.2 Using the Parts Tab for Other Functions" on page 6-4.
▲	**Start Placing the Unpositioned Parts** button. For details, see "6.1.2.1 Using the Parts Tab to Place Parts" on page 6-4.
🔍	**Preview** button. Toggles the Spreadsheet View's Preview function on and off. For details, see "6.1.2.2 Using the Parts Tab for Other Functions" on page 6-4.

Note You can also access the above commands from a pop-up menu by right-clicking in the **Spreadsheet View**.

3.6.1 Spreadsheet View: DRC Tab

The **DRC** tab displays errors (e.g., Design Rule Errors) and warnings as they occur while you work. Double-clicking on an error will take you to the error on the workspace.
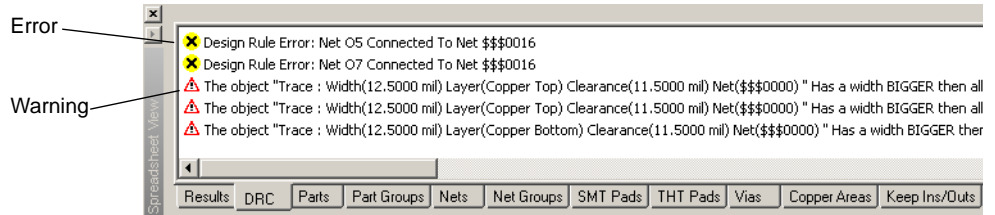
When you right-click on an item in the **DRC** tab, a pop-up menu displays with the following available selections:

- **Copy** — copies all items in the **DRC** tab and places them on the clipboard.
- **Go to Error/Tag** — click to go to the highlighted error on the workspace.
- **Add to Filter** — click to filter out the selected error/warning type. This type of error/warning will no longer show in the **DRC** tab.
- **Remove all filters** — click to remove all error/warning types from the filter. All errors and warnings will now show in the **DRC** tab.
- **Start Filtermanager** — click to start the **Filter Manager**.
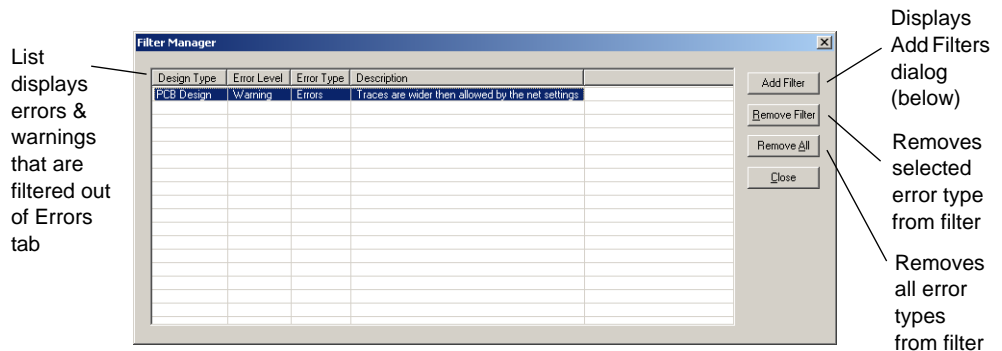
Select error types
to add to the filter.
You can use the
CTRL and SHIFT
keys to select
multiple iterms.
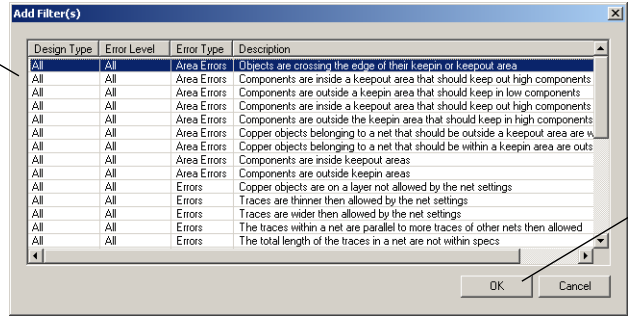
Click to
add
selected
items to
the filter
manager

3.6.2 Spreadsheet View: Results Tab

The **Results** tab displays the results of searching for elements in the design. For details, see
"4.12 Searching for Design Elements" on page 4-11.

It also displays the results of running a connectivity check.

Note   The **Results** tab flashes red when it contains new data.

Double-click a line in this tab to zoom in on the corresponding area of the design. For details,
see "A.5.2 Design/Connectivity Check" on page A-17.

Right-click an item to display pop-up

Double-click an
item to zoom to the
corresponding area
on workspace
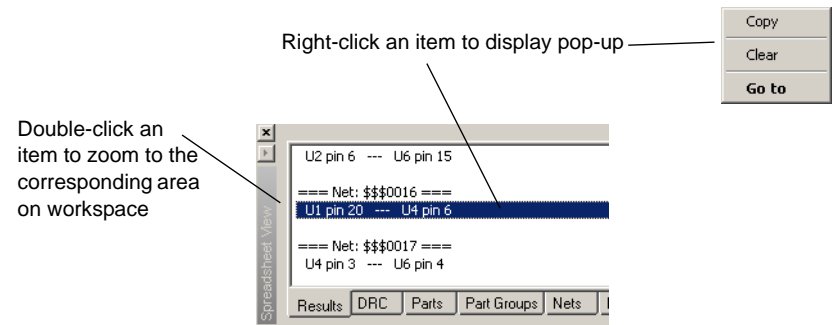
3.6.3 Spreadsheet View: Parts Tab

The **Parts** tab lets you work with the parts in your design, as explained in "6.1.2 Using the Parts Tab in the Spreadsheet View" on page 6-3.

Column	Description
(unlabelled)	The colored circle indicates whether the part has been placed on the board outline (bright green), or is off to the side awaiting placement (dark green). Orange indicates the part has been locked.
Refdes	The component's Reference Designator (unique identifier).
Value	The component's value, for example, 150 pF for a capacitor.
Shape	The physical footprint of the component.
Locked	"Yes" indicates that the component cannot be moved. "No" indicates that the component can be moved.
Trace Clearance	This is the minimum spacing allowed between the component and any trace. You can enter a value here, or use the Group Editor. For details, see "5.5 Working with the Group Editor" on page 5-14.
Component Spacing	This is the minimum spacing allowed between components. You can enter a value here, or use the Group Editor. For details, see "5.5 Working with the Group Editor" on page 5-14.
Pin Swap	If enabled, allows like-pins to be swapped during the routing process. You can choose either "No" (not enabled), "Yes" or "Use Group Settings". For details, see "5.5 Working with the Group Editor" on page 5-14.
Gate Swap	If enabled, allows like-gates, to be swapped during the routing process. You can choose "Internal Gates Only", which will swap gates within the same IC; "No Swapping"; "Advanced Swapping", which will swap gates between ICs (both devices must be set to Advanced Swapping); or "Use Group Settings" (for details, see "5.5 Working with the Group Editor" on page 5-14).
Part Group	The group in which the part is placed. You can select "No Group" or an existing group from the drop-down list. Parts groups are created in the Group Editor. For details, see "5.5 Working with the Group Editor" on page 5-14.

3.6.4   Spreadsheet View: Part Groups Tab

The **Part Groups** tab lets you work with part groups as described in the table below.

Column	Description
Part Group	This is the group in which the part is contained. For details, see "5.5 Working with the Group Editor" on page 5-14.
Trace Clearance	This is the minimum spacing allowed between the components in the group and any trace. You can enter a value here, or use the Group Editor. For details, see "5.5 Working with the Group Editor" on page 5-14.
Component Spacing	This is the minimum spacing allowed between components in the group. You can enter a value here, or use the Group Editor. For details, see "5.5 Working with the Group Editor" on page 5-14.
Pin Swap	If enabled, allows pins for like-components to be swapped during the routing process.
Gate Swap	If enabled, allows like-gates, to be swapped during the routing process. You can choose "Internal Gates Only", which will swap gates within the same IC; "No Swapping"; or "Advanced Swapping", which will swap gates between ICs (both devices must be set to Advanced Swapping).
Locked	"Yes" indicates that the component cannot be moved. "No" indicates that the component can be moved.

3.6.5   Spreadsheet View: Nets Tab

The **Nets** tab lets you work with the net lists in your design, as explained in "7.6.1 Using the Nets Tab" on page 7-22.

Column	Description
(unlabelled)	The colored circle indicates whether the copper for the net has been routed, i.e., placed on the board outline (bright green), or is awaiting placement (dark green). Orange indicates the copper for the net has been locked. You cannot lock the net until it has been routed.
Net Name	The net's unique identifier.
Locked	"Yes" indicates that the net cannot be moved. "No" indicates that the net can be moved. You cannot lock a net until it has been routed.

Column	Description
Trace Width	The "default" width of the trace that is placed during routing. You can enter a value here, or use the Group Editor. For details, see "5.5 Working with the Group Editor" on page 5-14.
Max Width	The maximum width to which a trace will be laid during routing. You can enter a value here, or use the Group Editor. For details, see "5.5 Working with the Group Editor" on page 5-14.
Min Width	The minimum width to which a trace will be laid during routing. You can enter a value here, or use the Group Editor. For details, see "5.5 Working with the Group Editor" on page 5-14.
Topology	The topology of the net as set in the Netlist Editor. Choices are "Shortest", "Daisy Chain" and "Star". For details, see "7.6.2 Using the Netlist Editor" on page 7-23. This feature is not available in all versions of Ultiboard.
Trace Length	Length of the routed net.
Max Length	Maximum allowable length of copper in a net. Not applicable (N/A) if Topology is set to "Shortest".
Min Length	MInimum allowable length of copper in a net. Not applicable (N/A) if Topology is set to "Shortest".
Trace Clearance	This is the minimum spacing allowed between the components in the group and any trace. You enter a value here, or use the Group Editor. For details, see "5.5 Working with the Group Editor" on page 5-14.
Routing Layers	The routing layers assigned to this net group. Click to display the "Layers to Apply" dialog box, where you set the layers to use for routing copper.
Routing Priority	The routing priority for the selected net. "1" is the highest priority, "2" the second highest, etc. Leave as None if priority routing is not required. This feature is not available in all versions of Ultiboard.
Net Group	This is the group in which the net is contained. You can either enter the group name here, or use the Group Editor. For details, see "5.5 Working with the Group Editor" on page 5-14.
Bus Group	This is the bus group in which the net is contained. You can either enter the group name here, or use the Group Editor. For details, see "5.5 Working with the Group Editor" on page 5-14.
Differential Pair	This is the differential pair to which the net belongs. For details, see "5.5 Working with the Group Editor" on page 5-14.
Shield Net	Use the drop-down list to select the net to use to shield this net. This feature is not available in all versions of Ultiboard.

Column	Description
Shield Width	Width of the net's shield. This feature is not available in all versions of Ultiboard.
Show Ratsnest	This is where you can show or hide the ratsnest for the selected net. For more information on ratsnests, see "6.1.3.1 Working with Ratsnests" on page 6-5.
Max Via Count	This is the maximum number of vias allowed for the selected net. You can use either the global settings, or the group settings.
Via Drill Diameter	This is the diameter of the via pad's hole.
Via Pad Diameter	This is the diameter of the total via pad.

3.6.6   Spreadsheet View: Nets Group Tab

The **Net Groups** tab lets you work with net groups.

Column	Description
Net Group	This is the group in which the net is contained. You can either enter the group name here, or use the Group Editor. For details, see "5.5 Working with the Group Editor" on page 5-14.
Trace Width	The "default" width of the traces in the group that are placed during routing. You can enter a value here, or use the Group Editor. For details, see "5.5 Working with the Group Editor" on page 5-14.
Max Width	The maximum width to which a trace in the group will be laid during routing. You can enter a value here, or use the Group Editor. For details, see "5.5 Working with the Group Editor" on page 5-14.
Min Width	The minimum width to which a trace in the group will be laid during routing. You can enter a value here, or use the Group Editor. For details, see "5.5 Working with the Group Editor" on page 5-14.
Max Length	Maximum allowable length of copper in a net group. Not applicable (N/A) if Topology is set to "Shortest".
Min Length	MInimum allowable length of copper in a net group. Not applicable (N/A) if Topology is set to "Shortest".

Column	Description
Trace Clearance	This is the minimum spacing allowed between the traces in the group and any other trace. You can enter a value here, or use the Group Editor. For details, see "5.5 Working with the Group Editor" on page 5-14.
Routing Layers	The routing layers assigned to this net group. Click to display the "Layers to Apply" dialog box, where you set the layers to use for routing copper.
Routing Priority	The routing priority for the selected net group. "1" is the highest priority, "2" the second highest, etc. Leave as None if priority routing is not required. This feature is not available in all versions of Ultiboard.
Locked	"Yes" indicates that the component cannot be moved. "No" indicates that the component can be moved.
Max Via Count	This is the maximum number of vias allowed in the net group.

3.6.7　Spreadsheet View: SMT Pads Tab

The **SMT Pads** tab lets you work with pad information for surface-mount devices.

Column	Description
Pad Name	The unique identifier for the pad, by Refdes and Pin Number. For example, U1, P1 is pin number one of the device with Refdes U1.
Pad Shape	The shape of the pad as set in the SMT Pin Properties dialog box. For details, see "6.5.4 Viewing and Editing SMT Pin Properties" on page 6-47.
Pad Radius	The radius of the pad as set in the SMT Pin Properties dialog box. For details, see "6.5.4 Viewing and Editing SMT Pin Properties" on page 6-47.
Pad Width	The width of the pad as set in the SMT Pin Properties dialog box. For details, see "6.5.4 Viewing and Editing SMT Pin Properties" on page 6-47.
Pad Height	The height of the pad as set in the SMT Pin Properties dialog box. For details, see "6.5.4 Viewing and Editing SMT Pin Properties" on page 6-47.
Trace Clearance	The clearance of the trace to components. You can use net settings, or as set in the SMT Pin Properties dialog box. For details, see "6.5.4 Viewing and Editing SMT Pin Properties" on page 6-47.
Neck Length	The default length of the neck where the trace attaches to the pin, as set in the SMT Pin Properties dialog box. For details, see "6.5.4 Viewing and Editing SMT Pin Properties" on page 6-47. You can also enter the value here.

Column	Description
Neck Max	The maximum permitted length of the neck where the trace attaches to the pin, as set in the SMT Pin Properties dialog box. For details, see "6.5.4 Viewing and Editing SMT Pin Properties" on page 6-47. You can also enter the value here.
Neck Min	The minimum permitted length of the neck where the trace attaches to the pin, as set in the SMT Pin Properties dialog box. For details, see "6.5.4 Viewing and Editing SMT Pin Properties" on page 6-47. You can also enter the value here.
Min Width	The minimum permitted trace width.

3.6.8   Spreadsheet View: THT Pads Tab

The **THT Pads** tab lets you work with pad information for through-hole technology devices.

Column	Description
Pad Name	The unique identifier for the pad, by Refdes and Pin Number. For example, U1, P1 is pin number one of the device with Refdes U1.
Top Pad Shape	The shape of the top layer pad as set in the Through Hole Pin Properties dialog box. For details, see "6.5.3 Viewing and Editing Through Hole Pin Properties" on page 6-43.
Inner Pad Shape	The shape of the inner layer pad as set in the Through Hole Pin Properties dialog box. For details, see "6.5.3 Viewing and Editing Through Hole Pin Properties" on page 6-43.
Bottom Pad Shape	The shape of the bottom layer pad as set in the Through Hole Pin Properties dialog box. For details, see "6.5.3 Viewing and Editing Through Hole Pin Properties" on page 6-43.
Annular Ring	The size of the annular ring for this pad as set in the Through Hole Pin Properties dialog box. For details, see "6.5.3 Viewing and Editing Through Hole Pin Properties" on page 6-43.
Pad Diameter	The diameter of the pad as set in the Through Hole Pin Properties dialog box. For details, see "6.5.3 Viewing and Editing Through Hole Pin Properties" on page 6-43.
Drill Diameter	The diameter of the drill hole in the pad, as set in the Through Hole Pin Properties dialog box. For details, see "6.5.3 Viewing and Editing Through Hole Pin Properties" on page 6-43.

Column	Description
Trace Clearance	The minimum clearance permitted between the pad and traces, as set in the Through Hole Pin Properties dialog box. For details, see "6.5.3 Viewing and Editing Through Hole Pin Properties" on page 6-43.

3.6.9   Spreadsheet View: Vias Tab

The **Vias** tab lets you work with via information.

Column	Description
Assume Net	The unique identifier for the net to which the via is connected. Can be changed by using the drop-down list.
Board Side	The side of the board on which the via appears.
Pad Diameter	The diameter of the via as set in the Via Properties dialog box. For details, see "7.4.2 Viewing and Editing Via Properties" on page 7-17.
Drill Diameter	The diameter of the drill hole in the via, as set in the Via Properties dialog box. For details, see "7.4.2 Viewing and Editing Via Properties" on page 7-17.
Trace Clearance	The minimum clearance permitted between the via and traces, as set in the Via Properties dialog box. For details, see "7.4.2 Viewing and Editing Via Properties" on page 7-17.
Locked	"Yes" indicates that the via cannot be moved. "No" indicates that the via can be moved.
Soldermask	The side of the board on which the solder mask for the via is found. Choices are: Both; Bottom; None; Top.

3.6.10 Spreadsheet View: Copper Areas Tab

The **Copper Areas** tab lets you work with information for copper areas.

Column	Description
Layer	The layer on which the copper area is found, as set in the Copper Area Properties dialog box. For details, see "7.3 Viewing and Editing Copper Properties" on page 7-14.
Locked	"Yes" indicates that the copper area cannot be moved. "No" indicates that the copper area can be moved.
Net	The unique identifier for the net to which the copper area is connected.
Trace Clearance	The minimum clearance permitted between the copper area and traces, as set in the Copper Area Properties dialog box. For details, see "7.3 Viewing and Editing Copper Properties" on page 7-14.
Thermal Relief Style	The style of thermal relief for the copper area, as set in the Copper Area Properties dialog box. For details, see "7.3 Viewing and Editing Copper Properties" on page 7-14.

3.6.11 Spreadsheet View: Keep Ins/Outs Tab

The **Keep Ins/Outs** tab lets you work with information for keepin or keepout areas.

Column	Description
Name	Name of the keepin/out. Can be entered here, or in the Keepin/out Properties dialog box. For details, see "7.1.7 Working with Keep-in/Keep-out Areas" on page 7-7.
Type	Indicates whether the area is a keepin or a keepout. Can be set here or in the Keepin/out Properties dialog box. For details, see "7.1.7 Working with Keep-in/Keep-out Areas" on page 7-7.
Locked	"Yes" indicates that the keepin/out area cannot be moved or edited. "No" indicates that the keepin/out area can be moved.
Layers To Apply	Double-click to display the Layers To Apply dialog box, where you select the layers to which you wish to apply the keepin/out. You can also set this in the the Keepin/out Properties dialog box. For details, see "7.1.7 Working with Keep-in/Keep-out Areas" on page 7-7.

Column	Description
Net Group	Double-click to display the Select Groups dialog box, where you select the net group to which you wish to apply the keepin/out. You can also set this in the the Keepin/out Properties dialog box. For details, see "7.1.7 Working with Keep-in/Keep-out Areas" on page 7-7.
Component Group	Double-click to display the Select Groups dialog box, where you select the component group to which you wish to apply the keepin/out. You can also set this in the the Keepin/out Properties dialog box. For details, see "7.1.7 Working with Keep-in/Keep-out Areas" on page 7-7.
Heights Bigger Than	Assign a height (z-axis) value to the keepin/out. Can be entered directly, or in the Keepin/out Properties dialog box. For details, see "7.1.7 Working with Keep-in/Keep-out Areas" on page 7-7.

3.6.12 Spreadsheet View: Layers Tab

The **Layers** tab lets you work with layer information.

Column	Description
Layer Name	The name of the layer, for example, Copper Top.
Routable Layer	Select "Yes" to allow trace routing on the layer; select "No" to prevent trace routing on the layer.
Trace Bias	Set the trace bias by selecting "H" for horizontal; "V" for vertical; or "None".
Type	The type of layer. Choices are Ground, Power, Signal or Unassigned.

3.6.13 Spreadsheet View: Parts Position Tab

The **Parts Position** tab lets you view and export part position information.

Column	Description
RefDes	The part's Reference Designator.
Position X	The part's position on the X axis.
Position Y	The part's position on the Y axis.
Side	The side of the PCB on which the part appears.
Rotation	The orientation of the part on the PCB.

3.6.14 Spreadsheet View: Statistics Tab

This tab displays the statistics shown below.

Total number of pins:	190
Pins in a net:	159
Not connected pins:	31
Test pins:	0
Jumpers:	0
Total number of vias:	35
Total number of connections:	123
Unrouted connections:	0
Completion:	100%
Total number of components:	24
Total number of nets:	36

3.7   Customizing the Interface

The Ultiboard user interface is highly customizable. Toolbars may be docked in various positions and orientations. The contents of the toolbars may be customized. New toolbars may be created. The menu system is fully customizable, including all pop-up menus for the various object types.

As well, the keyboard shortcut system is customizable. This allows for any keys or key combinations to be assigned to any command that may be placed in a menu or on a toolbar.

➤ To customize the interface:

1. Select **Options/Customize User Interface**.

2. Make changes in the **Customize** dialog's tabs as detailed in the following sections:

   • "3.7.1 Commands Tab" on page 3-45
   • "3.7.2 Toolbars Tab" on page 3-46
   • "3.7.3 Keyboard Tab" on page 3-47
   • "3.7.4 Menu Tab" on page 3-48.
   • "3.7.5 Options Tab" on page 3-49

3.7.1   Commands Tab

The **Commands** tab in the **Customize** dialog box is used to add commands to menus and toolbar.
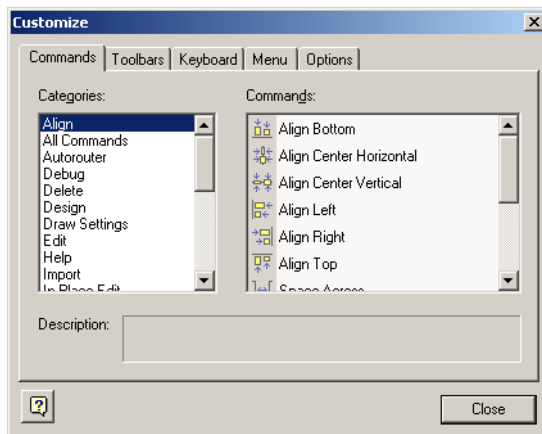
➤ To add a command to a menu or toolbar:

1. Drag it from the **Commands** list to the desired menu or toolbar. When a command is selected in the **Command** list, its description is displayed in the **Description** field.

2. If you do not see the command that you require, click on another selection in the **Categories** list to display more commands.

3. If you wish to add a button to a toolbar that is not showing, click on the **Toolbars** tab and click in the checkbox beside the desired toolbar to make it visible.

4. Click **Close** when customizations are complete.

➤ To remove a command from a menu or toolbar, right-click on it and select **Delete** from the pop-up that appears. The **Customize** dialog box must be open when you do this.

➢ To change the position of a command that is in a menu or toolbar, drag it to its new location. The **Customize** dialog box must be open when you do this.

3.7.2   Toolbars Tab

The **Toolbars** tab in the **Customize** dialog box is used to show or hide toolbars, and to add new custom toolbars.
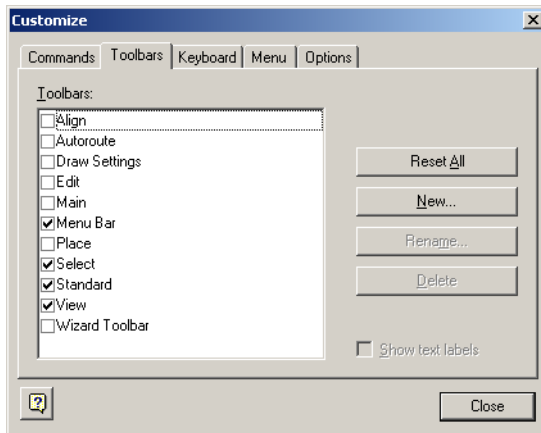
➢ To use the features in this tab:

1. To display a toolbar, switch on the checkbox beside the desired toolbar in the **Toolbars** list.

2. Switch off a checkbox to hide a toolbar.

Note   You cannot switch off the **Menu** bar.

3. The buttons in this tab function as follows:

   • **Reset All**—displays the **Reset Toolbars** dialog box, where you select whether to reset the currently selected toolbars, or all toolbars. You are prompted to select the configuration file you wish to use, for example, "default.ewcfg".
   • **New**—displays the **Toolbar Name** dialog box, where you enter the name for a new toolbar. When you click **OK**, a new toolbar with the name that you entered is created. Follow the steps in "3.7.1 Commands Tab" on page 3-45 to add buttons to the toolbar.
   • **Rename**—use to rename a toolbar that you have created yourself. You cannot rename toolbars that are included in Multisim by default. for example, **Component**s, **Menu Bar**.
   • **Delete**—use to delete the selected toolbar. You cannot delete toolbars that are included in Multisim by default. for example, **Component**s, **Menu Bar**.
   • **Show text labels**—select this checkbox to show the text labels (for example, "Save") in the toolbar, along with the command's icon.

4. Click **Close** when customizations are complete.

3.7.3   Keyboard Tab

The **Keyboard** tab is used to set up keyboard shortcuts.
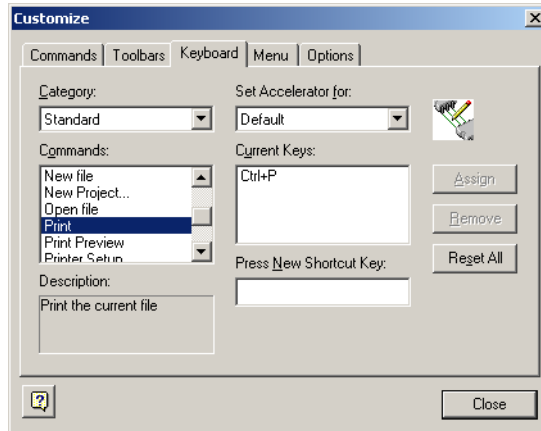
➢ To set up keyboard shortcuts:

1. Choose a menu from the **Category** drop-down list and the desired command from the **Commands** list. If a shortcut is already assigned, it appears in the **Current Keys** field.

2. Enter a new shortcut in the **Press New Shortcut Key** field.

3. Click **Close** when customizations are complete.

3.7.4   Menu Tab

The **Menu** tab is used to modify the various context-sensitive menus that appear when you right-click from various locations in Ultiboard.
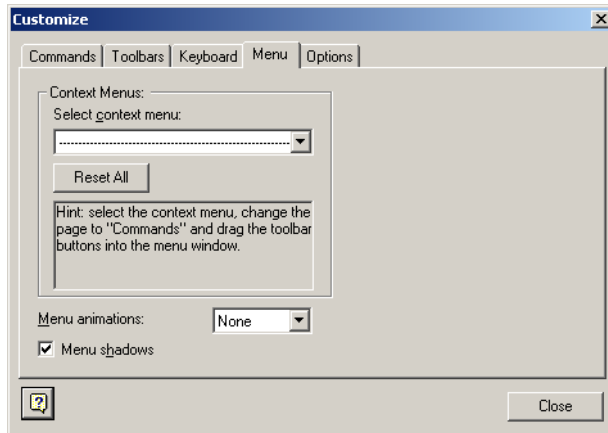
➢ To display the desired menu:

1. Select the desired menu set from the **Select Context Menu** drop-down list.

2. Right-click on the menu that appears and edit as desired.

3. Select the desired menu effects using the **Menu animations** drop-down list and the **Menu shadows** checkbox.

3.7.5   Options Tab

The **Options** tab in the **Customize** dialog box is used to set up toolbar and menu options.
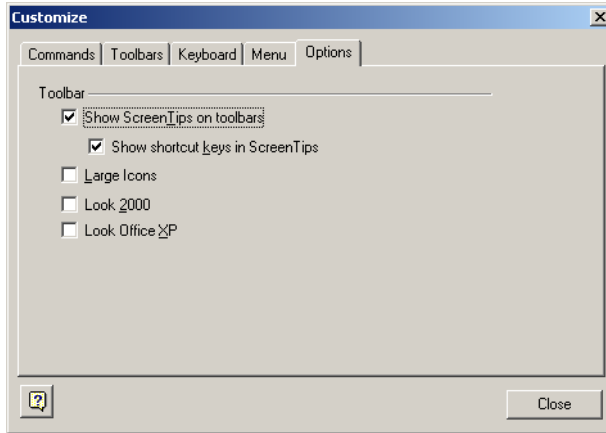
➢ To set up menu and toolbar options, switch the checkboxes on or off as desired.

3.7.6   Customization Pop-up Menus

To customize the appearance of toolbar buttons and menu items, a pop-up menu is available when the **Customize** dialog box is open.
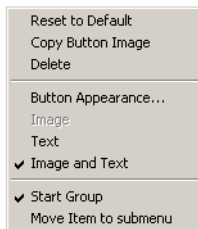
➢ To display the above pop-up:

1. Be sure you have the **Customize** dialog box open.
2. Right-click on either a menu item or toolbar and select the desired option.

3. When you select **Button Appearance**, the **Button Appearance** dialog box appears, where you can change the appearance of the selected toolbutton.
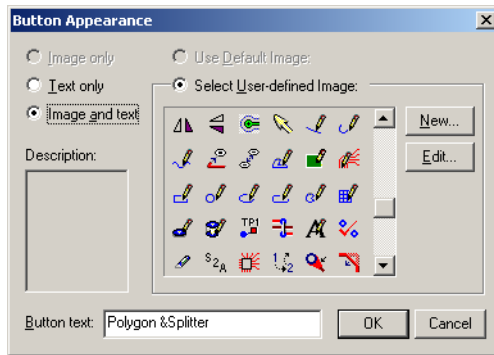
Chapter 4
Beginning a Design

This chapter explains how to start a design in Ultiboard.

The following are described in this chapter.

Subject	Page No.
About Designs and Projects	4-2
Creating a Project	4-2
Creating a Design	4-3
Importing a Netlist File	4-3
Working with Projects	4-6
Opening an Existing File	4-6
Saving and Closing	4-7
Saving Technology Loading Technology Files	4-8 4-9
Selecting and Unselecting Elements	4-10
Using Selection Filters	4-10
Working with Modes	4-10
Searching for Design Elements	4-11
Options for Viewing the Design Using the Full Screen Magnifying and Shrinking the View Refreshing the Design Tool-tip Label	4-12 4-12 4-13 4-13 4-14

4.1 About Designs and Projects

Designs are stored inside projects, allowing you to group them together for easy access. In this way, all designs that have a logical connection between them (that is, they may all relate to a specific project) are conveniently located in one file.

Depending on your version of Ultiboard, you can have as many projects and designs open as you wish.

4.2 Creating a Project

➢ To create a new project file:

1. Choose **File/New Project**. The **New Project** dialog box appears:

Type a name for the project file

Select PCB Design

Accept the default location for the project

Or

Click the Browse button to select a location
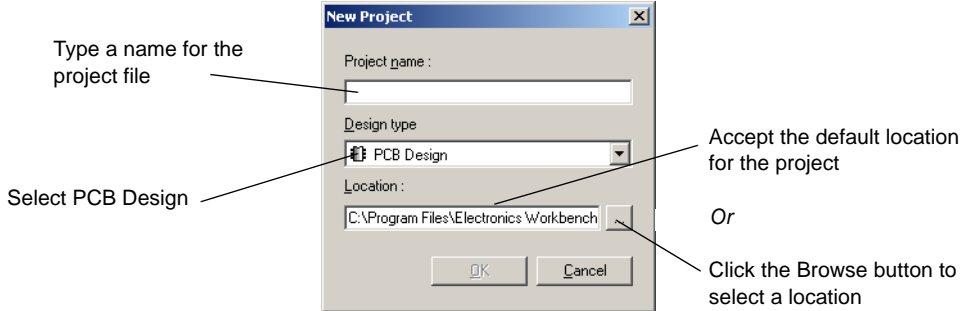
2. Type the file name in the **File name** field.

3. From the **Design Type** drop-down list, select PCB Design. The other option allows you to use Ultiboard's mechanical CAD capabilities (for front panels, enclosures, etc.). For more on these capabilities, see " Using Mechanical CAD" on page 12-1.

4. Indicate the folder where the file is to be kept. Either accept the default shown in the **Location** field, or click the browse button to select a different location.

5. Click **OK**. The **New Project** dialog box disappears, and a blank design is opened, with the same name as that of the project file. The file you just created is also shown in the **Projects** tab of the **Design Toolbox**, along with its corresponding design.

Note To give a new name to a design, right-click on it and choose **Rename**.

4.3   Creating a Design

➤ A design is created automatically when you create a project file. You can also create a design and assign it to an existing project file:

1. Choose **File/Open** and open the project to which the design is to be added, if it is not already open.

2. Choose **File/New Design**. The **New Design** dialog box appears:
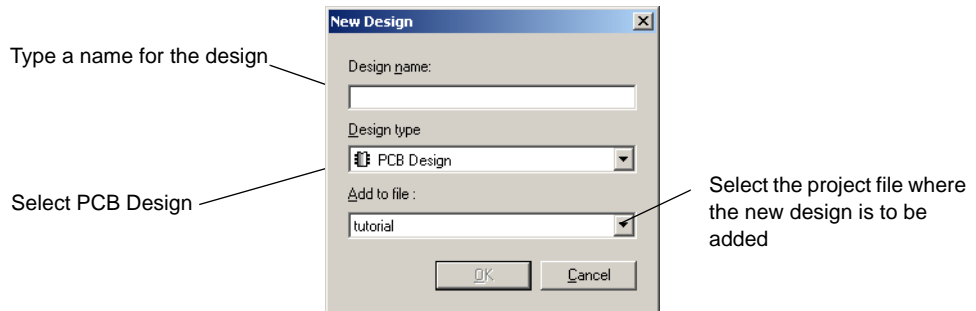
Type a name for the design

Select PCB Design

Select the project file where the new design is to be added

3. Type the design name in the **Design Name** field.

4. From the **Design Type** drop-down list, select PCB Design. (For more information on the Mechanical CAD Design file type, see " Using Mechanical CAD" on page 12-1.)

5. Choose which of the open projects is to contain the design. The **Add to file** drop-down list shows only the open projects.

6. Click **OK**. The **New Design** dialog box disappears, and a blank design with the name that you specified is opened. The **Projects** tab shows that the new design is a part of the file that you specified.

4.4   Importing a Netlist File

You can create a design based on a netlist file. A netlist contains information about a given board's nets (the electrical connections between pins) and components. Multisim generates a `.EWNET` file which has all the details required to import the schematic into Ultiboard.

Note   The file also contains Trace Width Analysis results if you checked the **Set Node Trace Widths Using the Results from This Analysis** check box when doing the analysis in Multisim. If PCB constraints are set in the Multisim file, they will also be included in the netlist file.

.

```
(ToolInfo
        (netlist "ULTIboard" 8 0 0)
        (tool "Multisim" 8 0 0)
        (timestamp "15:4:0"  "6-1-2005")
        (version 3 0 0)
        (gateswap 2)
        (pinswap 1)
        (crossprobe {8530DE70-86AA-464F-AA09-1A35B78302C7})
        (units Mil)
)
(nets
        (net "10"
                (trackwidth "-1.00")
                (trackwidth_max " -1.00")
                (trackwidth_min "589.30")
                (tracklength_max " -1.00")
                (tracklength_min " -1.00")
                (clearance_to_trace " -1.00")
                (clearance_to_pad " -1.00")
                (clearance_to_via " -1.00")
                (clearance_to_copper " -1.00")
                (routing_layer "Copper Top")
                (settings_locked "0")
                (net_group "")
        )


(components
        (instance "7SEG8DIP10A" "U1"
                (device "SEVEN_SEG_DECIMAL_COM_A_BLUE")
                (value "SEVEN_SEG_DECIMAL_COM_A_BLUE")
                (gateswap "0")
                (pinswap "0")
                (component_space "  0.00")
                (component_group "")
                (settings_locked "0")
                (comp_variants "Default1;")
                (comp_variant_independent "0")
                (pin "5"
                        (net "10")
                        (pintype "BIDIR")
                        (gategroup "")
                        (pingroup "")
                        (label "D")
                        (gate "")
                )
                (pin "4"
                        (net "11")
                        (pintype "BIDIR")
                        (gategroup "")
                        (pingroup "")
                        (label "E")
                        (gate "")
                )
                (pin "2"
                        (net "12")
                        (pintype "BIDIR")
                        (gategroup "")
                        (pingroup "")
                        (label "F")
                        (gate "")
```

Example of Net Information in a netlist file.

A value of -1.00 indicates that the Ultiboard default value will be used for that parameter when the file is imported into Ultiboard.

Example of Component Information in a netlist file.

➢ To import a netlist file:

1. Select **File/Import/UB Netlist**, navigate to the desired file (e.g., Tut2.EWNET) and click **Open**. The following dialog appears.
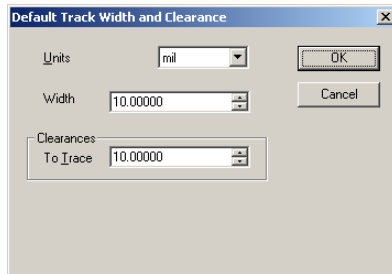
2. Enter the desired parameters and click **OK**.
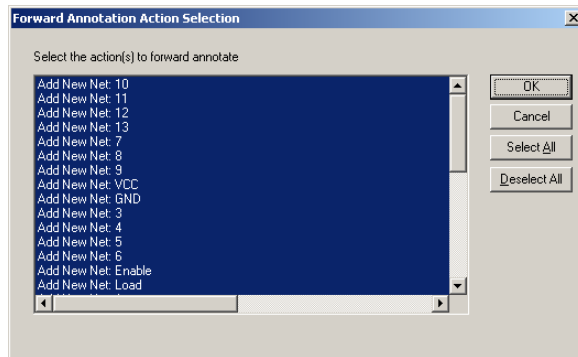
3. Select the desired actions (default is All) and click **OK**. A board outline is placed on the workspace with the components ready to be placed.

4.5   Working with Projects

Designs and projects appear in the **Projects** tab of the **Design Toolbox**. .
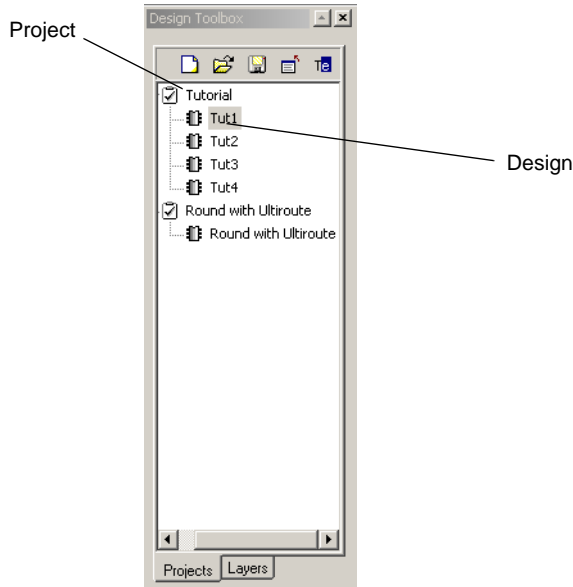
Project

Design

> ➤ To open a project or design shown in the **Projects** tab, click on it or right-click on it and, from the context menu, choose **Open Window**.

> ➤ To rename a design, right-click on it and, from the context menu, choose **Rename**. You can then type a new name for the design.

> ➤ To delete a design shown in the **Projects** tab, right-click on it and, from the context menu, choose **Remove Design**. (To delete a project file, delete it from its current location on the system.)

4.6   Opening an Existing File

> ➤ To open an existing file:

1. Choose **File/Open**. A standard file section dialog box appears, with the **Files of Type** list defaulted to Ultiboard files.

2. In the **Files of Type** list, choose the kind of file to open. You can open the following:

   • Ultiboard project files (`*.ewprj`)
   • Ultiboard 7 and Ultiboard 6 files

- Orcad files (*.max) Orcad Library (`*.llb`)
- Protel files (`*.pcb`, `*.ddb`)
- Gerber files (`*.g`)
- DXF files (`*.dxf`)
- Calay Netlist files (`*.*.net`)
- TC7 files (`*.tc7`)

Note To view all files in the folder, both those created by Electronics Workbench software and any other software, select `*.*`.

3. Select a file from the list displayed and click **Open**. The file opens, along with its associated project.

- If you opened an Ultiboard project file, you see the designs within that project in the **Projects** tab:
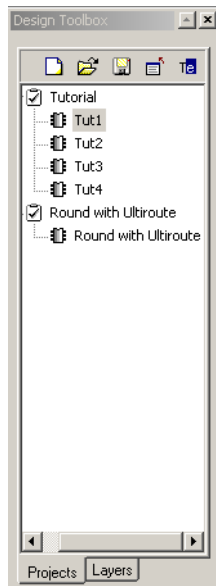
- If you opened a version 5 `.ddf` file, you see the design appear in the workspace. A file with the same name is created automatically.

4.7   Saving and Closing

➢ To save a file, right-click on the file name in the **Projects** tab and, from the context menu, choose **Save** or choose **File/Save**. Saving a file also saves the designs in the file. You can also

select a file in the **Projects** tab and click the save icon. Saving a design also saves a file (and therefore any other designs in the file).

➢ To save a file with a new name and/or location, choose **File/Save As** and provide the new name and/or location. All designs in the file are saved in the new location.

➢ To save all open file and designs, choose **File/Save All**.

➢ To close the current file and its designs, choose **File/Close**. If you have any unsaved changes in the fiile or designs, you are prompted to save the file and/or designs.

➢ To close all open projects and designs, choose **File/Close All**. If you have any unsaved changes in the projects or designs, you are prompted to save the projects and/or designs.

4.8   Saving Technology

➢ You can save the technology specifications from a current design into a technology file. This technology file can be used to import technology settings into another design. To save technology:

1. Select **File/Save Technology**. The **Technology File Settings** dialog box appears.
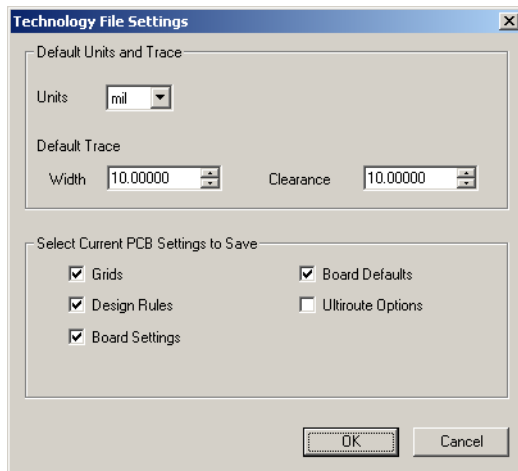
2. Select the technology parameters you want to save and click **OK**. A standard Windows Save dialog appears. Select the desired filepath for the technology file, enter its name in the **File name** field and click **Save**.

For details on loading a technology file, see "4.8.1 Loading Technology Files" on page 4-9.

4.8.1   Loading Technology Files

You can load technology from a technology file that you saved earlier, or from another Ultiboard file.

➢ To load technology into the active file:

1. Select **File/Load Technology**. A standard Windows Open dialog appears.
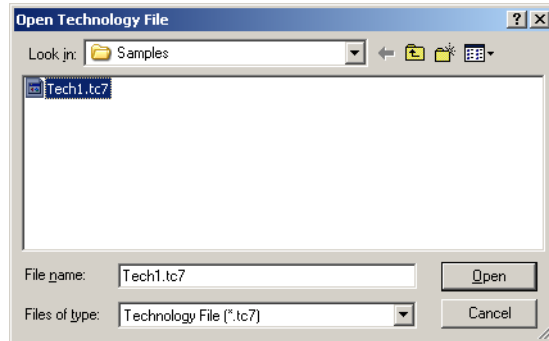
2. Navigate to the desired folder and highlight the desired technology file and click **Open**. The following dialog displays.
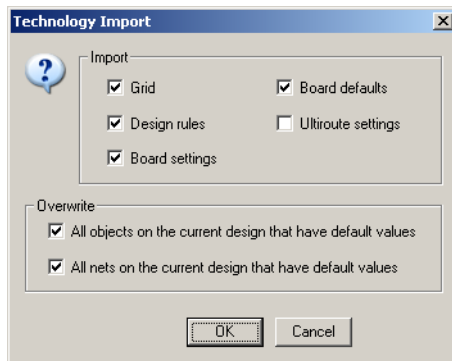
3. Select the desired items to load into your open design file and click **OK**.

For details on saving technology, see "4.8 Saving Technology" on page 4-8.

4.9   Selecting and Unselecting Elements

➢ To select a single element on a board, click the element. A dotted line around the element, or running through the trace, indicates that it has been selected.

➢ To select multiple elements on a board:

Hold the SHIFT key down while clicking the elements that you want to select.

Or

Click and drag to draw a box around the elements to be selected. When you release the mouse button, the elements that were inside the box are selected.

➢ To select all elements on a board, choose **Edit/Select All** or press CTRL+A.

➢ To select an entire trace, not just a trace segment:

1. Select a segment belonging to the trace you want to select.

2. Choose **Edit/Select Entire Trace**. The entire trace is then selected.

➢ To unselect one element, select another element. This unselects the first.

➢ To unselect one element in a group of selected elements, hold the CTRL key down while clicking the element to be unselected.

4.10   Using Selection Filters

As you add more components and traces to a board, it can become more difficult to select just the elements which you want to use. Ultiboard provides you with selection filters to allow controlled selections. The selection filters are found in **Edit/Selection Filter** and the **Select** toolbar. By default, all the selection filters are on, that is, you can select any element. Use the filters to select or not select specific elements by toggling the choices on and off. For example, to select only traces, you would disable the other selection filters and enable only the traces one. As you work through your design, you will find different combinations of selection filters helpful to prevent accidentally selecting (and potentially modifying) elements.

4.11   Working with Modes

Ultiboard assumes that placing shapes, parts, or traces on a board are actions you are likely to repeat. As a result, when you place items on the board, you remain in "place mode" (the cursor has a small icon attached, indicating what is being placed) so that you can continue to place the same type of item repeatedly. After placing a shape, part, or trace on a board, the

pointer continues to carry the icon, and is ready to place another object like the one you just placed. You must cancel this mode and return to "select mode" in order to perform other functions.

➢ To cancel the place mode and return to select mode:

Right-click

Or

Press ESC

Or

Choose **Place/Select**.

4.12   Searching for Design Elements

To find out if an element exists in an open project, you can search for it with the **Edit/Find** command. While this command works much like a Find function in other applications, it also allows you to search for an element by name, by number, by shape, by value, or by all variables.

➢ To find an element in an open design:

1. Choose **Edit/Find**. The **Find** dialog box appears.
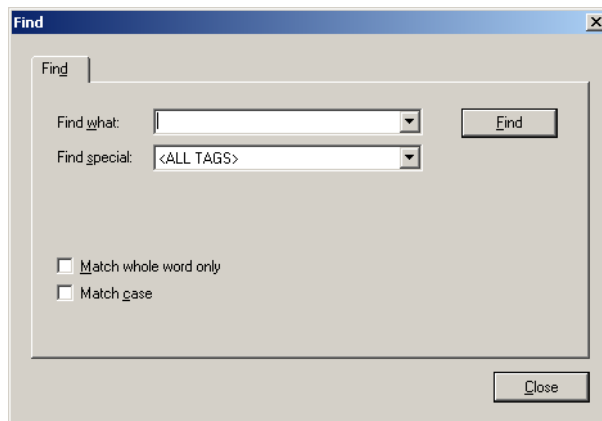
2. In the **Find what** field, enter what you want to search for. You can search for an element by specifying any of the values of its attributes (for example, name, number, shape), with or without wildcards. If you leave this field blank, you will find all elements in the design.

Note   The **Find what** drop-down list shows all the searches you have made in this session.

3. To constrain your search to only certain attributes, choose from the **Find special** drop-down list. For example, if you enter "test" in the **Find what** field, you will find all elements with the string "test" in any of their attributes. However, if you choose **Name** from the **Find special** drop-down list, you will find only elements with "test" in their name.

4. Optionally, enable the options to match the whole word or to match the case.

5. Click **Find**. The search results appear in the **Results** tab of the **Spreadsheet View**.

```
Ultiboard  -  2005-05-20 10:50:18

Searching For 'U2' ...
   Default | tutorial | Tut1: Tag = REFDES, Value = U2
   Default | tutorial | Tut3: Tag = REFDES, Value = U2
   Default | tutorial | Tut4: Tag = REFDES, Value = U2
   Default | tutorial | Tut2: Tag = REFDES, Value = U2
4 occurrence(s) have been found.
```
Results | DRC | Parts | Part Groups | Nets | Net Groups
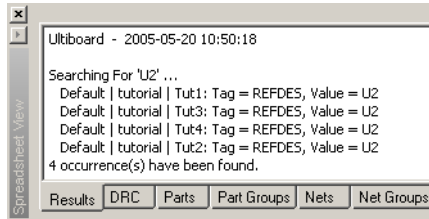
6. Click on the **Results** tab.

7. To navigate to any of the found elements, double-click them in the **Results** tab.

Or

Right-click on the desired element and select **Go to** from the pop-up menu that appears.

The element is selected and shown in the workspace.

➢ To clear the information in the **Results** tab, right-click anywhere in the **Results** tab and select **Clear** from the pop-up menu that appears.

4.13 Options for Viewing the Design

Your view of the design can be manipulated a number of ways to make things easier to see while editing, as well as to see what the finished design will look like.

4.13.1 Using the Full Screen

When you open Ultiboard for the first time, the program displays the workspace with the menu and toolbars, the **Birds Eye View** and the **Design Toolbox**. To get a better view of the workspace, you can either use the zoom functions to magnify it (for more on the zoom functions, see "Magnifying and Shrinking the View"), or have the workspace take up the full screen.

When you display a design on the full screen, everything except the design disappears (depending on your settings in the **Preferences** dialog box, scrollbars may or may not appear). Menu functions can still be used through their keyboard equivalents—for example, you can

use F8 to zoom in, and F9 to zoom out. Again, depending on the **Preferences** dialog box settings, you may be able to pan through the design by moving your cursor over the outside edges.

➢ To switch the workspace to a full screen display, choose **View/Full Screen**.

➢ To return from the full screen display to the multi-screen display:

1. Locate the **Full Screen Close** button. It will be floating over the design:

2. Click the **Full Screen Close** button to return to the multi-screen display.

4.13.2 Magnifying and Shrinking the View

You can use the zoom functions to magnify or shrink all or part of the design.

➢ To magnify part of the design, choose **View/Zoom In**, or press F8. You may need to adjust your view of the magnified design by using the scroll bars.

➢ To magnify a selected area on the design:

• Choose **View/Zoom Window**, or press F6, then click and drag a rectangular area on the board to define the area to be zoomed in on.

Or

• Define an area by clicking and dragging on the **Birds Eye View**.

➢ To shrink the view of the design, choose **View/Zoom Out**, or press F9.

➢ To return to the full view of the design after zooming in or out, press F7.

4.13.3 Refreshing the Design

After adding and changing elements, the design can begin to look a little confusing, with bits and pieces of elements looking like they have been left behind on the design after being moved, for example. This can be for many reasons, including the limitations of the computer monitor being used, although it does not affect the design.

➢ To clean up the design, removing any extraneous images that should not be there:

Choose **View/Redraw Screen**.

Or

Click **Redraw** in the **View** toolbar or press F5.

4.13.4 Tool-tip Label

➢ To change your cursor to a tool-tip label that contains object information, hold down the CTRL key as you move your mouse over the workspace. The information in the label will change depending on the type of component over which it hovers.

Tool-tip Label
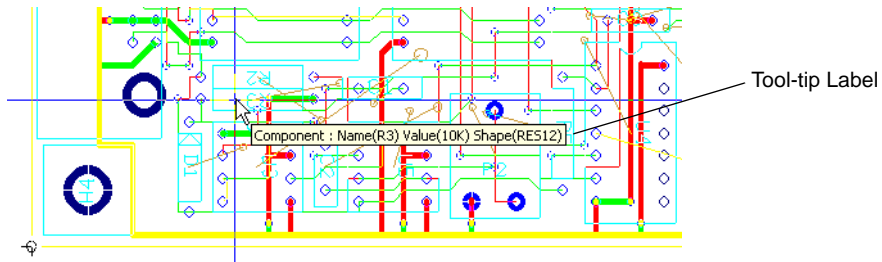
Chapter 5
Setting Up a Design

This chapter explains the basic functions you must perform to get your board set up. These tasks should be performed before you begin placing components and/or traces.

The following are described in this chapter.

5.1 Working with Layers

The following are discussed in this section:

❒ "5.1.1 Defining Copper Layers" on page 5-2

❒ "5.1.2 Accessing Layers" on page 5-3

5.1.1 Defining Copper Layers

Ultiboard lets you define boards from 2 to 64 layers thick. Before you can create multi-layered boards, you need to know how they are to be manufactured.

Your initial design decisions are important because it is difficult to change the design, for example, from a 6-layer with blind and buried vias to a normal feed-through design, after the board has been completed. Your decisions are also important in terms of manufacturing cost. A 6-layer board with blind and buried vias will cost significantly more to manufacture than a 4-layer board with normal feed-through vias.

When setting the number of copper layers you also select the lamination settings that will be used when the board is manufactured. This is important as these settings are used by Ultiboard's internal Design Rule engine when placing blind and buried vias or micro vias.

There are two methods of fabricating a PC board (both methods can be used in combination):

- The first method uses layered pairs of copper on a substrate (usually cured fiberglass/resin) that have been etched and are then laminated together with a partially cured fiberglass/resin substrate (prepreg). Heat and mechanical pressure are used to activate the prepreg and bond layered pairs with other layered pairs.
- The other method typically uses a layered pair as a core to which single layers of copper foil are added to build up the board. Prepreg is also used to bond the layers. Build-up layers are usually added in equal numbers to the top and bottom of the core to prevent warping of the final product.

A via is a plated through-hole in a printed circuit board used to connect two or more layers, as well as the top and bottom surfaces of the board.

- *Blind via* — any via that connects the top or bottom layer of a board to one of the internal layers.
- *Buried via* — any via that connects internal layers.
- *Normal through-hole via* — any via that connects all layers (top, bottom, and internal).
- *Micro via* — a via that is less than 5 mils in diameter, that connects a maximum of two build-up layers only.

The lamination sequence used determines the acceptable layer combinations for placing blind and buried vias or microvias. For example, if all layers are layered pairs then blind vias can only be placed between an outer layer and completely through a layered pair (see diagram for an example). Once the lamination sequence is determined, Ultiboard will calculate the

acceptable layer combinations for blind and buried vias or micro vias. You then choose which ones to use in your design from these possible combinations.

Cross-section of
multi-layer PCB

☐ Normal through-hole via
■ Blind vias
▨ Buried via
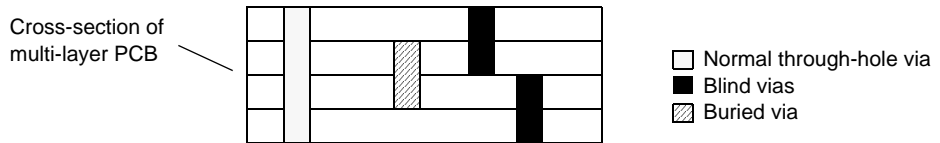
The number of copper layers in a board can be set in either the **Board Wizard** (see "5.2.4 Using the Board Wizard" on page 5-7), or in the Board Settings (see "3.4.3 Copper Layers Tab" on page 3-23).

5.1.2   Accessing Layers

The **Layers** tab of the **Design Toolbox** allows you to move through the layers of your design, as well as control their appearance.

➢ To display the **Layers** tab, click **Layers** (in the **Design Toolbox**). The tab appears, as shown here:

Layers with a check mark are displayed on the workspace.

Layers with a check mark in a grey box are displayed in the workspace, but are dimmed.

Layers with no check mark are not displayed in the workspace, but are available to be used.
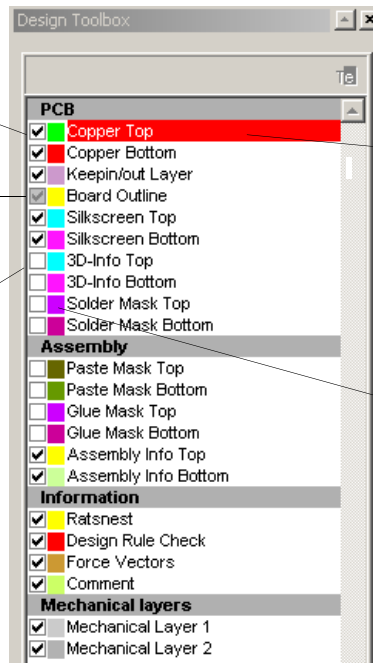
The highlighted layer is the active layer, the one you are working with.

Use the color icons to control the layer's color.

Design Toolbox

PCB
- ☑ Copper Top
- ☑ Copper Bottom
- ☑ Keepin/out Layer
- ☑ Board Outline
- ☑ Silkscreen Top
- ☑ Silkscreen Bottom
- ☐ 3D-Info Top
- ☐ 3D-Info Bottom
- ☐ Solder Mask Top
- ☐ Solder-Mask Bottom

Assembly
- ☐ Paste Mask Top
- ☐ Paste Mask Bottom
- ☐ Glue Mask Top
- ☐ Glue Mask Bottom
- ☑ Assembly Info Top
- ☑ Assembly Info Bottom

Information
- ☑ Ratsnest
- ☑ Design Rule Check
- ☑ Force Vectors
- ☑ Comment

Mechanical layers
- ☑ Mechanical Layer 1
- ☑ Mechanical Layer 2

The tab is divided into four sections:

- **PCB** — these are the working layers of your design. For more information on setting up the properties of the PCB layers, see "3.4 Setting PCB Properties" on page 3-21.
- **Assembly** — these are the layers associated with production of your board.
- **Information** — these are "virtual" layers, which provide useful design information but are not part of the physical board itself.
- **Mechanical Layers** — these are the layers to be used for documentation (e.g. showing dimensions) or other mechanical CAD-related properties. You can also set up projects and files that contain mechanical CAD information. For details, see " Using Mechanical CAD" on page 12-1.

The layer highlighted in red is the active layer — the one which any functions you choose will affect. Before you can work on a particular layer, you must ensure that layer is active. Depending on what layer is active, your available commands and toolbars may change.

> **Tip** Ultiboard presents only those functions that are appropriate for the current layer. For example, if you are on a Paste Mask layer, you cannot use the **Place** menu to place a copper area.

➢ To make a layer the active layer, double-click on the name of the layer in the **Design Toolbox**.

You can also affect the visibility of layers by making them visible, dimmed or hidden.

➢ To dim a layer in the workspace, making it easier to see the elements on other layers, click the layer's check box. The check box turns gray, and the elements of that layer dim.

➢ To hide a layer in the workspace, click the layer's check box twice. The check mark disappears, and the elements on that layer are hidden.
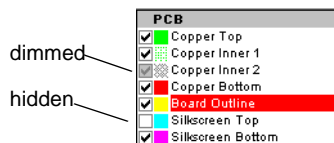
> **Note** While this action removes the elements of the layer from view, neither the layer nor the elements are deleted from the design.

> **Tip** You can show or hide individual ratsnests from the **Show Ratsnests** column in the **Nets** tab of the **Spreadsheet View**, and also from the **Net Edit** dialog box (see "7.6.2 Using the Netlist Editor" on page 7-23.

➢ To change the color of the elements in a layer, click the color box beside the layer name and, from the dialog box that appears, choose a color. All existing elements on that layer, and any elements added to that layer, are displayed in that color, with the exception of shapes where you can chose the line or fill color from the toolbar prior to placing them (see "6.3.3 Placing Shapes and Graphics" on page 6-32).

5.2 Working with the Board Outline

If you have created a new design, the board outline layer will be empty. You can create a board outline in one of the following ways:

- draw a board outline using the drawing tools (see "5.2.1 Using the Drawing Tools" on page 5-5)
- import a DXF file (see "5.2.2 Importing a DXF File" on page 5-5)
- place a predefined outline from the database (see "5.2.3 Using a Pre-Defined Outline" on page 5-6)
- use the Board Wizard (see "5.2.4 Using the Board Wizard" on page 5-7).

5.2.1 Using the Drawing Tools

➢ To create a new board outline using the drawing tools:

1. Double-click the **Board Outline** layer in the **Layers** tab of the **Design Toolbox**.

2. Using **Place/Shape**, draw the shape you want for your board outline. For example, choose **Place/Shape/Rectangle** and drag the mouse until the rectangle is the desired height and length. Click to anchor it in place.

Note To draw the shape based on precise coordinates, press the asterisk key (*) on the numeric keypad. The **Enter Coordinate** dialog box appears. Enter the x and y co-ordinates and press **OK**. Continue to use the asterisk key (*) for the remaining co-ordinates. You can mix the use of the asterisk key (*) and clicking to position each corner.

➢ To edit the properties of the placed board outline, select the outline and select **Edit/Properties**. (You must be on the **Board Outline** layer).
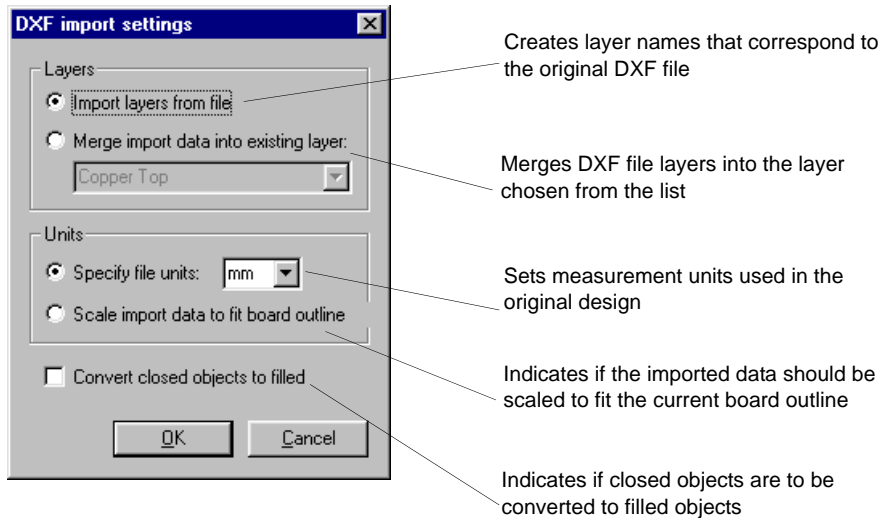
5.2.2 Importing a DXF File

➢ To import a DXF board outline from a CAD program such as AutoCAD©:

1. Choose **File/Import/DXF**. A standard file selector appears.

2. Navigate to the correct location for the .dxf file, select it and click **OK**.

3. The **DXF import settings** dialog box appears.

Creates layer names that correspond to the original DXF file

Merges DXF file layers into the layer chosen from the list

Sets measurement units used in the original design

Indicates if the imported data should be scaled to fit the current board outline

Indicates if closed objects are to be converted to filled objects

5.2.3   Using a Pre-Defined Outline

➤ To use one of the commonly-used board outlines provided in Ultiboard's database:

1. Choose **Place/From Database** to start the database manager.

2. Navigate to the **Board Outlines** section and select the outline you want to use.

3. Click **OK** to finish.

5.2.4 Using the Board Wizard

➢ To use the **Board Wizard**:

1. Choose **Tools/Board Wizard**. The first wizard dialog box appears.
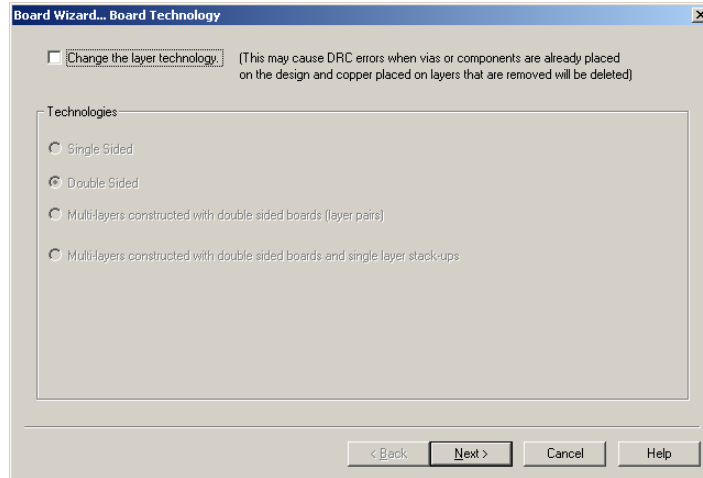
2. Enable the **Change the Layer Technology** option. Choose the board technology and click **Next**. For more information about board technology, see "5.1.1 Defining Copper Layers" on page 5-2.

   The next step of the wizard depends on which type of technology you chose.

3. If you chose **Multi-layers constructed with double sided board (layer pairs),** define the lamination settings for the board:
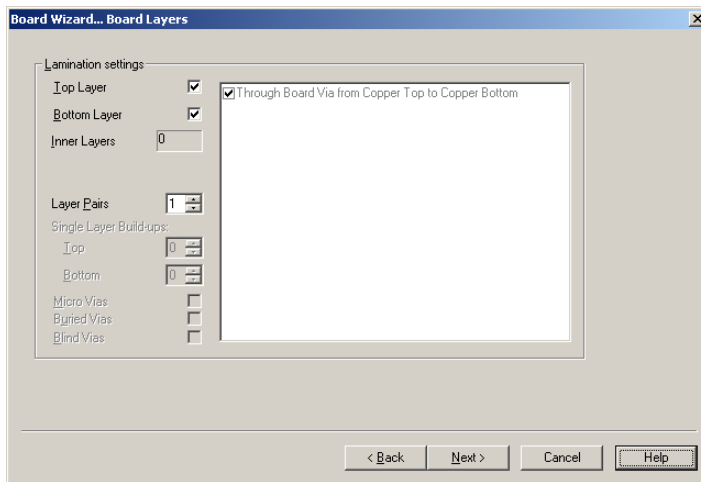

• Set the number of layer pairs you intend to use by entering the value or using the up/down arrows in the **Layer Pairs** field. There should be at least one layered pair to act as a core.

• Select the **Micro Vias**, **Buried Vias**, or **Blind Vias** checkboxes to use these in your design.

• As you make changes to the layer settings, the information window shows the acceptable layer combinations for blind and buried vias or microvias. In this window, select the layer combinations you want to allow in your design.

• Click **Next**.

4. If you chose **Multi-layers constructed with double sided boards and single layer stack-ups**, define the lamination settings for the board as follows:
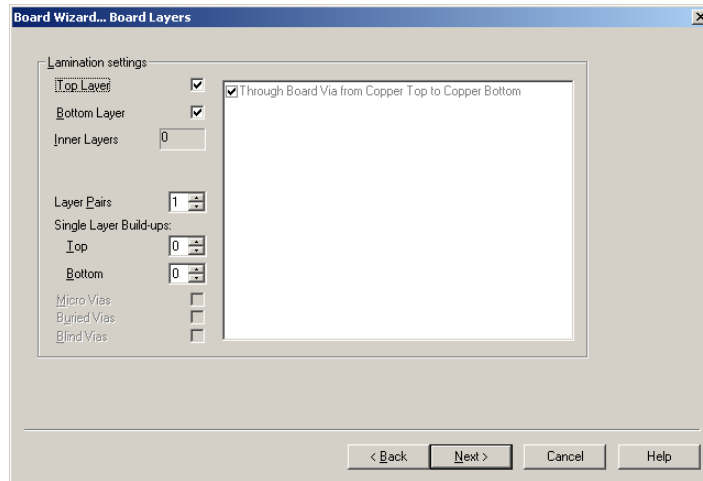
- Set the number of layer pairs you intend to use by entering the value or using the up/down arrows in the **Layer Pairs** field. There should be at least one layered pair to act as a core.
- Set the number of single layer build-ups for both the top and bottom. There should be at least one layered pair to act as a core.
- Select the **Micro Vias**, **Buried Vias**, or **Blind Vias** checkboxes to use these in your design.
- As you make changes to the layer settings, the information window shows the acceptable layer combinations for blind and buried vias or microvias. In this window, select the layer combinations you want to allow in your design.
- Click **Next**.

5. If you chose single or double-sided boards, or upon clicking **Next** after defining the lamination settings for a multi-layer board:
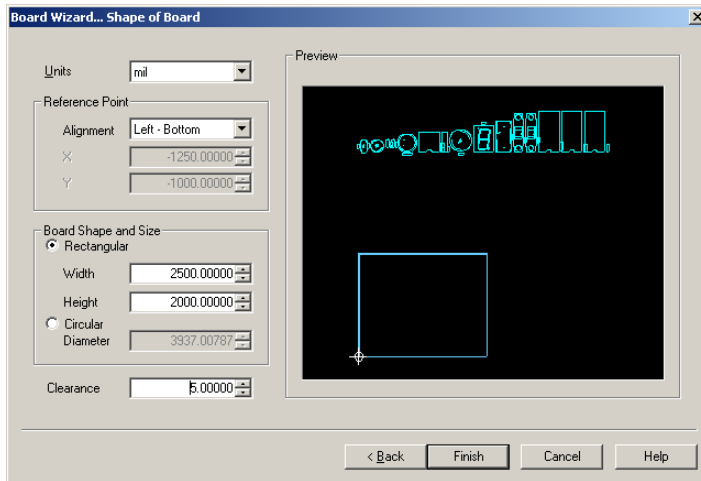
- Define the default units of measurement for the design.
- Define the board reference point. This can be changed later; see "5.3 Setting the Board's Reference Point" on page 5-11.
- Define the shape and size of the board.
- Set the default clearance for the board — the distance from the edge of the board that is to be kept free of any other elements. Trying to run a trace through a clearance, or trying to place a part so that a pad is put within a clearance, for example, results in a design rule error.
- Click **Finish**. The board outline is placed on your design.

5.3 Setting the Board's Reference Point

The reference point of the board is important for relating physical dimensions to PCB layouts, since all measurements are shown relative to the origin. If you used the **Board Wizard**, this reference may already have been set. For more details, see "5.2.4 Using the Board Wizard" on page 5-7.

The reference point looks like this:

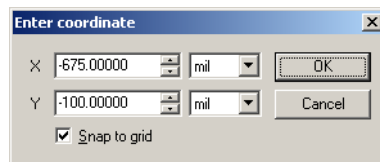➢ To set a reference point:

1. Choose **Options/Set Reference Point**.

   Your cursor changes to look like this:

2. Click to place the reference point.

   Alternatively, press the asterisk key (*) to enter set the reference point at precise X,Y coordinates. Enter the coordinates and units of measurement. To have the reference point snap to the closest grid, select **Snap to grid**.

5.4   Design Rule Errors

Design rule errors appear in the **DRC** tab of the **Spreadsheet View** as they occur, and disappear as they are corrected.
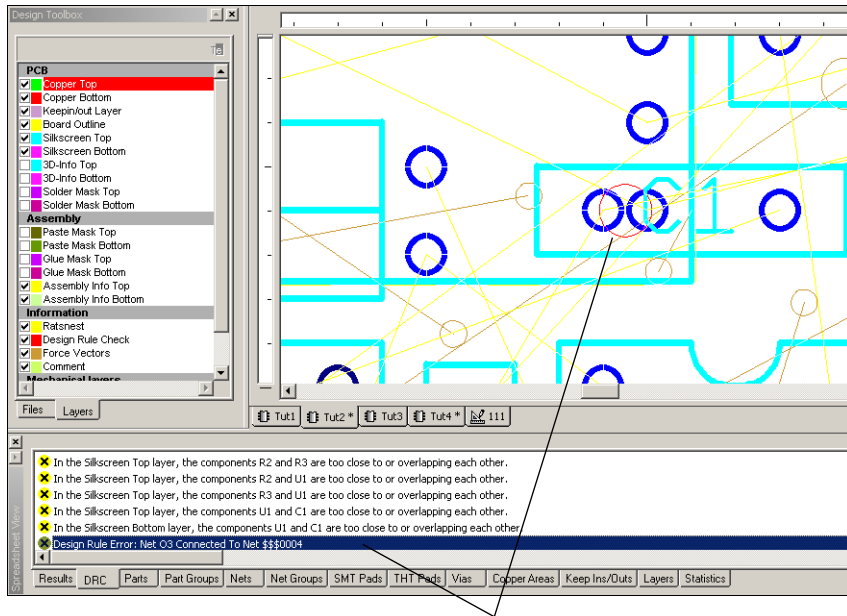
Double-click on an error in the list to zoom in on the affected area on the design (indicated with a red circle). This feature is not available in all versions of Ultiboard.

Depending on what you are doing on the board, you may see the following kinds of errors if you keep the **DRC** tab open while you work:

• *Component "[refdes]"(value) has an unknown shape (shape name)* — The given component has a shape defined that does not exist in the database

- *Component "[refdes]" is Not On the design* — The component with the given refdes was specified in the netlist but is not present on the design
- *Pin "[Pin number]" from Component "[refdes]"(value) in Net "[Net name]" is missing from shape "[shape name]"* — A pin belonging to the specified component was given in the netlist but does not exist in the shape that was given for the component.
- *Unused Pin [Pin name] is {close to, connected to} {Unused Pin, Copper}* — The given pin that was not assigned to a net is close to or connected to another unused pin or copper (which can be traces, powerplanes, copper areas, etc.)
- *Design Rule Error: Net [Net1 name] { connected to} { Net2 name, Unused pin, copper, Board outline }* — The given net was connected to another net, an unused pin, copper or the board outline.
- *Design Rule Error: Net Gnd Close to Net [Net name] [RefID:PIN#-netname]* — The given net was too close to another net.

You can set up whether the design rule check runs in "real time", and define the actions to be taken when Ultiboard encounters a design rule error, such as cancelling the current action, asking for confirmation, or overruling the error. For details, see "3.3.4 PCB Design Tab" on page 3-17.
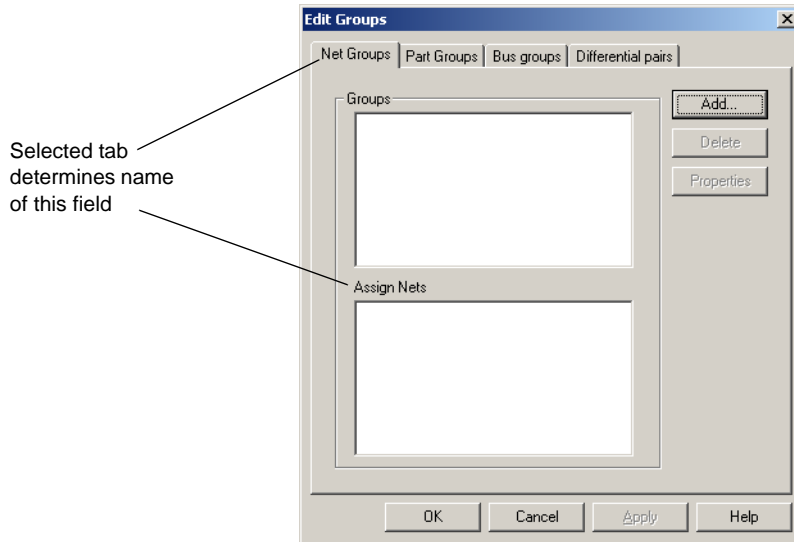
5.5　Working with the Group Editor

The group editor lets you create and edit net groups, part groups, bus groups and differential pairs.

➢ To create a group:

1. Select **Tools/Group Editor**.

Selected tab determines name of this field

2. Select one of the following tabs:

    Net Groups — groups consisting of selected nets.

    Part Groups — groups consisting of selected parts.

   **Bus Groups** — bus groups consisting of selected nets.

    Differential Pairs — groups consisting of selected pairs of nets. This feature is only present if you have Ultiroute installed with your Ultiboard installation.

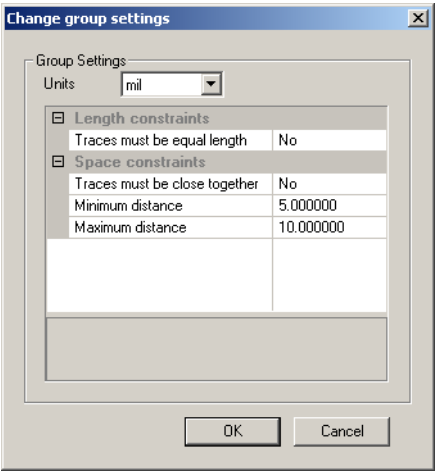3. Click **Add**. The **Add group** dialog box appears.
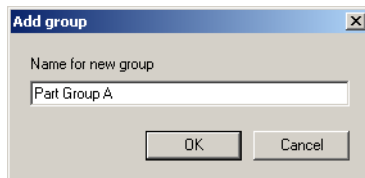
4. Enter the desired name and click **OK**.

If you are entering a **Net**, **Part** or **Differential Pairs** group, the **Change group settings** dialog box appears. Change information in the **Group Settings** area as desired and click **OK**.
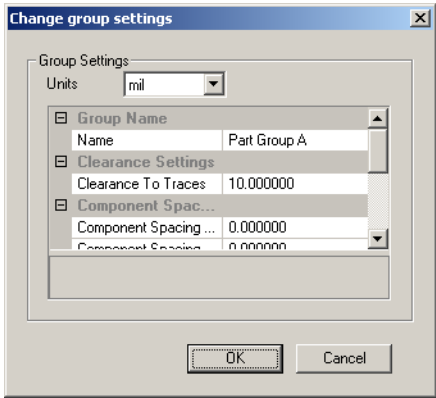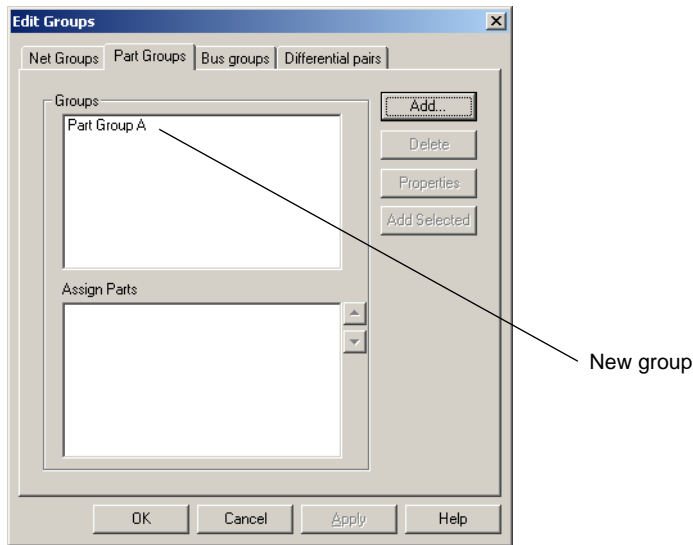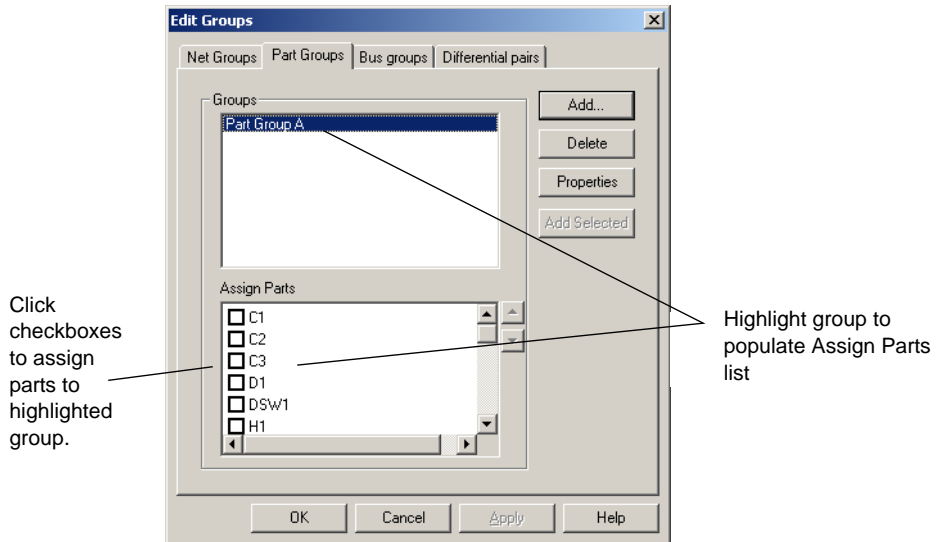

Group settings available for Net or Part groups.

Group settings available for Differential Pairs groups.

5. The **Edit Groups** dialog box appears with the new group name displayed in the **Groups** field.
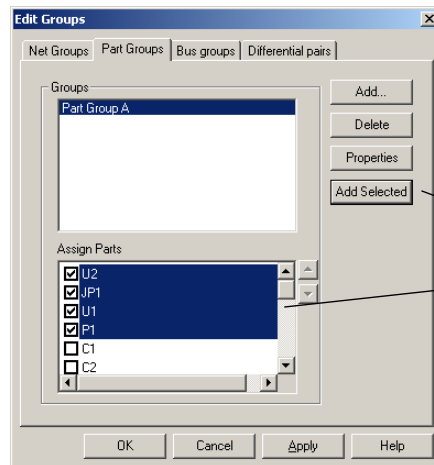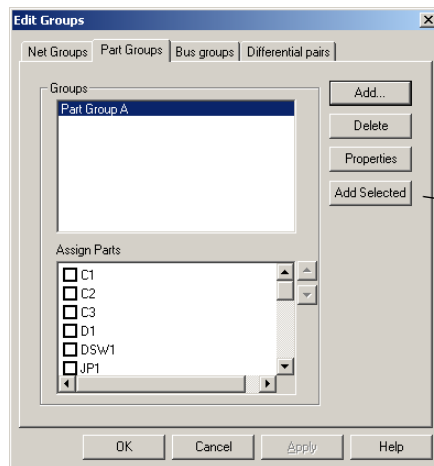
New group

6. Highlight the new group to display a list of elements you can add to the group.

Click checkboxes to assign parts to highlighted group.

Highlight group to populate Assign Parts list

7. Click checkboxes beside desired components and click **Apply** to add them to the group.

8. Click **OK** to close the **Edit Groups** dialog box.

Tip When adding components to a **Part Group**, you can select components on the workspace and then click **Add Selected**.

Button is active when one or more parts are selected on workspace.

After button is clicked, selected parts are added to the group.

➢ To edit a group:

1. Select **Tools/Group Editor** to display the **Edit Groups** dialog box.

2. Select the tab for the type of group you wish to edit.

3. Highlight the desired group in the **Groups** list.

4. Change net or part assignments as desired and click **Apply**.

5. For net or part groups you can also highlight the desired group in the **Groups** list and click the **Properties** button to display the **Change group settings** dialog box.

Make the required changes in the **Group Settings** area and click **OK**.

➤ To delete a group:

1. Select **Tools/Group Editor** to display the **Edit Groups** dialog box.

2. Select the tab for the type of group you wish to edit.

3. Highlight the desired group in the **Groups** list and click **Delete**.

Chapter   6
Working with Parts

This chapter explains how to work with parts as you create and edit designs. It covers the ways that you can place parts on the board, as well as the tools included to help you with part location and placement. It also includes information on Ultiboard's parts database, and editing the parts in the database and on the board.

The following are described in this chapter.

6.1 Placing Parts

You can place parts on the design:

- by dragging parts from outside the board outline (see "6.1.1 Dragging Components from Outside the Board Outline" on page 6-2)
- by using the **Spreadsheet View** (see "6.1.2 Using the Parts Tab in the Spreadsheet View" on page 6-3)
- by importing a netlist (see "4.4 Importing a Netlist File" on page 4-3)
- by selecting parts from the database (see "6.4 Placing Parts from the Database" on page 6-39)

Note Before placing a part, make sure that you are on the layer where the part is to be placed. For information on selecting a layer, see "5.1.2 Accessing Layers" on page 5-3.

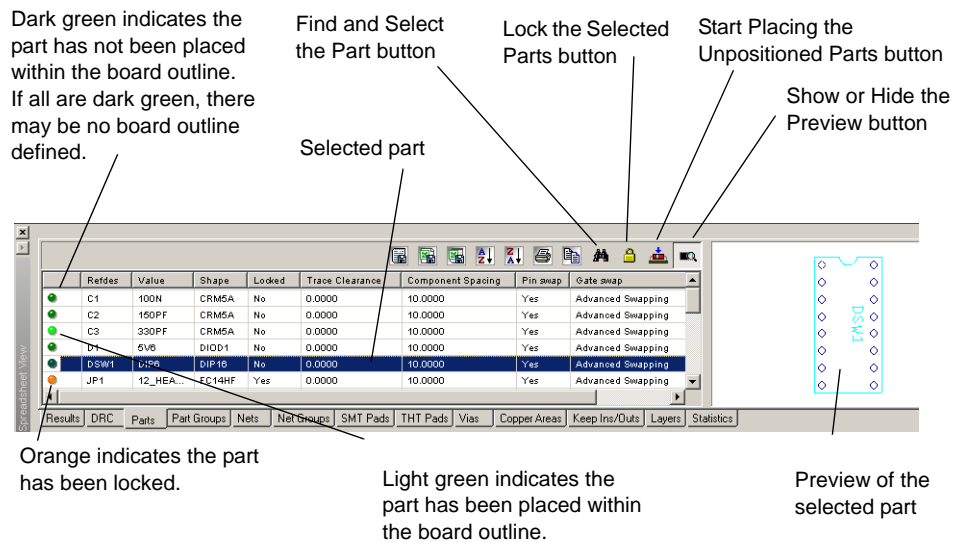# 6.1.1 Dragging Components from Outside the Board Outline

By default, components are placed outside the board outline when you open a netlist from Multisim or another schematic capture program. These can be dragged to the appropriate location on the board.

➢ To drag a component from outside the board outline:

1. Click on the component and drag it to the appropriate location.

2. The placed component remains highlighted. Click anywhere on the workspace, or right-click, to de-select the component.

6.1.2   Using the Parts Tab in the Spreadsheet View

The **Parts** tab in the **Spreadsheet View** shows a list of all the parts in your design. The color indicator beside the part indicates whether the part has been placed on the board outline (bright green), or is off to the side awaiting placement (dark green).

The **Parts** tab allows you to select parts, to lock parts so they cannot be accidentally moved, and to place parts on the board. It also contains functions to help you find parts on the board, and to preview what a part looks like.

Dark green indicates the part has not been placed within the board outline. If all are dark green, there may be no board outline defined.

Find and Select the Part button

Lock the Selected Parts button

Start Placing the Unpositioned Parts button

Show or Hide the Preview button

Selected part

Orange indicates the part has been locked.

Light green indicates the part has been placed within the board outline.

Preview of the selected part

Note   For details on the other buttons in the **Spreadsheet View**, see "3.6 Spreadsheet View" on page 3-32.

6.1.2.1   Using the Parts Tab to Place Parts

➢ To place a single part on the board:

1. Click-and-drag the part from the list to the design. The selected part will be attached to the pointer as you move it over the board.

2. Release the left mouse button to drop the part onto the board. The part can then be adjusted or moved further into position if necessary. For more information on moving parts and adjusting their placement, see "6.1.3 Tools to Assist Part Placement" on page 6-5.

➢ To place a series of parts on the board using the Parts Sequencer

1. Click an unplaced part in the list (a part with a dim button). The **Start Placing the Unpositioned Parts** button is activated on the Parts tab.

2. Click the **Start Placing the Unpositioned Parts** button.

3. Move the pointer over the board. The selected part will be attached to the pointer as you move it over the board.

4. When the part is in position, left-click to drop the part on the board. The next part in the list appears on the mouse pointer. Continue left-clicking to drop a part and pick up the next until you have placed as many as you planned to.

Note   Right-click to drop the last part or to cancel the action.

Parts that you place this way can then be adjusted or moved further into position if necessary. For more information on moving parts and adjusting their placement, see "6.1.3 Tools to Assist Part Placement" on page 6-5.

6.1.2.2   Using the Parts Tab for Other Functions

The **Parts** tab of the **Spreadsheet View** can also be used to select a part, lock parts in their current position, find and select a part, or preview a part.

➢ To select a part using the **Parts** tab:

Double-click the part in the list. The part appears selected in the design.

➢ To lock and unlock parts:

1. Click a part in the list to select it. To select multiple parts, click one part, hold the SHIFT key down, then click the last part you want to select. The two parts that you clicked, and any parts listed between them, are now selected.

2. Click the **Lock the Selected Part** button to lock all selected unlocked parts or to unlock the selected locked parts.

➢ To find a part in the design:

1. Click the part in the list.

2. Click the **Find and Select the Part** button. The view zooms in on the part, which appears selected.

➢ To preview a part:

1. Click the **Preview** button to toggle the Preview function on, if required.

2. Click the part in the list. A picture of the part displays.

You can use the **Parts** tab to place parts when the part is not on the board but is listed in the **Parts** tab. Parts listed on the tab are either placed on the board or have been imported but not yet placed within the board outline. Parts that are listed in the **Parts** tab but are not on the board are indicated with a dim button, while parts that have been placed on the board are indicated with a bright button.

Note For details on the other buttons in the **Spreadsheet View**, see "3.6 Spreadsheet View" on page 3-32.

6.1.3   Tools to Assist Part Placement

This section tells you how to use the tools that Ultiboard provides to help you place parts and other elements on the design.

### 6.1.3.1   Working with Ratsnests

A ratsnest is a straight line connection between pads, indicating their connectivity. The ratsnest identifies the pads which should be connected according to the netlist, but which are not yet connected with traces. Because these represent logical connections, and not the physical copper connections referred to as traces in Ultiboard, they are just straight line connections that can overlap components and other ratsnest lines.

In Ultiboard, ratsnests are represented by colored lines, although they can be dimmed so that they appear gray. They appear by default, exist on their own layer, and can be controlled through the **Information** section of the **Layers** tab in the **Design Toolbox**. For information on dimming and hiding ratsnests in the **Design Toolbox**, see "5.1.2 Accessing Layers" on page 5-3.

Tip You can also show or hide individual ratsnests from the **Show Ratsnests** column in the **Nets** tab of the **Spreadsheet View**, and also from the **Net Edit** dialog box (see "7.6.2 Using the Netlist Editor" on page 7-23).

The following diagram illustrates the ratsnests as they connect pads in a design without regard to running through components:
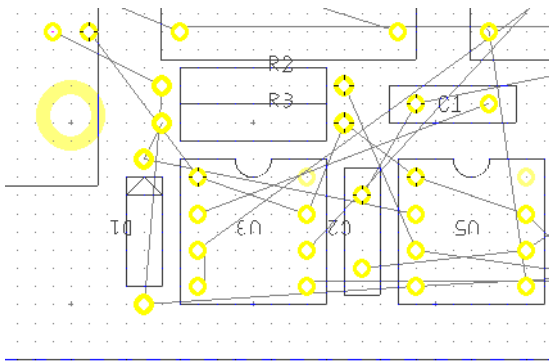

6.1.3.2 Working with Force Vectors

Force vectors are powerful aids that help you place components on the PCB. When you place components manually on the board, you should pay careful attention to the force vectors coming from that component. They allow you to place the component as close as possible to other components that are part of the same net.

You should try to minimize the ratsnest distances from that component to other pads on the board. Force vectors work by treating the force vector lines coming from each component as if they were vectors, adding them together as a vector sum, and producing a resultant force vector. The resultant force vector has a length and direction. By moving the component in the direction of the force vector, and trying to minimize the force vector length, you are moving the component to a location that results in the shortest possible combination of ratsnest lines.

Note Force vectors are extremely valuable as a guide, but you should not follow them blindly. By the nature of the algorithm, all force vectors have a natural tendency to point toward the center of the board, because all ratsnests would have their shortest connections if every component were located directly on top of each other in the very center of the board.

In Ultiboard, force vectors are represented by colored lines, although they can be dimmed. They appear by default, but can be controlled through the **Information** section of the **Layers** tab in the **Design Toolbox**. For more information on dimming and hiding force vectors, see "5.1.2 Accessing Layers" on page 5-3.

The following diagram illustrates force vectors coming from components:
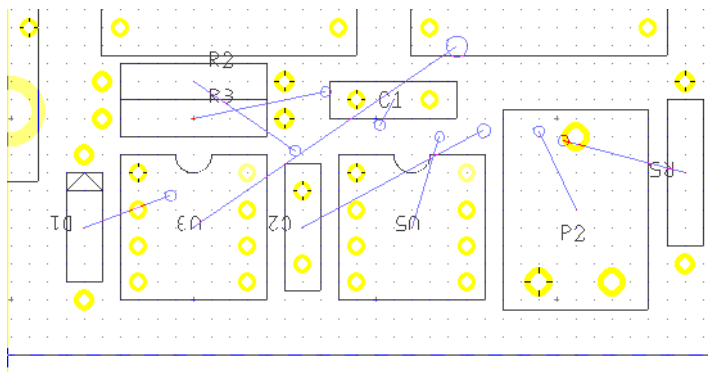
6.1.3.3 Dragging Components

Depending on your settings in the **PCB Design** tab of the **Preferences** dialog box, design rule checking may be enabled while you drag. This monitors for potential short circuits and clearance errors. If the move would cause short circuits or clearance errors, the connections are not made. If connections are lost, they can be restored by putting the component's pad back on the trace to which it belongs (same net name).

➤ To drag a component, click on the component you want to move, and drag it to the location where you want it placed.

➤ To specify the x/y coordinates to which the part is to move, press the * key on the numeric keypad or use the x/y coordinates on the status bar to get a precise reading on the location of the cursor. When you are on the exact location of the component, release the mouse button.

Rubber Banding

When you move a component that has connected traces, its connections will be maintained — this is called **Rubber Banding**. (The optional Ultiroute application must be installed).
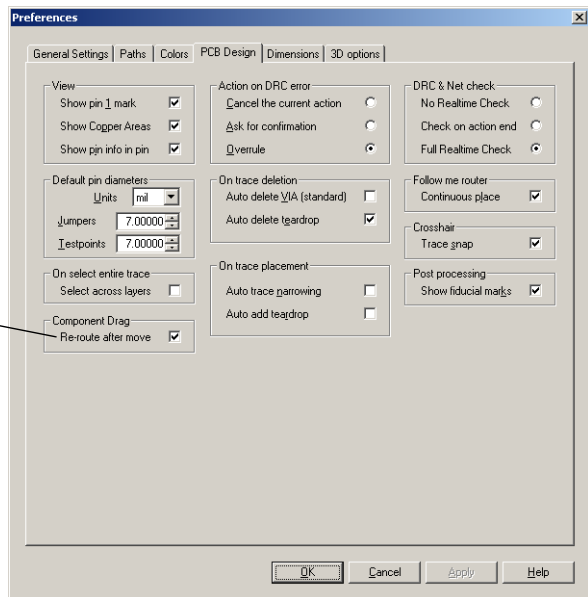
➢ To be sure that rubber banding will function when a trace is moved:

1. In the **Autorouter Settings** area of the **General** tab of the **Track Properties** dialog box, **Trace can be moved** must be selected.

Must be selected
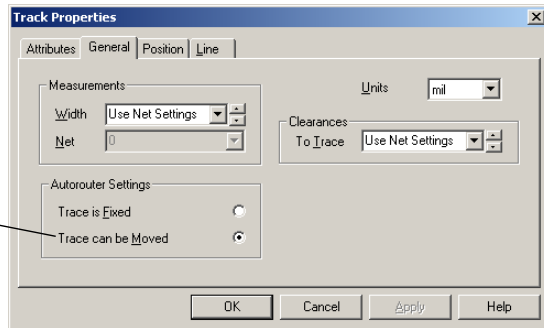
2. In the **Component Drag** area of the PCB Design tab of the Preferences dialog box, **Re-route after move** must be selected.

Must be selected

3. Unlock any locked traces by right-clicking on the trace and selecting **Unlock** from the pop-up that appears.

6.1.3.4 Shoving Components

Component shoving allows you to move one component and have Ultiboard automatically push other components on the board out of the way to create enough available space for the component. The amount of space considered for this shoving is set in the component properties, as described in "6.2 Viewing and Editing Properties" on page 6-17.

Note Shoving does not work if there is any copper connected to the component.

➢ To toggle the Shoving command on and off, choose **Options/Part Shoving**.

➢ To adjust the shove spacing around a part:

1. Select the part.

2. Choose **Edit/Properties**. The **Component Properties** dialog box for the part appears.

3. Click the **Component** tab. This tab allows you to adjust the distance of component shove spacing:

Change the spacing

Set minimum allowable space between component and traces
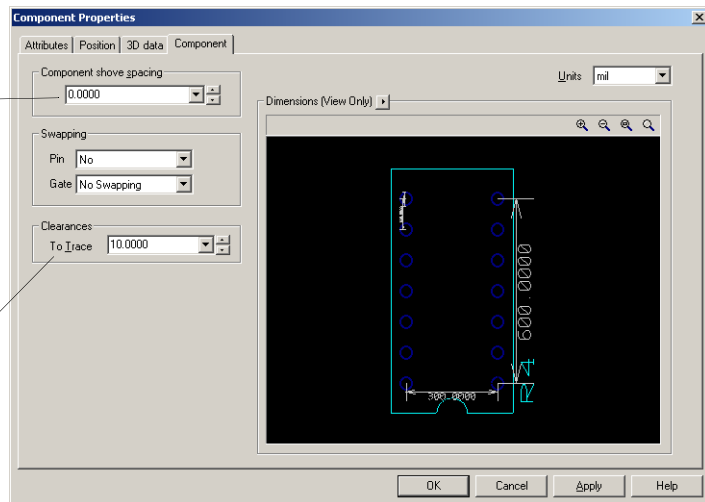
➢ To enter swapping parameters for the selected component:

1. In the **Swapping** area of the Component tab:

   • **Pin** — select **Yes**, **No** or **Use Group Settings** from the drop-down list.
   • **Gate** — select **Internal Gates only** to permit swapping of gates between gates within the same component; **No Swapping** to prohibit gate swapping for this component; **Advanced Swapping** to permit gate swapping between this component and another (both components must have **Advanced Swapping** selected); **Use Group Settings** to use group swapping options.

To assist you in setting the shove spacing and clearances, the **Dimensions (View Only)** area displays a preview of the selected component with its dimensions displayed:

➢ To change the dimensions that are displayed in the **Dimensions (View Only)** area:

1. Click on the arrow to display the pop-up menu and select/deselect the desired dimensions.

Click to display pop-up
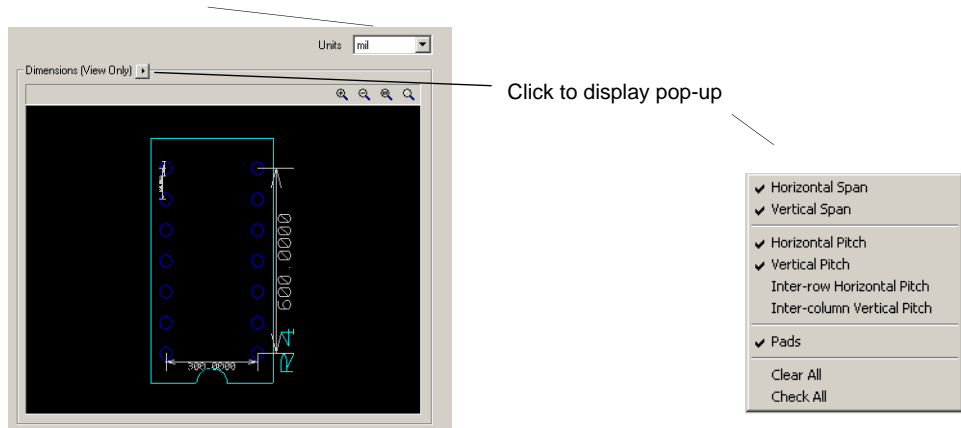
2. Your selections will be reflected in the preview area.

3. To manipulate the view of the part, click in the **Dimensions** area and use any of the following:

- **Zoom In** button — click to zoom in on the component for more detail. You can also press the F8 key.
- **Zoom Out** button — click to zoom out. Shows less detail and more of the whole component. You can also press the F9 key.
- **Zoom Window** button — click (or press F6) and then drag a rectangle around the portion of the part you want to enlarge. The area inside the rectangle enlarges to fill the **Preview** panel.
- **Zoom Full** button — click to view the entire part. You can also press CTRL + F7.
- **Mouse Wheel** — if your mouse has a center wheel, you can use it to zoom in and out on the part.
- **Scroll bars** — when the part has been enlarged beyond the borders of the **Preview** area, scroll bars appear that you can move in the usual manner to locate the desired section of a component.
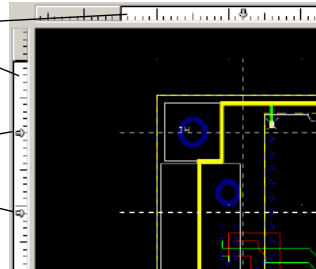
6.1.3.5 Using Ruler Bars

Use the ruler bars to place guides on the design, or to measure distances.

Click the ruler bar to set or control the guides

Guides appear like this

Dotted lines appear on the design when a guide has been set

Elements on the design will snap to the dotted lines representing the guides on the design.

➢ To toggle the ruler bars off or on, choose **View/Ruler bars**.

➢ To place a ruler guide on the workspace, click in the ruler bar at the measurement where you want the guide placed.

➢ To move a guide, click on its location on the ruler bar, and drag it to a new location.

   Note If a guide crosses elements on the design, moving the guide will also move the elements.

➢ To remove a guide, right-click on its location on the ruler bar, and choose either **Clear** (for a single guide) or **Clear All** (for multiple guides).

➢ To use a guide to measure, click on its location on the ruler bar. Measurements from the edge of the window and other guides appear, and change if you move the guide.

6.1.3.6 Orienting Components

Components are placed on the board in a certain orientation, which may not be the orientation in which you need them. You can, however, orient them by rotating them, or by swapping them to another layer.

Make sure that the part to be oriented is selected and choose the following commands from the **Edit/Orientation** menu to orient components:

Button	Menu command	Result
	Flip Horizontal	Flips the component from left to right.
	Flip Vertical	Flips the component from top to bottom.
	90 Clockwise	Rotates the component 90 degrees clockwise.

Button	Menu command	Result
	90 CounterCW	Rotates the component 90 degrees counter-clockwise.
	Angle	Rotates the component at an angle that you define.
	Swap Layer	Swaps the selected component to the mirror layer (e.g. from Silkscreen Top to Silkscreen Bottom).

6.1.3.7 Aligning Components

Shapes and components can be aligned with other shapes and components.

Select the elements to be aligned and choose the following commands from the **Edit/Align** menu to align the elements:

| Button	Menu command	Result
	Align Left	Aligns the left edges of the selected elements.
	Align Right	Aligns the right edges of the selected elements.
	Align Top	Aligns the top edges of the selected elements.
	Align Bottom	Aligns the bottom edges of the selected elements.
	Align Center Horizontal	Shifts the selected elements horizontally so their centers are aligned.
	Align Center Vertical	Shifts the selected elements vertically so their centers are aligned.

6.1.3.8 Spacing Components

Shapes and components can be spaced relative to each other on the board.

Select the elements to be spaced and choose the following commands from the **Edit/Align** menu to space the elements:

Button	Menu command	Result
	Space Across	Spaces three or more objects beside each other evenly.

Button	Menu command	Result
	Space Across Plus	Increases horizontal space between two or more objects.
	Space Across Min	Decreases horizontal space between two or more objects.
	Space Down	Spaces three or more objects above each other evenly.
	Space Down Plus	Increases vertical space between two or more objects.
	Space Down Min	Decreases vertical space between two or more objects.

6.1.3.9 Placing a Group Array Box

A group array box is used to place components in an array, such as memory chips. You create the array box first and then place the parts.

➢ To place a group array box:

  1. Select **Place/Group Array Box**.
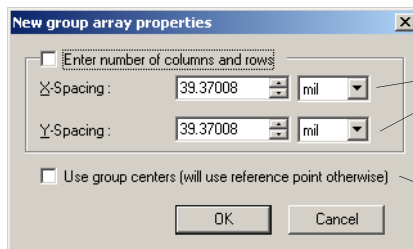
Enter desired "X" and "Y" spacing for the array.

Select if you wish to use group centers. (See diagram in step 3).

If you wish to enter the number of rows and columns instead of the x-y spacing, enable **Enter number of columns and rows**.
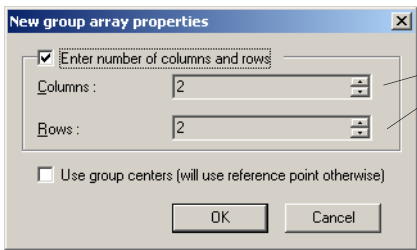
Enter desired number of columns and rows for the array.

2. Click **OK** and click and drag the mouse to place the array as desired.

Use handles to size a selected box as desired

Placed Group Array box

3. Begin selecting and dragging components. As they are placed, the components snap to the array.
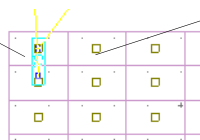
Placed component · Next component will be placed here

Use Group Centers checkbox disabled

Use Group Centers checkbox enabled

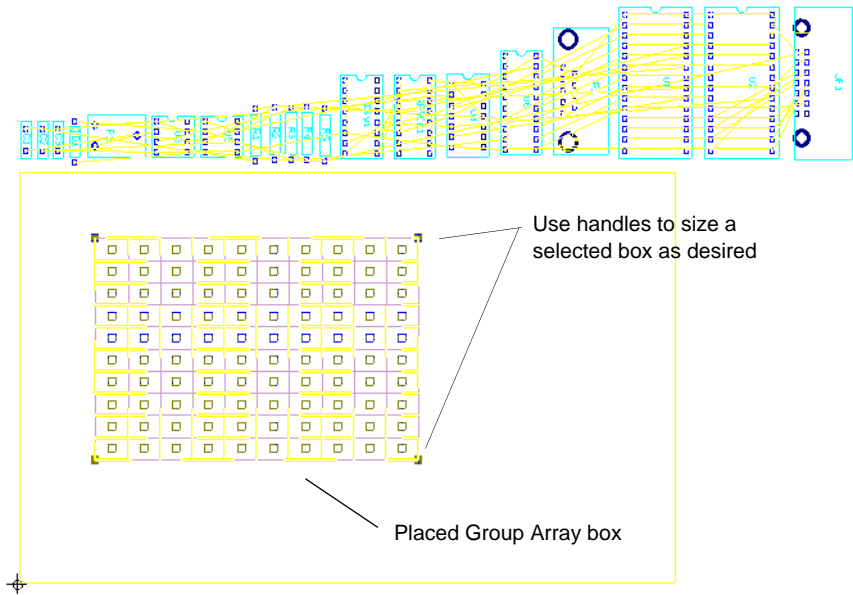Place the part in the top left cell. The other parts will be placed at the same relative position in the other cells, beginning in the top left row and working to the right.

4. Continue placing components.

6.1.3.10 Replicating a Group

The **Group Replica Place** function allows you to rapidly duplicate the placement of a group of components. This is especially useful when duplicating the layout of channels in multi-channel PCBs.

This example uses the following design:

Group 1 consists of U1, U2 & U3

Group 2 consists of U4, U5 & U6
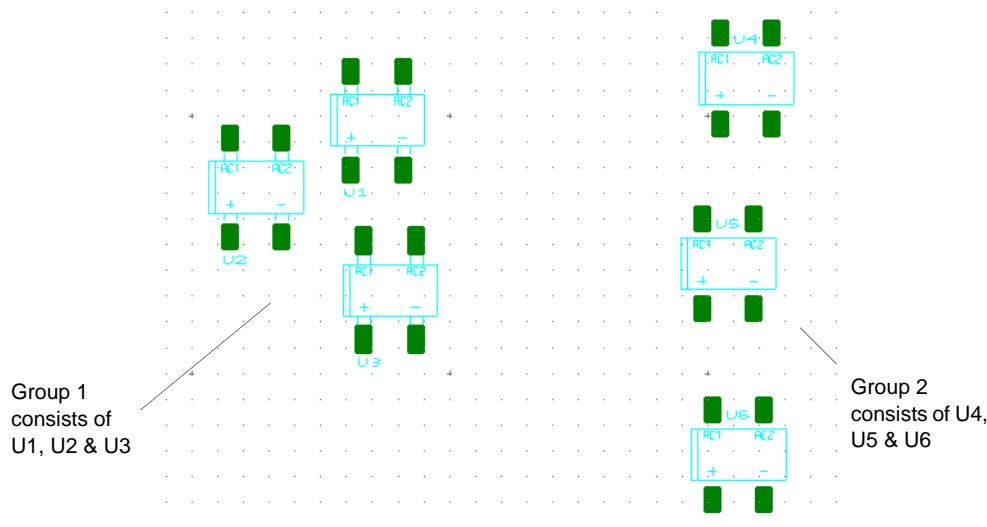
Note For instructions on group creation, see "5.5 Working with the Group Editor" on page 5-14.

➢ To replicate the positioning of a group of components:

1. Select **Design/Group Replica Place** to display the following dialog box.
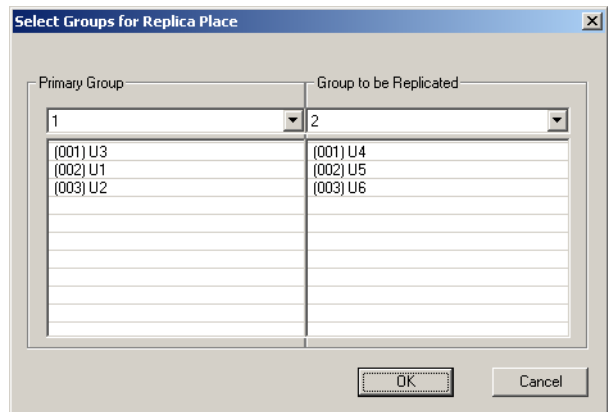
The Group to be Replicated's positioning will be changed to match that of the Primary Group

2. In the **Primary Group** drop-down list, select the group whose *positioning* is to be *copied*.

3. In the **Group to be Replicated** drop-down list select the group whose positioning you wish to change to match the **Primary Group**.

4. Click **OK**. The dialog closes and the **Group to be Replicated** (in this example, group 2) is placed on your cursor in the shape of the **Primary Group**.

Primary Group (group 1)

Shape of Group to be Replicated (group 2) changes to match Primary Group's shape.

5. Drag your mouse to the desired location and click to place the group.

6.1.4 Unplacing Parts
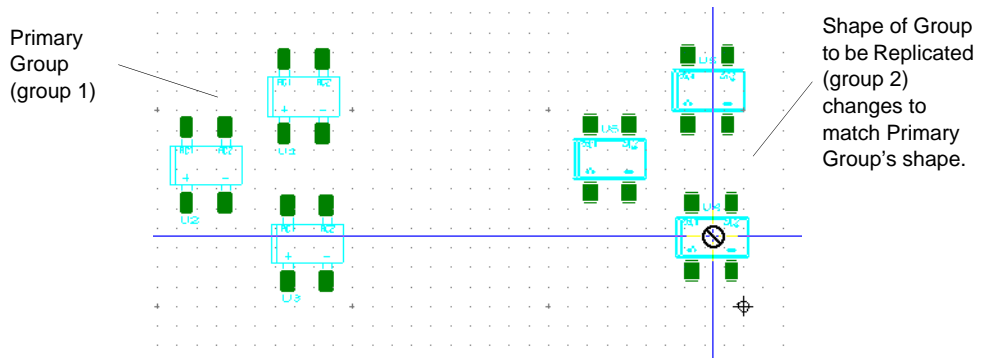
➢ To unplace all non-locked components:

1. Select **Place/Unplace Components**. The following dialog displays.
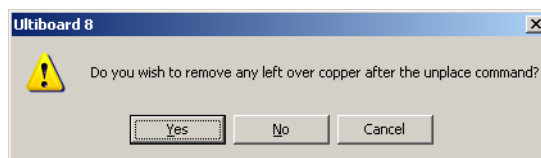
2. Select either **Yes** or **No**. All unlocked parts are removed from the PCB and positioned outside of the board outline.

If you chose **Yes** in the above dialog, the copper that was connected to the unplaced components is also removed. If you chose **No**, the copper remains in place.

6.2 Viewing and Editing Properties

Once a shape or a part has been created, its properties can be viewed and edited. This section explains how to display the property dialog boxes of components, attributes, and shapes, and what each tells you about the element.

6.2.1   Attributes

All properties dialog boxes for all elements have an **Attributes** tab. Parts are, in general, the only elements that already have attributes, typically including a name, a value, and a shape. For other elements, you can add attributes. All attributes can be changed or added.

In the **Attributes** tab, you can change an attribute's:

*   tag (the name for the attribute)
*   value (the value of the attribute)
*   visibility (invisible, value shows, value and tag show).

You make these changes by clicking on a row to make it active and entering new information or, for visibility, choosing from the drop-down list.
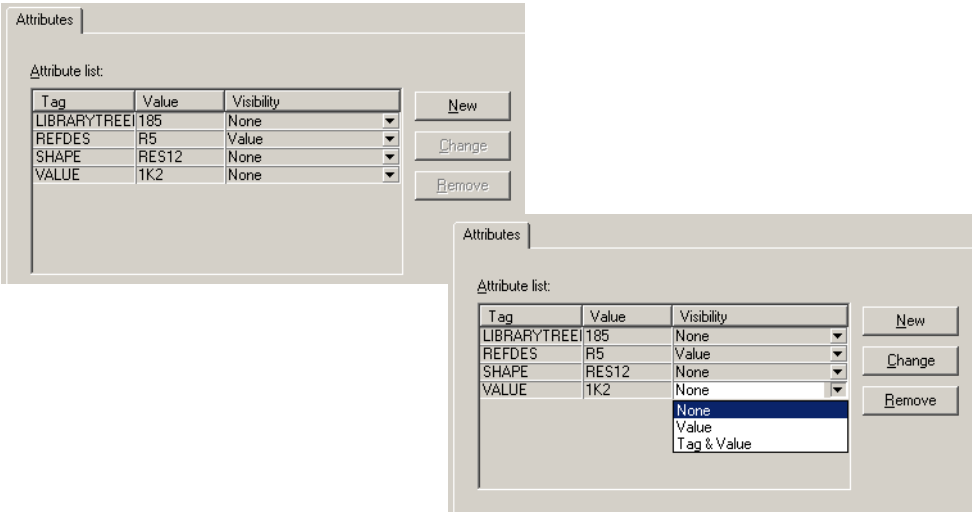
Note   You can sort attribute information by clicking on the column header.

You can also make these changes through the **Attribute** dialog box, described below.

➢ To delete an attribute, select it and click **Remove**.

Variant Attributes

If you are looking at the attributes of a component that was imported from Multisim, and that component has variants assigned, the tab will also have a variant attribute as shown below.

Variant attribute

Variant's name from Multisim
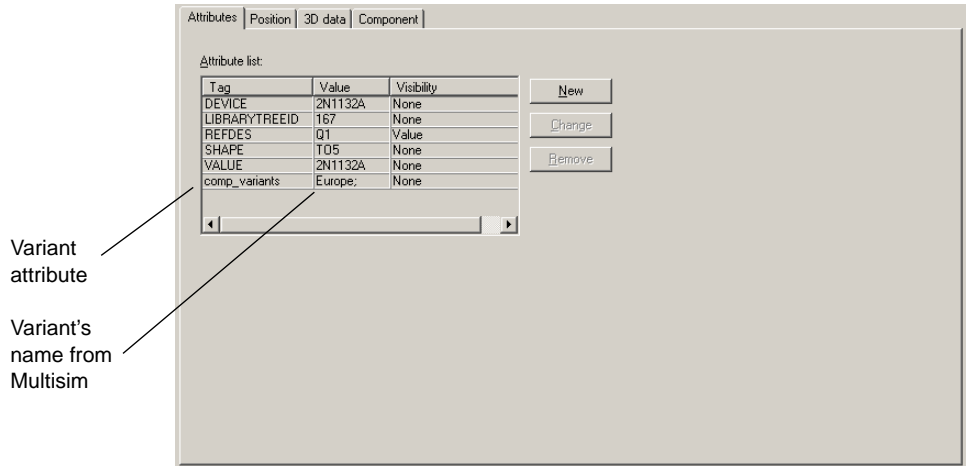
Note   For complete information on variants, refer to the *Multisim 9 User Guide*.

➢ To add or modify an attribute:

1. Select the attribute you want to modify and click **Change**.

   Or

   Click **New**.

   The **Select layer for attribute** dialog box appears.

2. Select the desired layer and click **OK**.

3. The **Attribute** dialog box appears.

4. Do some or all of the following:
   - Change or set the attribute's tag by choosing from the **Tag** list.
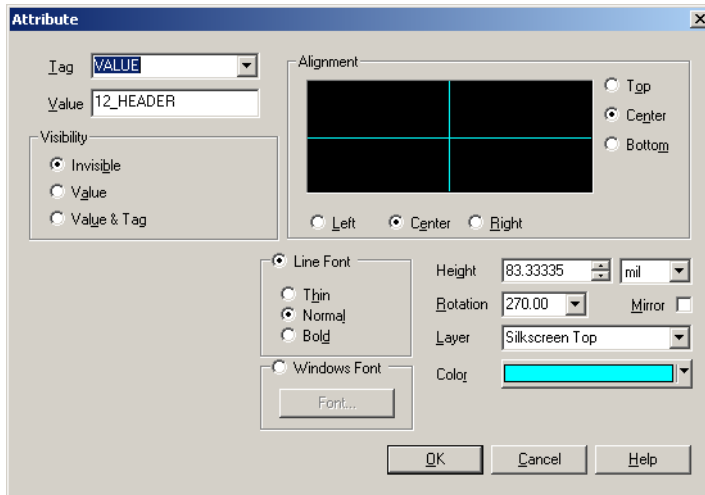   - Change or set the attribute's value by entering it in the **Value** field.
   - Choose the attribute's **Visibility** by enabling the desired option. Visible attributes are previewed in the **Alignment** area of the dialog box.
   - Use the **Alignment** area to set the attribute's alignment (when visible) relative to the component footprint.
   - Use the **Line Font** area of this dialog box to specify the weight of font used to display the attribute (when visible). Your choices are reflected in the preview area.
   - Set the height and rotation of the attribute (when visible). Your choices are reflected in the preview area.
   - Choose the color of the attribute when visible on a specific layer. Choose the layer from the **Layer** drop-down list, the color by clicking **Color**.
5. To save your changes, click **OK**. To cancel them, click **Cancel**.

6.2.2   Viewing and Editing Component Properties

➢ To view and edit the properties of a component:
   1. Select the part.
   2. Choose **Edit/Properties**. The part's **Component Properties** dialog box appears.

The **Position** tab is the default, and appears when you choose **Edit/Properties**. It displays the coordinates of the selected part:

Locks the part in place

Select the units of measurement

Change the X-axis coordinate

Change the Y-axis coordinate

Select the angle

Select the board side (top or bottom)
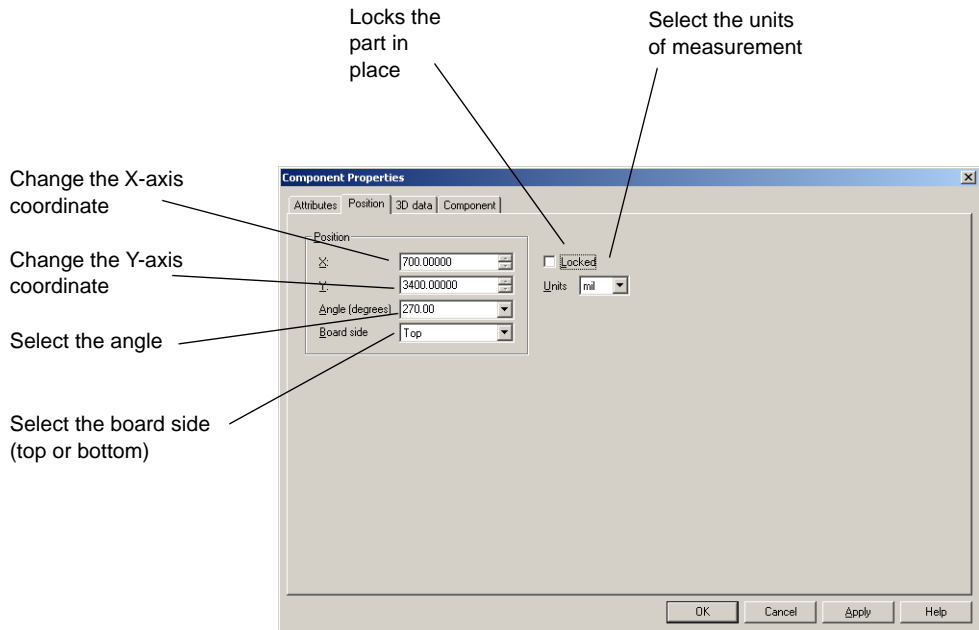
The **Attributes** tab appears when you choose **Edit/Properties**. It allows you to edit the properties of the selected part:

Click a column header to sort the column

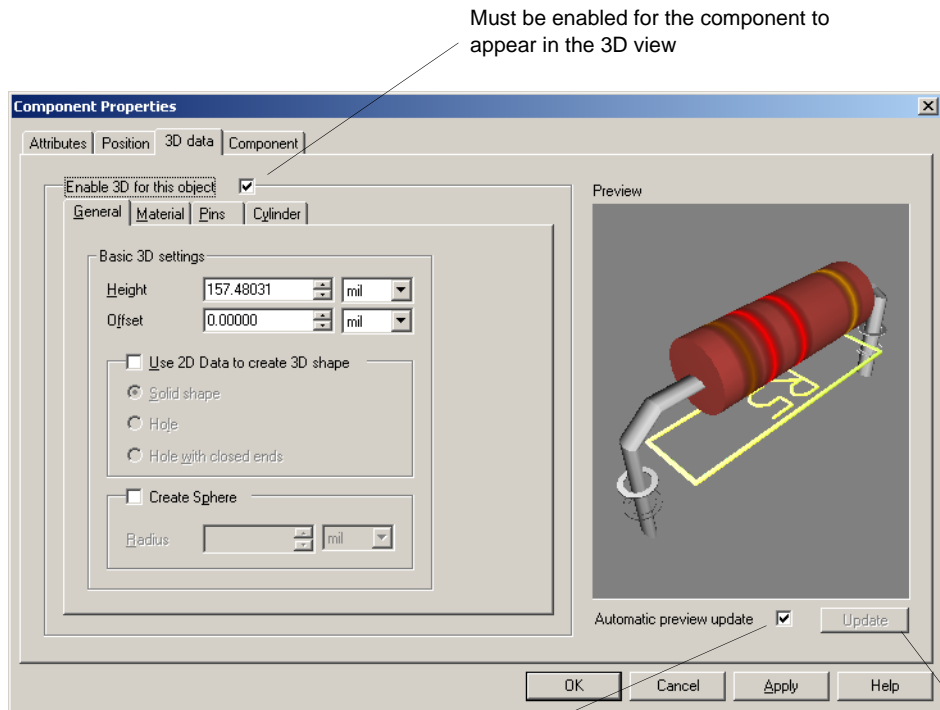Click to select Visibility from a list
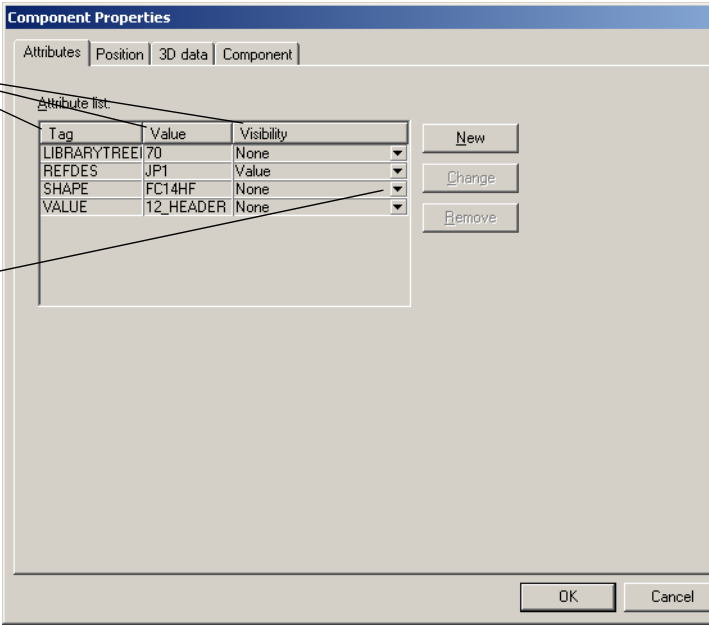
For more information on editing properties in the **Attributes** tab, see "6.2.1 Attributes" on page 6-18.

The **3D data** tab allows you to control the properties of the 3D image of the selected part. Any changes to the 3D properties are reflected in the **Preview** area.

Must be enabled for the component to appear in the 3D view

Enable checkbox to have the preview update as changes are made in the General, Material, Pins or Cylinder tab

For slower computers, disable checkbox and click Update button when you wish to refresh the preview.

General Tab

➢ To specify the distance between the component and the board:

1. In the **Height** field, enter the distance to the top of the component from the surface of the board.

2. In the **Offset** field, enter the distance from the component underside to the board.

➢ To produce the 3D image by projecting the silkscreen information upwards according to the height and offset specified for the component:

1. Enable the **Use 2D Data to create 3D shape** option.

2. Choose the type of object to be created:

- for a hole, enable the **Hole** option (the hole extends from the Offset to the Height) — If you want to make the selected part a hole in the 3D view, the part must be completely enclosed (e.g. a circle or rectangle) and contained within another larger object. For example, this could be used to create a notch or dimple in a DIP (dual-in-line package.) These settings should be applied by first editing the component (using either **In-place Edit** or the **Database Manager**) and choosing the object to which the settings apply.
- for a sphere, enable the **Create Sphere** option and enter a value in the **Radius** field.

Material Tab
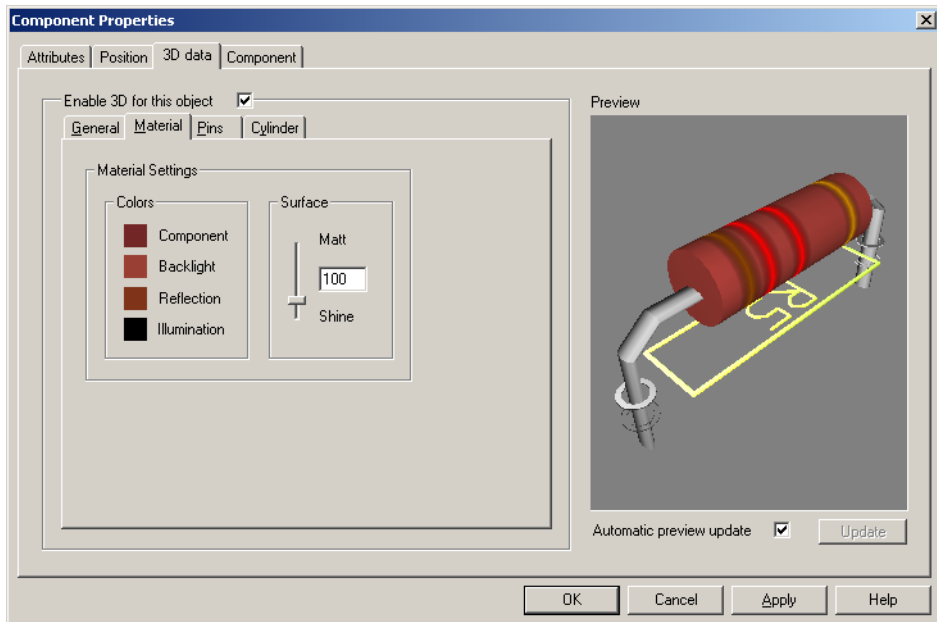
➤ To choose the colors to display for the component, click on the color box beside each of the following field labels, and choose a color from the dialog box that appears:

- **Component** — the color used when the component's face is viewed at right angles to a line from the viewer to the component.
- **Backlight** — the color used on any component face for which the light source is not directly incident.
- **Reflection** — the color used for reflected light. This works in conjunction with the **Surface** setting. The more shine, the more the component will display reflected light.
- **Illumination** — the color used when the component emits light, for example, a light emitting diode.

Pins Tab
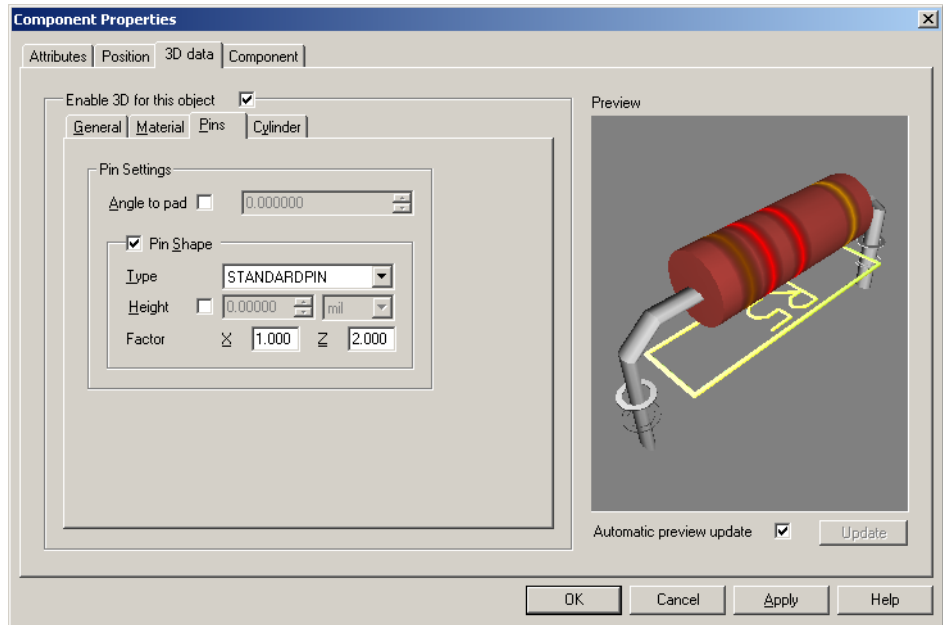
> To choose the pin model to be used in the 3D rendering, choose from the **Type** list. By default the pin will attach at the mid-point of the body of the component. Enable the **Height** option and enter a height value to use a value other than the default.

Cylinder Tab
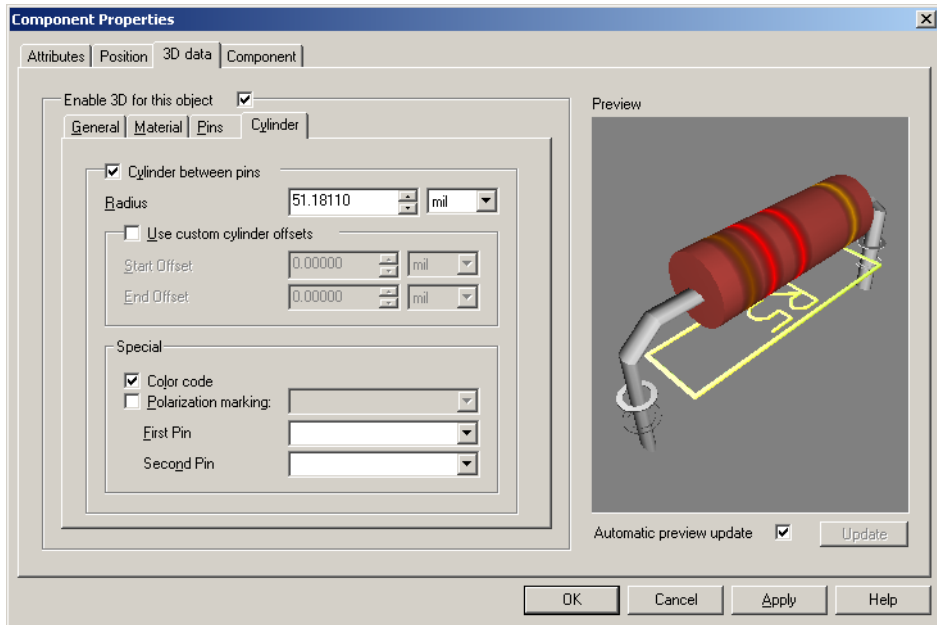
➤ To model a component in a cylindrical package such as a resistor or diode:

1. Enable the **Cylinder between pins** option.

2. If the component is a resistor, enable the **Color code** option.

3. To place a band to indicate polarization (e.g., for a diode), enable the **Polarization marking** option and choose the pin to mark.

4. To set an offset for the cylinder, enable **Use custom cylinder** offsets and enter the **Start Offset** and **End Offset** values.

The **Component** tab allows you to adjust the distance of component shove spacing and also enter pin and gate swapping settings.
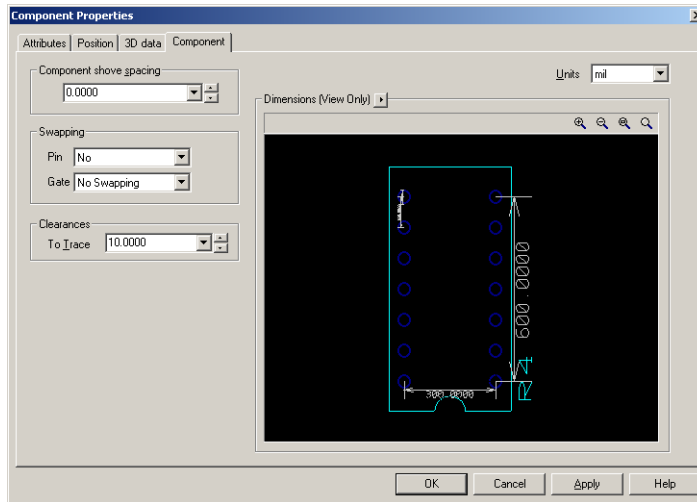
For details, see "6.1.3.4 Shoving Components" on page 6-9.

6.2.3 Viewing and Editing Attributes

The attributes associated with a component that are visible on the workspace (typically this is the component's Reference Designator) also have properties. These are made up of three tabs: **General**, **Position** and **Attribute**.

➢ To view an attribute's properties:

1. Select the attribute (for example, the Reference Designator).

2. Select **Edit/Properties**. The **Attribute Properties** dialog box appears.

➢ To edit an attribute's display style, use the **General** tab.

Set the line style for the attribute's text line

Set the attribute's color

Set the width of the attribute's text line and the units of measurement

Attribute Properties

General | Position | Attribute

Line

Style

Color

Width 5.91665   mil

OK   Cancel   Apply   Help
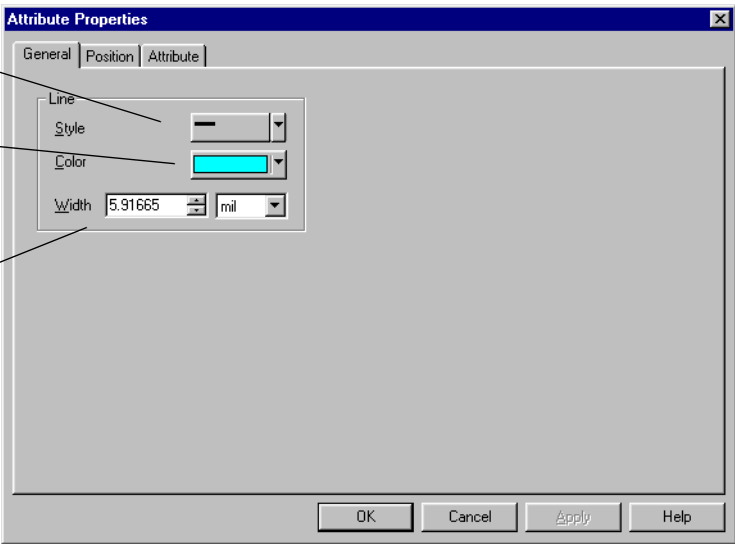
➢ To edit an attribute's coordinates, angle and layer, use the **Position** tab.

Locks the attribute in place

Select the units of measurement

Change the X-axis coordinate

Change the Y-axis coordinate

Select the angle

Select the layer on which the attribute appears

Attribute Properties

General | Position | Attribute

Position

X: -8.33331

Y: 763.74996

Angle (degrees) 90.00

Layer   Silkscreen Top

Locked

Units mil

OK   Cancel   Apply   Help
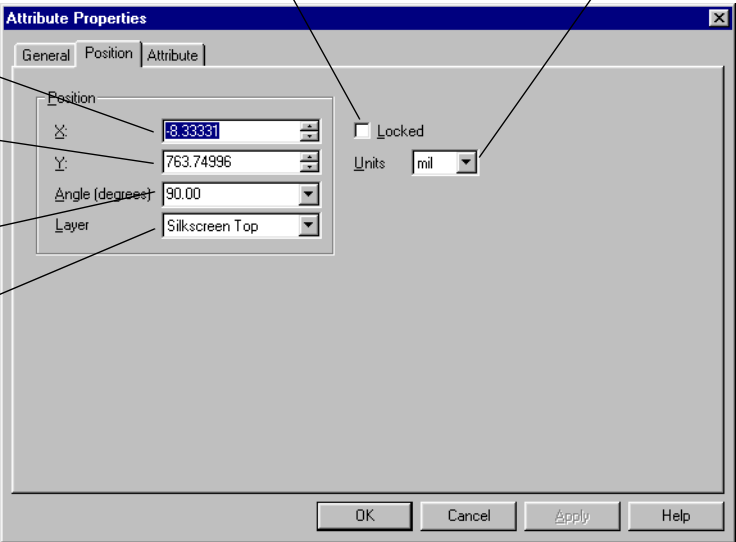
➢ To edit an attribute's visibility, alignment, and so on, use the **Attribute** tab. For information on the contents of this tab, see "6.2.1 Attributes" on page 6-18.

6.2.4　Viewing and Editing Shape/Graphics Properties

As with components and traces, the properties of shapes can be viewed and edited.

➢ To edit the properties of a shape that you have placed on the design:

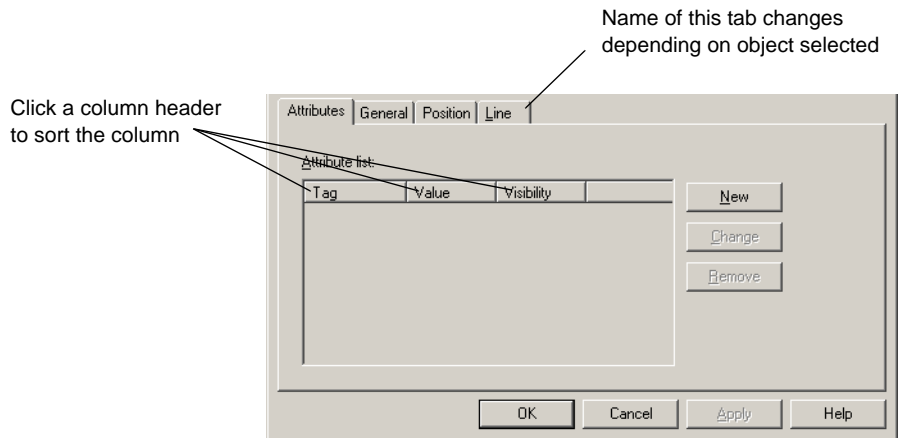1. Select the shape.

2. Choose **Edit/Properties**.

    Or

   Right-click, and select **Properties** from the context menu that appears.

To edit a shape's attributes, use the **Attributes** tab:

For more information on editing properties in the **Attributes** tab, see "6.2.1 Attributes" on page 6-18.

Depending on the shape/graphic selected, the **General** tab lets you change the width and the clearance of the selected element and define the units of measurement; or change the line style, color and width.

To change the layer on which a shape exists, use the **Position** tab:

Select the
layer

Use to lock
the shape to
the layer
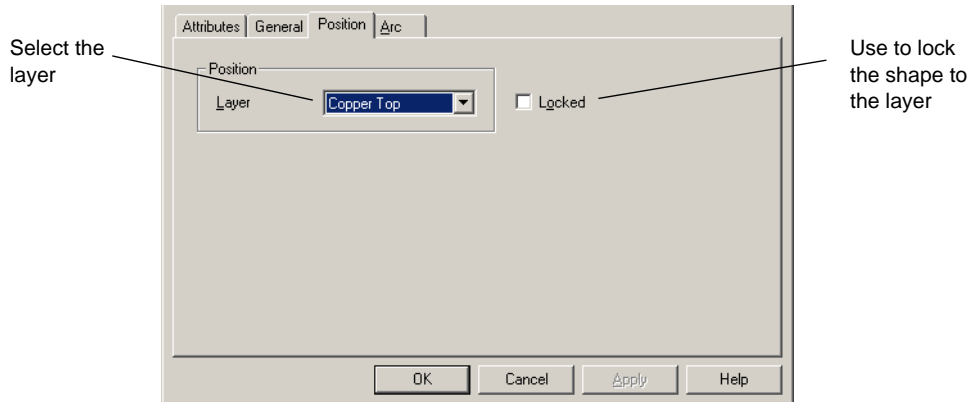
The fourth and final tab is used to change the selected shape's size. The name and contents of this tab change depending on the shape selected.
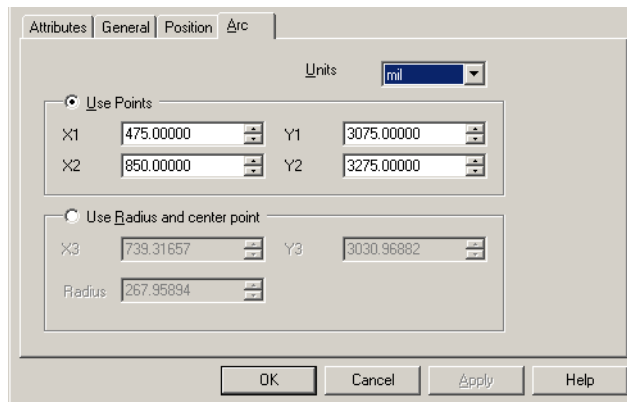
6.3 Placing Other Elements

This section explains about placing the following elements:

❒ "6.3.1 Placing Mounting Holes and Connectors" on page 6-31

❒ "6.3.2 Placing Holes" on page 6-31

6.3.1 Placing Mounting Holes and Connectors

Mounting holes and connectors are added from the database. For details, see "6.4 Placing Parts from the Database" on page 6-39.

6.3.2 Placing Holes

You can also place holes directly onto the workspace, without using the database

1. Select **Place/Hole**. The **Through Hole Pin Properties** dialog box appears.
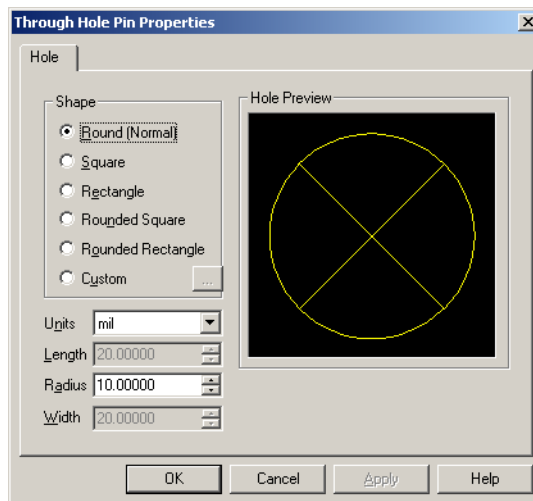
2. Enter the desired parameters for the hole and click **OK**. The dialog closes and the hole is placed on the cursor.

3. Move the cursor to the desired location and click to place it on the workspace.

4. You can continue clicking to place more instances of the same hole, or right-click to cancel placement.

6.3.3   Placing Shapes and Graphics

Ultiboard allows you to place various shapes and graphics on your design, and to move them once they have been placed. Depending on your active layer, the set of available shapes and graphics for placement, and what they represent, may differ.

Choose the following commands to place shapes and graphics:

Button	Command	Description
	Place/Line	Left-click two points to draw a line between them. Continue clicking to draw another segment of the same line, or right-click to stop.
	Place/Shape/ Ellipse	Left-click two points that define the ellipse's focuses, then move the pointer to define the ellipse's size.
	Place/Shape/ Rounded Rectangle	Left-click to define the opposite corners of the rectangle, then move the pointer towards the middle of the rectangle to define the roundness of the corners.
	Place/Shape/ Circle	Left-click two points that define the circle's diameter.
	Place/Shape/ Pie	Left-click to define two points that define the diameter of the pie, then move the pointer back and forth to define whether the shape looks like a single slice of pie or like a pie with a slice missing.
	Place/Shape/ Rectangle	Left-click to define the opposite corners of the rectangle.
	Place/Shape/ Polygon	Left-click all points that are to define the polygon, ending with the starting point.
	Place/Graphics/ Line	Left-click two points to draw a line between them. Continue clicking to draw another segment of the same line, or right-click to stop.
	Place/Graphics/ Arc	Left-click two points to draw an arc between them, then move the pointer to change the degree of arc.
	Place/Graphics/ Circle	Left-click two points that define the circle's diameter.
	Place/Graphics/ Bezier	Left-click two points to draw a bezier curve between them, then move the pointer to change the degree of arc.

After creating a shape/graphic, right-click to cancel the **Place** command.

Note   Shapes and graphics can be moved, oriented, and aligned like components, and their properties can also be viewed and edited. Use the properties to change the line color, style, and width, and fill color and style if applicable. For details, see "6.2.4 Viewing and Editing Shape/Graphics Properties" on page 6-29.

6.3.4 Working with Jumpers

6.3.4.1 Placing Jumpers

Default jumper pin settings are defined on the **PCB Design** tab of the **Preferences** dialog box. The default pad settings are based on the settings defined in the **Pads/Vias** tab of the **PCB Properties** dialog box. They can be manually set to use the annular ring specification or pad diameter settings by setting the properties of the pad when the jumper has been placed on the design.

➢ To place a jumper:

1. Be sure a copper layer is selected.

2. Choose **Place/Jumper**.

3. Move the pointer over the design. The pointer has the first prong of the jumper attached.

4. Click to drop the first prong of the jumper, then move the pointer to where the second prong should be placed.

5. Click to drop the second prong of the jumper.

6. Click to drop the first prong of another jumper, or right-click to cancel the **Place/Jumper** command.
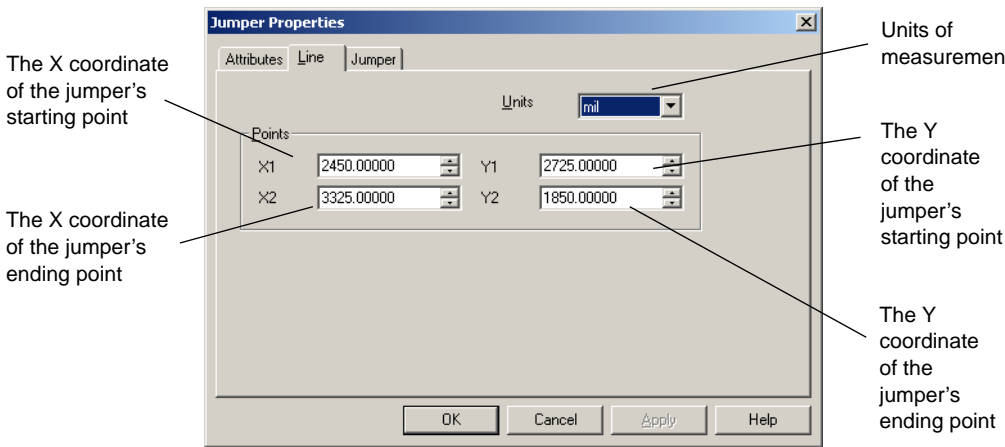
Once both prongs have been placed, jumpers can be moved, oriented, and aligned like parts.
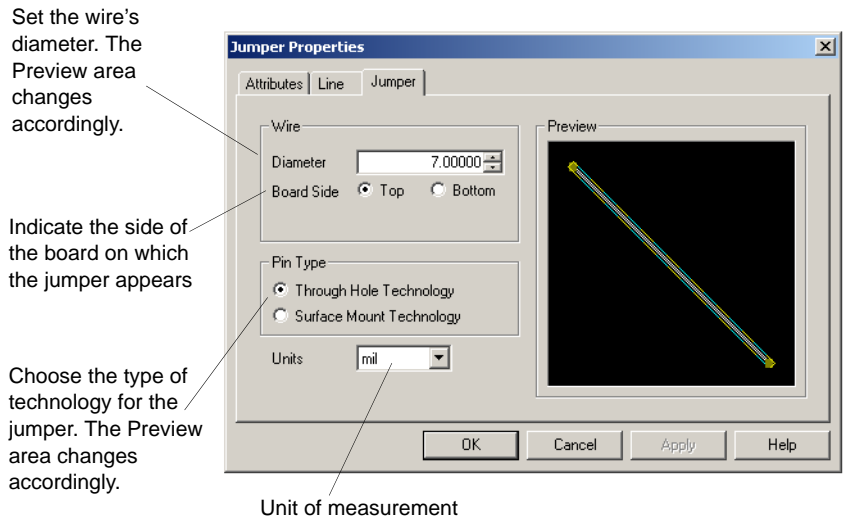
6.3.4.2 Viewing and Editing Jumper Properties

Jumper properties consist of three tabs: **Attributes**, **Line** and **Jumper**.

To edit a jumper's attributes, use the **Attributes** tab. For more information on editing properties in the **Attributes** tab, see "6.2.1 Attributes" on page 6-18.

To control the coordinates for the jumper's starting and ending points, use the **Line** tab:

The X coordinate of the jumper's starting point

The X coordinate of the jumper's ending point

Units of measuremen

The Y coordinate of the jumper's starting point

The Y coordinate of the jumper's ending point

To control the jumper's wire and pin type, use the **Jumper** tab:

Set the wire's diameter. The Preview area changes accordingly.

Indicate the side of the board on which the jumper appears

Choose the type of technology for the jumper. The Preview area changes accordingly.

Unit of measurement

6.3.5 Working with Test Points

6.3.5.1 Placing Test Points

◈ Default test point pin settings are defined in the **PCB Design** tab of the **Preferences** dialog box. The default pad settings are based on the settings defined in the **Pads/Vias** tab of the **PCB Properties** dialog box. They can be manually set to use the annular ring specification or pad diameter settings by setting the properties of the pad when the test point has been placed on the design

➢ To place a test point:

1. Be sure a copper layer is selected.

2. Choose **Place/Test Point**.

3. Move the pointer over the design. The pointer now has a test point attached.

4. Click to drop the test point on the design.

Test points can be moved, oriented, and aligned like parts. For more information on moving parts, see "6.1.3 Tools to Assist Part Placement" on page 6-5.

6.3.5.2 Viewing and Editing Test Point Properties

➢ To edit a test point's properties:

1. Select the test point (on the silkscreen layer) and select **Edit/Properties**. The **Testpoint Properties** dialog appears.
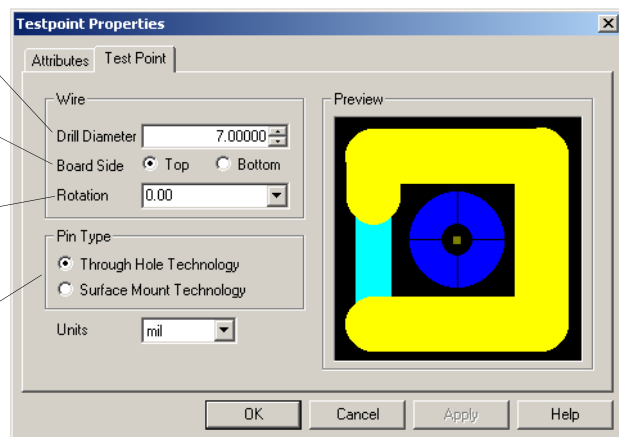
2. Click the **Test Point** tab and set the paramaters as desired and click **OK**.

Set the drill's diameter. The Preview area changes accordingly.

Indicate on which side of the board the test point appears.

Set the rotation for the test point. The Preview area changes accordingly.

Choose the type of technology for the test point. The Preview area changes accordingly.

Note To edit a test point's attributes, use the **Attributes** tab. For more information on editing properties in the **Attributes** tab, see "6.2.1 Attributes" on page 6-18.

6.3.6   Working with Dimensions

6.3.6.1 Placing Dimensions

Dimensions can be placed on a silkscreen (top or bottom) layer.

➢ To set Dimensions parameters, including arrow style, text style, position, orientation, and alignment, choose **Options/Global Preferences** and select the **Dimensions** tab.

➢ To place a dimension on the board:

1. Be sure you have selected the silkscreen (top or bottom) layer.

2. Choose **Place/Dimension Lines** and choose the type of dimension to be placed:

   **Standard**: If the dimension is to be placed at an angle.

   Horizontal: If the dimension is to be placed horizontally.

   Vertical: If the dimension is to be placed vertically.

3. Click to define the starting point of the dimension.

4. Move the pointer to the end point of the dimension. Ultiboard measures as you move the pointer.

5. Click to indicate the end point. Ultiboard stops measuring the length, and draws an arrow between your start and end points.

6. Move the pointer to position the stub line, and click when you're done. For example:
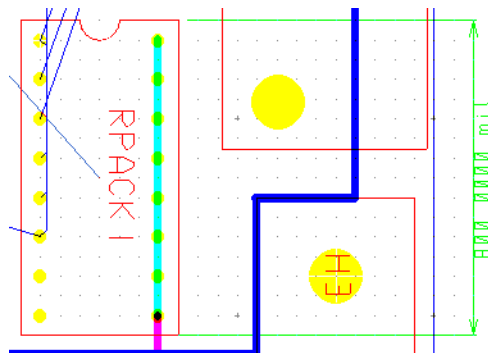
The vertical dimension of part RPACK1 has been moved to the outside of the board outline.

6.3.6.2 Viewing and Editing Dimension Properties

Dimension properties consist of five tabs: **Attributes**, **General**, **Position**, **Line** and **Dimension**.

➢ To edit a dimensions's properties, select the dimension and select **Edit/Properties**.

➢ To edit a dimension's attributes, use the **Attributes** tab. For more information on editing properties in the **Attributes** tab, see "6.2.1 Attributes" on page 6-18.

➢ To edit a dimension's display style, use the **General** tab.

Set the line style
for the
dimension's
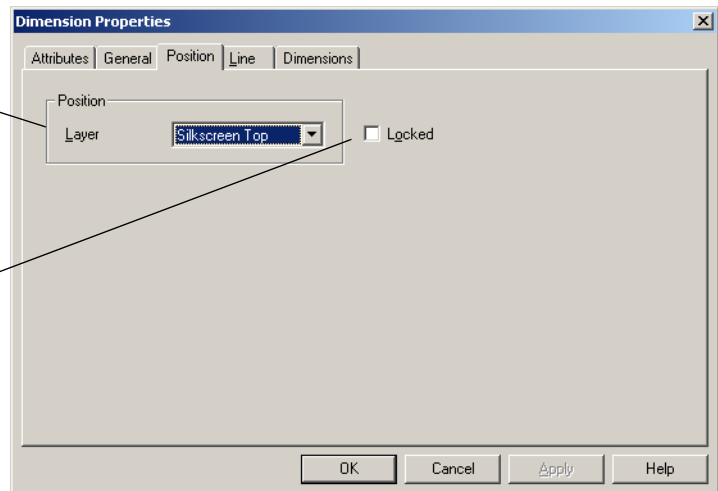border

Set the
dimension's
color

Set the width of
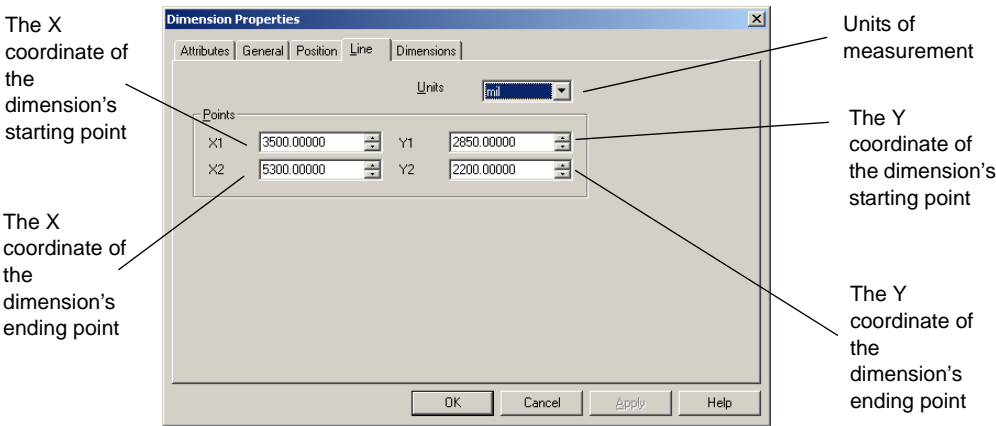the dimension's
lines and the
units of
measurement
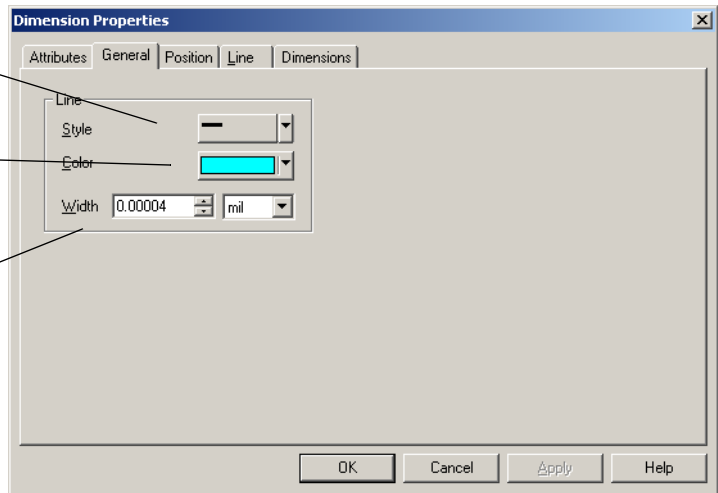
To change the layer on which a dimension exists, use the **Position** tab:
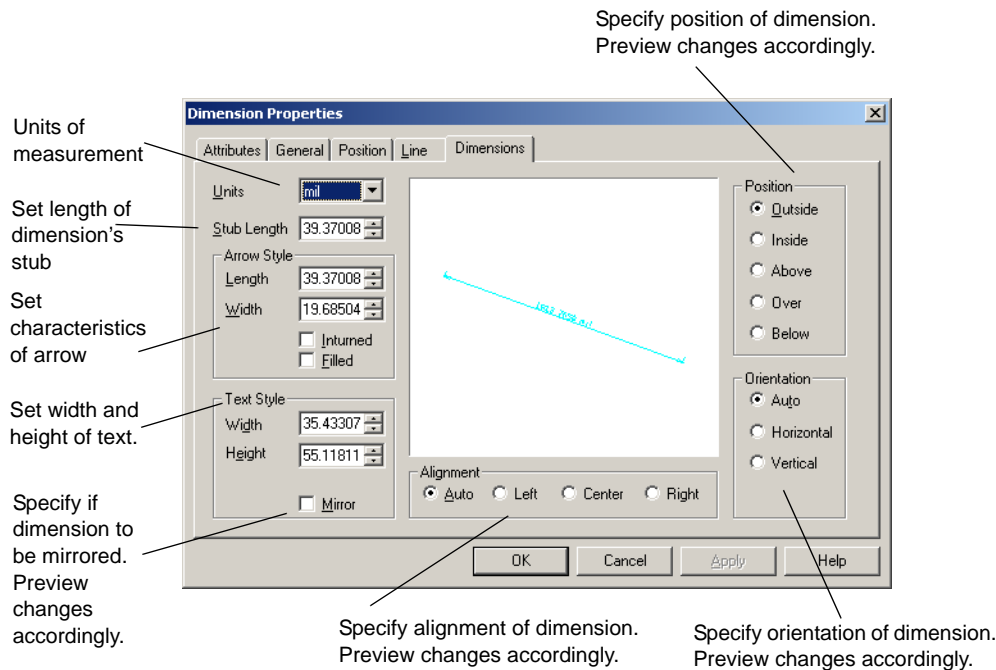
Select the
layer

Lock the shape
to the layer

To control the coordinates for the dimension's starting and ending points, use the **Line** tab:

The X coordinate of the dimension's starting point

Units of measurement

The Y coordinate of the dimension's starting point

The X coordinate of the dimension's ending point

The Y coordinate of the dimension's ending point

To control the various aspects of the dimension, use the **Dimensions** tab.

Specify position of dimension. Preview changes accordingly.

Units of measurement

Set length of dimension's stub

Set characteristics of arrow

Set width and height of text.

Specify if dimension to be mirrored. Preview changes accordingly.

Specify alignment of dimension. Preview changes accordingly.

Specify orientation of dimension. Preview changes accordingly.

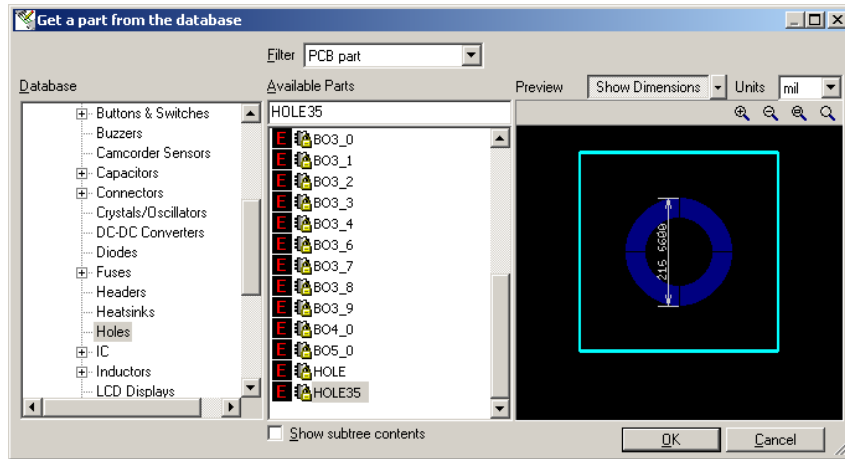# 6.4   Placing Parts from the Database

➢ To place parts from the database:

1. Choose **Place/From Database**. The **Get a Part from the Database** dialog box opens:

2. In the **Database** panel, expand the categories until you find the category where the part is. The parts appear in the **Available Parts** panel.

3. In the **Available Parts** panel, select the part you need. The part appears in the **Preview** panel. To manipulate the view of the part, click in the **Preview** area and use any of the following:

   • **Show Dimensions** button — displays selected dimensions of the part (unit of measure is set from the **Units** drop-down list). To change the displayed dimensions, click on the down-arrow beside the **Show Dimensions** button.
   • **Zoom In** button — 🔍 click to zoom in on the component for more detail. You can also press the F8 key.
   • **Zoom Out** button — 🔍 click to zoom out. Shows less detail and more of the whole component. You can also press the F9 key.
   • **Zoom Window** button — 🔍 click (or press F6) and then drag a rectangle around the portion of the part you want to enlarge. The area inside the rectangle enlarges to fill the **Preview** panel.
   • **Zoom Full** button — 🔍 click to view the entire part. You can also press CTRL + F7.
   • **Mouse Wheel** — if your mouse has a center wheel, you can use it to zoom in and out on the part.
   • **Scroll bars** — when the part has been enlarged beyond the borders of the **Preview** area, scroll bars appear that you can move in the usual manner to locate the desired section of a component.

4. Click **OK**. The **Get a part from the Database** dialog box disappears, and the following dialog box appears:

Enter Reference Designation for Component	✕
REFDES for shape HOLE35	
U7	OK
VALUE	Cancel
?	
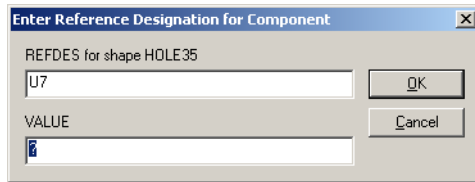
5. Enter the part's **REFDES** (Reference Designator) and **Value** (e.g., 20 Ohms for a resistor) and click **OK**.

6. Move the pointer over the board. The selected part is attached to the pointer.

7. When the part is in position, click to drop the part on the board.

8. If necessary, you can then adjust or move the part further into position. For more information on moving parts, see "6.1.3 Tools to Assist Part Placement" on page 6-5.

Tip When you place parts from the database you will want to add them to the netlist. For more information, see "7.6.2 Using the Netlist Editor" on page 7-23.

6.5   Editing Components and Shapes

6.5.1   Editing a Placed Part (In-Place Edit)

In-place part editing lets you add, delete, or change a part and the items that make it up. You can add, delete, or move pads, change or move the lines that define a part, or place new shapes or lines in the part.
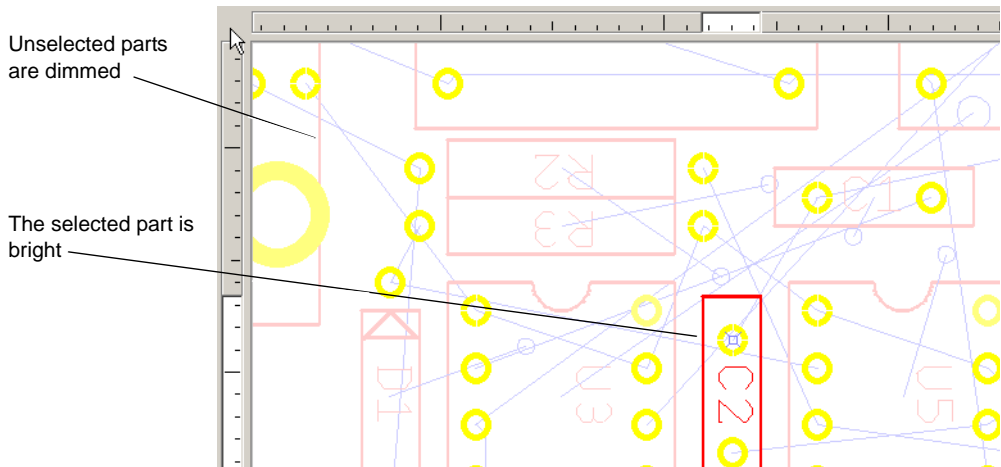
➢ To use **In-Place Edit** on a part:

1. Select the part and choose **Edit/In-Place Part Edit**.

   An editing window opens, showing the selected part. The window also shows the surrounding area of the board and the other components in a dimmed view, which cannot be edited:
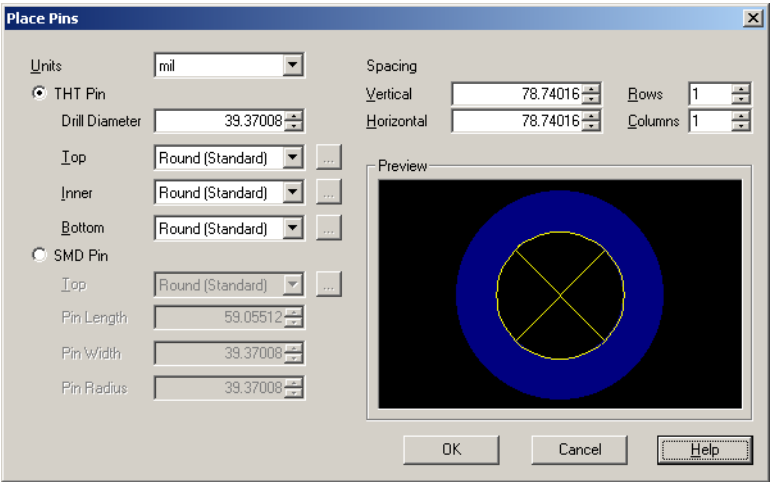

Unselected parts are dimmed

The selected part is bright

Note To change the degree of dimming displayed on the parts that are not being edited, see "3.3.3 Colors Tab" on page 3-16.

2. Edit the part using the Place and Draw tools described in "3.2 Toolbars" on page 3-4.

   To add a pin, choose **Place/Pins**. The **Place Pins** dialog box opens.

Indicate the pad type (THT or SMD) and its dimensions and spacing. A preview of the pad appears in the Preview panel. When you have made the necessary settings, click **OK**. The pad is attached to the pointer. Click on the part to place it.

3. When finished, choose **Edit/In-Place Part Edit** again to end the **In-Place Edit** function. The part appears with its changes.

Note You can save your edited part in the database for future use. For details, see "6.9.2.2 Adding Parts using the Add Selection to Database Command" on page 6-64.

6.5.2   Editing a Polygon

A vertex is a point of a polygon. You can add or remove vertices from polygons, whether copper or non-copper.
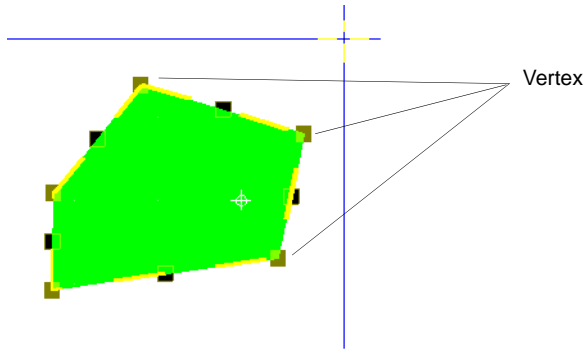
Vertex

➢ To add a vertex to any line segment of a polygon select the line (a selected line segment will have filled selection boxes while the other selection boxes in the polygon will be open) and

choose **Edit/Vertex/Add Vertex**. A vertex is added in the middle of the segment, and you can now move that vertex to change the shape of the polygon.

➢ To remove a vertex click on the point to be removed and choose **Edit/Vertex/Remove Vertex**. The vertex is removed, and the vertices on either side are joined by a straight line.

➢ To change the size of a design's vertices, edit the **Control Point Size** field in the **General Settings** tab of the **Preferences** dialog box. For details on this tab, see "3.3.1 General Settings Tab" on page 3-14.

6.5.3   Viewing and Editing Through Hole Pin Properties

➢ To edit through hole pin properties:

1. Select the desired through hole pin and select **Edit/Properties**.

2. The **Through Hole Pin Properties** dialog box appears, consisiting of five tabs: **Attributes**, **General**, **Pad**, **Autorouting Layers** and **Thermal Relief**.

➢ To edit a through hole pin's attributes, use the **Attributes** tab. For more information on editing properties in the **Attributes** tab, see "6.2.1 Attributes" on page 6-18.

To edit a through hole pin's display style, use the **General** tab:
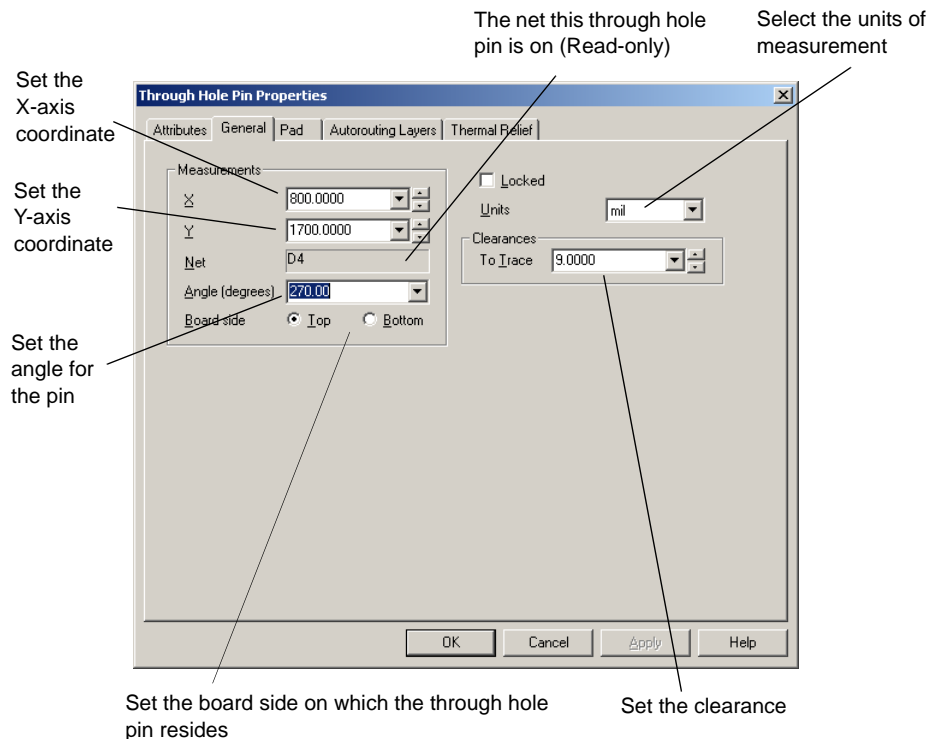
Set the
X-axis
coordinate

Set the
Y-axis
coordinate

Set the
angle for
the pin

The net this through hole
pin is on (Read-only)

Select the units of
measurement

Through Hole Pin Properties

Attributes | General | Pad | Autorouting Layers | Thermal Relief |

Measurements

X   800.0000

Y   1700.0000

Net   D4

Angle (degrees)   270.00

Board side   ⊙ Top   ○ Bottom

☐ Locked

Units   mil

Clearances

To Trace   9.0000

OK   Cancel   Apply   Help

Set the board side on which the through hole
pin resides

Set the clearance

Use the **Pad** tab to control the shape and size of the through hole pin. The **Pad Preview** on this tab shows the pad as it was created; the **Selection Preview** shows how the pad appears on the workspace.

Set the drill diameter and units of measurement.

Choose options to control the shape of the through hole pin. Preview changes accordingly. Shape settings can be set differently for each layer if desired.

Choose the appropriate option and, where necessary, value for the pad size. The Preview changes accordingly.

Choose the solder mask layer where the pad appears
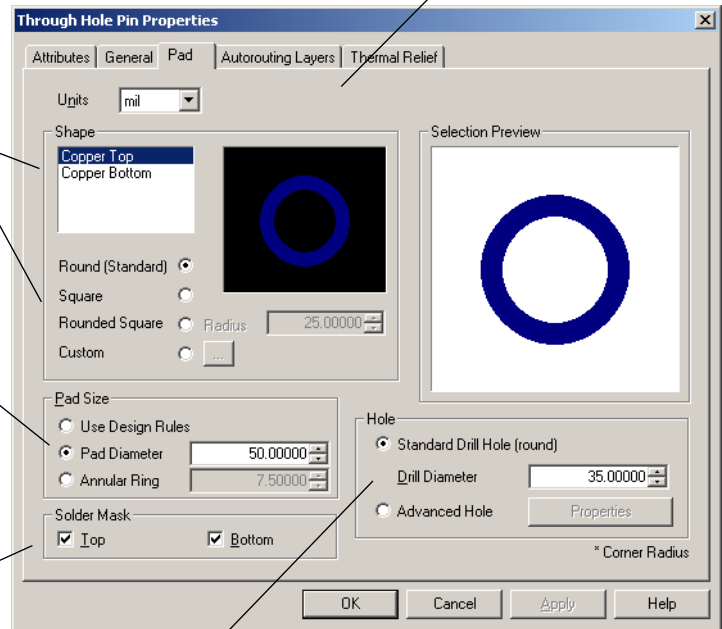
Choose the hole type. If you choose Advanced Hole, and click Properties, the Through Hole Pin Properties dialog appears. For details, see "6.3.2 Placing Holes" on page 6-31.

➢ Use the **Autorouting Layers** tab to choose which layers the through hole pin connects to.

Autorouting Layers

☑ Copper Top
☑ Copper Bottom

Autorouting Layers option is for the autorouting purpose only. It does not affect Netlist and DRC check. The via(s) still occupies the layers.
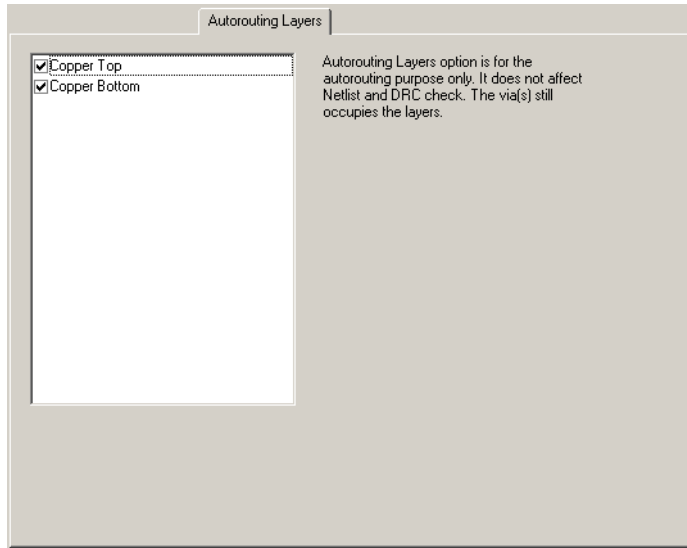
➢ Use the **Thermal Relief** tab to choose what type of thermal relief the pin will use when connecting to a copper area or power plane.

Thermal Relief

Relief Type

Use Polygon thermal relief settings ○
No thermal relief ○
X - Shaped thermal relief ⦿
+ - Shaped thermal relief ○
- - Shaped thermal relief ○
| - Shaped thermal relief ○
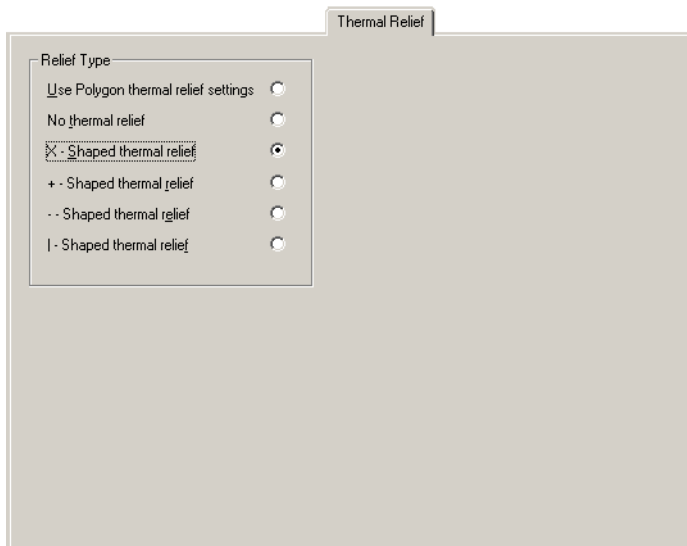
6.5.4   Viewing and Editing SMT Pin Properties

➢ To edit SMT (Surface Mount Technology) properties:

1. Select the desired SMT pin and select **Edit/Properties**.

2. The **SMT Pin Properties** dialog box appears, consisiting of five tabs: **Attributes**, **General**, **Pad**, **Thermal Relief**, and **Pin Neck**.

To edit an SMT pin's attributes, use the **Attributes** tab. For more information on editing properties in the **Attributes** tab, see "6.2.1 Attributes" on page 6-18.

To edit an SMT pin's display style, use the **General** tab:
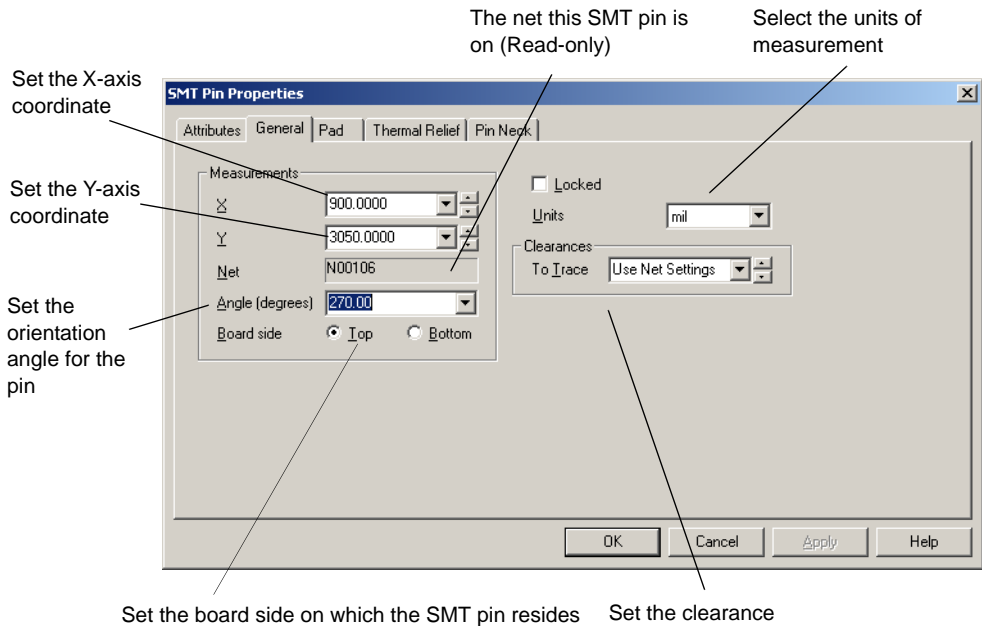
The net this SMT pin is on (Read-only)

Select the units of measurement

Set the X-axis coordinate

Set the Y-axis coordinate

Set the orientation angle for the pin

Set the board side on which the SMT pin resides

Set the clearance

Use the **Pad** tab to control the shape and size of the through hole pin. The **Pad Preview** on this tab shows the SMT pin as it was created, the **Selection Preview** shows how the SMT pin appears on the workspace.

Choose options to control the shape of the SMT pin. The Preview changes accordingly.
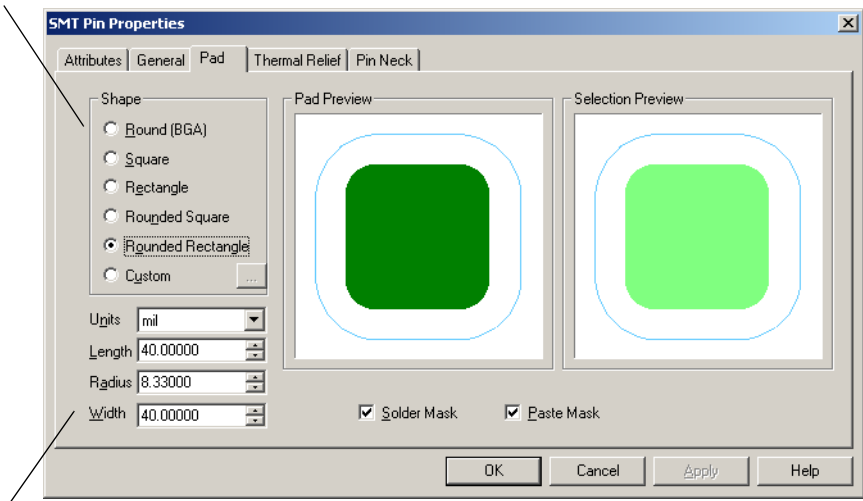
Set the appropriate values. The
Preview changes accordingly.

Use the **Thermal Relief** tab to choose what type of thermal relief the SMT pad will use.
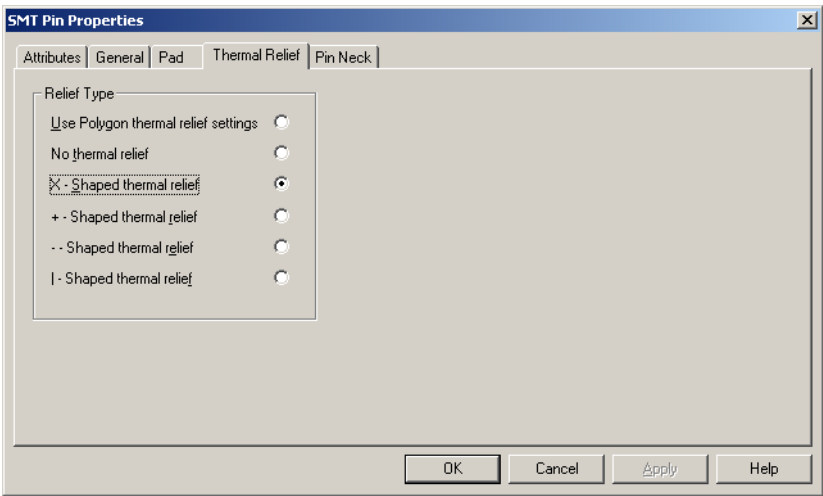
Use the **Pin Neck** tab to enter dimensions for the pin necks.
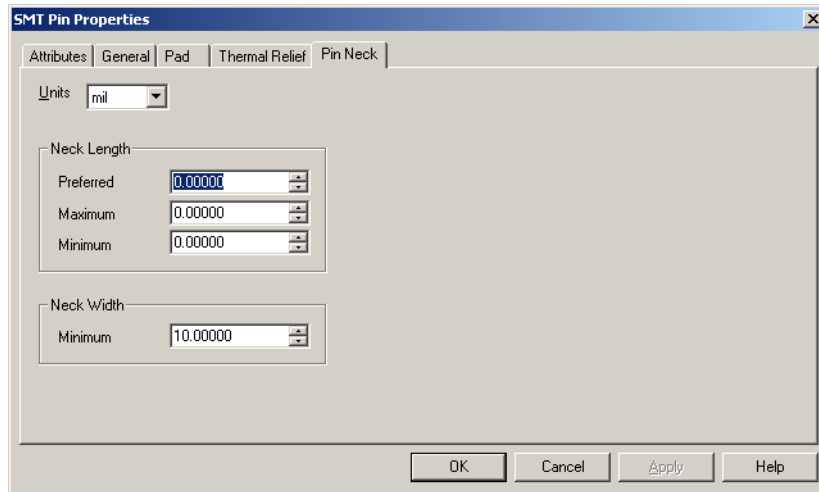
6.6 Searching For and Replacing Components

Ultiboard allows you to search for parts in two ways:

- Searching for parts in open designs. This method tells you if a specific part exists in all of the designs that are currently open.
- Locating a part in a design. This method finds a specific part in the design where you are currently working, and zooms in on the part.

You can also replace a part with one from the database.

6.6.1 Searching for Parts in Open Designs

To find out if a part exists in the open designs, you can search for it with the **Edit/Find** command. While this command works much like a Find function in other applications, it also allows you to search for a part by name, number, shape, value, or by all variables.

➢ To find out if a part exists in the open designs:

1. Choose **Edit/Find**. The **Find** dialog box opens.

Type the part name

Select a definition variable

Select to match whole word only

Select to match case

Click to start search
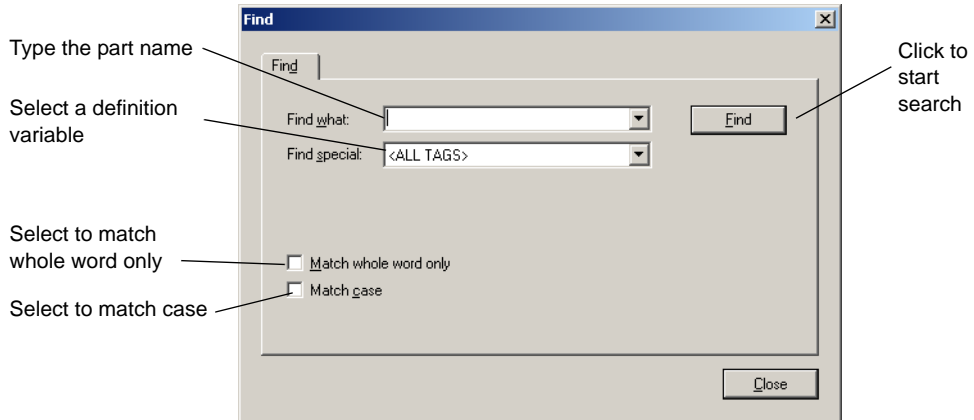
2. Enter your search criteria. You can search by all or part of the name of a part, or by variables that define parts: name, number, shape, or value. You can also refine the search by selecting the check boxes to match the whole word or to match the case.

3. Click **Find**. The search results appear in the **Results** tab of the **Speadsheet View**.

4. Double-click the item in the **Results** tab of the **Spreadsheet View** to zoom in and display the item in the workspace.

6.6.2   Locating a Part in a Design

To help you find specific parts in the open design, use the **Parts** tab:

Find and Select the Part button

	Refdes	Value	Shape	Locked	Trace Clearance	Component Spacing	Pin swap	Gate swap
●	C1	100N	CRM5A	No	0.0000	10.0000	Yes	Advanced Swapping
●	C2	150PF	CRM5A	No	0.0000	10.0000	Yes	Advanced Swapping
●	C3	330PF	CRM5A	No	0.0000	10.0000	Yes	Advanced Swapping
●	D1	5V6	DIOD1	No	0.0000	10.0000	Yes	Advanced Swapping
●	DSW1	DIP6	DIP16	No	0.0000	10.0000	Yes	Advanced Swapping
●	JP1	12_HEA...	FC14HF	Yes	0.0000	10.0000	Yes	Advanced Swapping

Results | DRC | Parts | Part Groups | Nets | Net Groups | SMT Pads | THT Pads | Vias | Copper Areas | Keep Ins/Outs | Layers | Statistics

➢ To display the **Parts** tab, click the **Parts** tab at the bottom of the **Spreadsheet View**.

➢ To find a part in the design:

1. Click the **Parts** tab in the **Spreadsheet View**.

2. Click the part in the list.

3. Click the **Find and Select the Part** button above the list of parts. The view zooms in on the part, which appears selected.

6.6.3 Replacing Parts

➢ To replace a part on the design with a part from the database:

1. Select the part.

2. Choose **Tools/Change Shape**. The **Get a Part from the Database** dialog box opens.
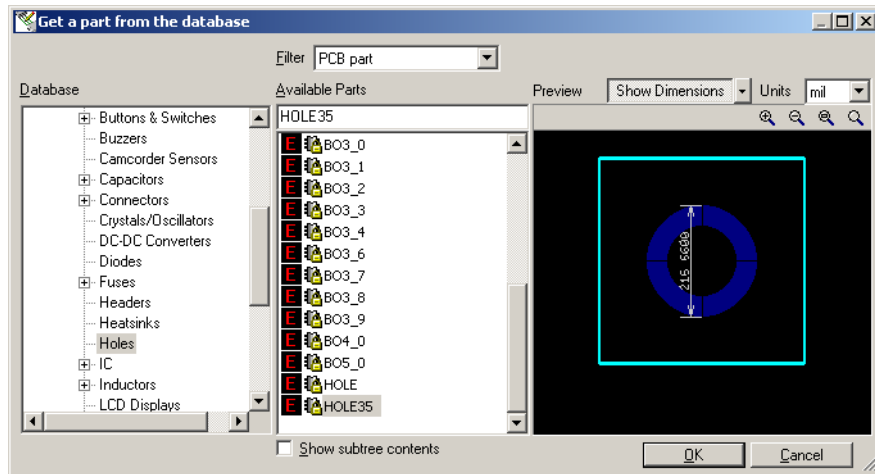
3. In the **Database** panel, expand the categories until you find the category where the part is. The parts appear in the **Available Parts** panel.

4. In the **Available Parts** panel, select the part you need. The part is previewed in the **Preview** panel when it is selected.

5. Click **OK** to replace the selected part on the design with the part you chose from the database.

6.7   Cross-probing

Cross-probing is the ability to highlight a selected component or group of components in Multisim or Multicap.

➢ To perform cross-probing:

1. Launch Multisim or Multicap and open the file that corresponds to the design you have open in Ultiboard.

2. In Ultiboard, select the components that you wish to highlight in Multisim/Multicap.

3. Select **Tools/Highlight Selection in Multisim**. The components are highlighted in Multisim/Multicap.

6.8   Creating New Parts

You can design your own parts if necessary, either through the **Database Manager**, or using the **Component Wizard**.

6.8.1   Using the Database Manager to Create a Part

➢ To design a part:

1. Choose **Tools/Database/Database Manager**.

2. Click the **New** button above the **Parts** panel. The **Select the part you want to create** dialog box appears.

Select the part you want to create

Net bridge
Custom Pad Shape
PCB part
CAD Part

OK   Cancel
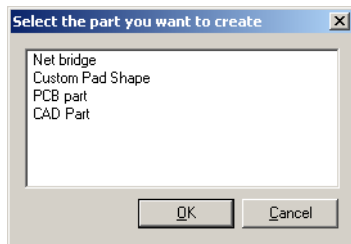
3. Select the type of part you want to create: a net bridge, custom pad shape, PCB part or mechanical CAD part and either double-click or click **OK**. The **Edit Mode** dialog box opens:

Footprint edit mode
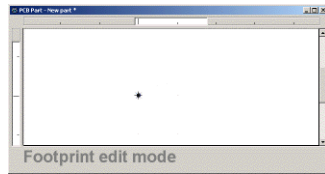
4. Design your part using the Place and Draw tools described in "3.2 Toolbars" on page 3-4.

5. To add a pin to a PCB part or net bridge, choose **Place/Footprint Pins**. The **Place Pins** dialog box opens.
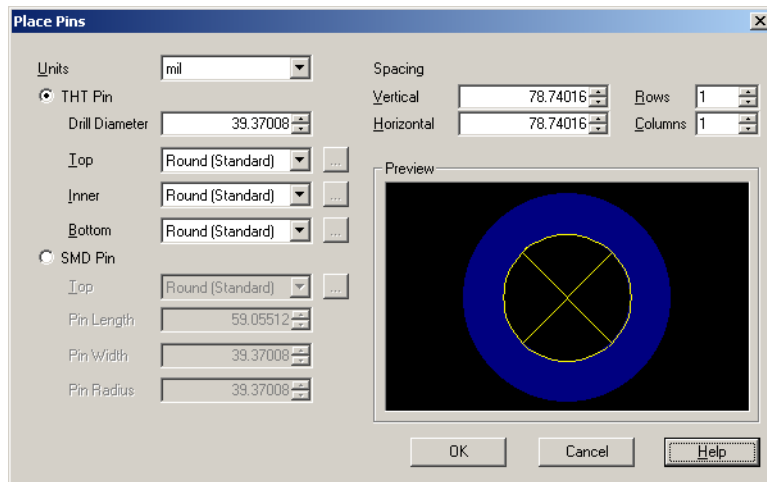
Indicate the pad type (THT or SMD), its dimensions and spacing, and click **OK**. The pin is attached to the pointer. Click on the part to place it.

6. When finished designing the part, choose **File/Save to database as**. The **Insert the selected Item(s) into the database** dialog box appears, with the cursor in the **Existing Parts** field.

7. Type a name for the new part. The name must be unique.

8. Click **OK**. The **Insert the selected Item(s) into the database** dialog box disappears, but the edit mode dialog box stays visible.

    The part can be placed from the database. For details, see "6.4 Placing Parts from the Database" on page 6-39.

➢ To return to the design screen:

Click the design in the **Projects** tab of the **Design Toolbox**.

6.8.2   Using the Component Wizard to Create a Part

The **Component Wizard** steps you through the process of creating a part.

➢ To use the **Component Wizard**:

1. Choose **Tools/Component Wizard**. Step 1 of the **Component Wizard** appears.

2. Choose one of the following:

    *   **TH** — Through Hole pin technology.
    *   **SMT** — Surface Mount pin technology.

3. Click **Next** to display step 2 of the wizard.

    If you chose **TH** in step 1 of the wizard, the following **Package Type** choices appear:

    Package Type
    *   ⊙ DIP (Dual in Line Package)
    *   ○ PGA (Pin Grid Array Package)
    *   ○ SPGA (Staggered Pin Grid Array Package)
    *   ○ SIP (Single in Line Package)
    *   ○ ZIP (Zigzag in Line Package)
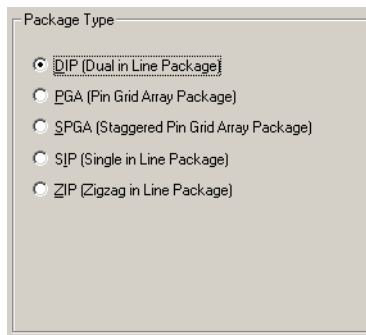
    If you chose **SMT** in step 1 of the wizard, the following package type choices appear:

    Package Type
    *   ○ SOT (Small Outline Transistor)
    *   ○ TO (Transistor Outline)
    *   ⊙ SO-Gullwing (Small Outline SOIC, SOP, TSSOP)
    *   ○ SO-J (Small Outline J Lead)
    *   ○ PLCC (Plastic Leaded Chip Carrier Package)
    *   ○ QFP (Quad Flat Package)
    *   ○ BGA (Ball Grid Array Package)
    *   ○ SBGA (Staggered Ball Grid Array Package)
    *   ○ SIP (Single in Line Package)
    *   ○ ZIP (Zigzag in Line Package)
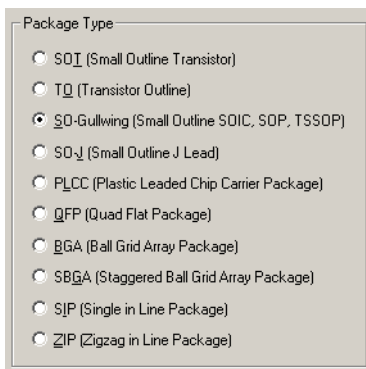
4. Choose the desired **Package Type** and click **Next** to display step 3 of the wizard.

    The following **Package Dimension** choices appear:

    *   **Units** — select the unit of measure from the drop-down list.

- **X** — the "x" dimension of the component (displayed on the preview).
- **Y** — the "y" dimension of the component (displayed on the preview).
- **3D Height** — the height of the the component, as displayed in the 3D view.
- **3D Offset** — the distance between the PCB and the bottom of the component.
- **Corner Cutoff - Notch** — the size of the notch on the component.
- **Corner Cutoff - Left Top** — places the notch on the left top of the component.
- **Corner Cutoff - Right Top** — places the notch on the right top of the component.
- **Corner Cutoff - Left Bottom** — places the notch on the left bottom of the component.
- **Corner Cutoff - Right Bottom** — places the notch on the right bottom of the component.
- **Circle Pin 1 Indicator** — enable to show a circle around pin 1 of the component.
- **Diameter** — the diameter of the circle around pin 1 of the component. Becomes active when **Circle Pin 1 Indicator** is selected.
- **Distance from Edge** — the distance between the circle around pin 1 and the edge of the component. Becomes active when **Circle Pin 1 Indicator** is selected.

Note Depending on the **Package Type** selected in step 3 of the wizard, some settings may not be available.

5. Make the desired **Package Dimension** settings and click **Next** to display step 4 of the wizard.

   The following 3D settings appear:

   - **Shininess** — use the slider to adjust the shininess of the component when seen in the 3D view.
   - **Colors** — click on the button beside each of the following parameters and select the desired color: **Background Light**; **Direct Light**; **Reflection Light**; **Emit Light**.

6. Make the desired 3D settings and click **Next** to display step 5 of the wizard. The options available will depend on whether you selected **THT** or **SMT** in step 1 of the wizard.

   If you chose **THT**, the following options appear:

   - **Units** — the unit of measure for the settings.
   - **Drill Hole - Diameter** — diameter of the through-hole pin's drill hole. This is displayed in the lower right preview (dimension "D").
   - **Shape** area — use to set the shape for the **Top**, **Bottom** and **Inner** sections of each pad. For each of these sections of the pad, you can choose individual shapes: **Round**; **Square**; **Rounded Square**; **Custom**. As you make your selections, they are reflected in the lower right preview area, which displays the currently active settings. If you choose **Custom**, the **Get a Part from the Database** dialog box appears, where you can selected any previously-created pad shapes. (See "6.8.1 Using the Database Manager to Create a Part" on page 6-52 for details).
   - **Pad Size** — select **Use Design Rules** if you wish to use the default values set up in the **PCB Properties** dialog box. Otherwise, enable **Pad Diameter** and **Annular Ring** and enter the desired values.

- **Settings Management** — this is useful if you wish to have different pad types and dimensions on the same component footprint. Click **Add** to create a new pad and then change the settings that are detailed above. You can create as many of these as you like, and choose between them by clicking on the "<<" and ">>" buttons. Remove any undesired pad types by clicking **Remove**. To change a pad type on the component, use the "<<" and ">>" buttons to select the desired pad in the lower right preview area and click on the pin you wish to change in the upper right component preview. The component preview will change to reflect the new pad information.

Note Pad types that you make using the **Add** button are for the convenience of adding unique pads to the current multiple pin component. If you create another component, these will not be accessible from the **Settings Management** area.

If you chose **SMT**, the following options appear:

- **Circular** — enable to make the pads circular and enter the desired **Diameter**. The changes are displayed in the lower right preview area.
- **Rectangular** — enable to make the pads rectangular and enter the desired **X** and **Y** dimensions. The changes are displayed in the lower right preview area.
- **Rounded Corner** — enable to make pads with rounded corners and enter the desired **X**, **Y** and **Corner Radius** settings. The changes are displayed in the lower right preview area.
- **Custom Pad** — enable to display the **Get a Part from the Database** dialog box, where you can selected any previously-created pad shapes. (See "6.8.1 Using the Database Manager to Create a Part" on page 6-52 for details).
- **Settings Management** — this is useful if you wish to have different pad types and dimensions on the same component footprint. Click **Add** to create a new pad and then change the settings that are detailed above. You can create as many of these as you like, and choose between them by clicking on the "<<" and ">>" buttons. Remove any undesired pad types by clicking **Remove**. To change a pad type on the component, use the "<<" and ">>" buttons to select the desired pad in the lower right preview area and click on the pin you wish to change in the upper right component preview. The component preview will change to reflect the new pad information.

Note Pad types that you make using the **Add** button are for the convenience of adding unique pads to the current multiple pin component. If you create another component, these will not be accessible from the **Settings Management** area.

7. Make the desired pad settings and click **Next** to display step 6 of the wizard.

The following **Pins** information appears:

- **Units** — the unit of measure.
- **Number of Pins** — the number of pins in the component.
- **Distances - Between Pins (A)** — the "A" distance between pins as shown on the preview.
- **Distances - Between Rows (B)** — the "B" distance between rows as shown on the

preview.

Note **Distances** information changes depending on the **Package Type** you selected in step 2 of the wizard.

8. Make the desired pin settings and click **Next** to display step 7 of the wizard.

   The following **Pad Numbering** information appears:

   • **Type of Pad Numbering** — available options will appear here. Many components only have the Numeric option; others are alpha-numeric; numeric-alpha.
   • **Direction of Pad Numbering** — displays the available options for the selected package type.
   • **Start Number Offset** — offsets the starting position of the pad numbers.

9. Make the desired **Pad Numbering** settings and click **Finish**.

10. The wizard closes, and the part is available for further editing in the **Footprint edit mode** using the Place and Draw tools described in "3.2 Toolbars" on page 3-4. For example, selecting and deleting extra pads in a BGA.

11. When you are finished, choose **File/Save to database as**. The **Insert the selected Item(s) into the Database** dialog box appears, with a cursor in the **Existing Parts** field.

12. Type a name for the new part. The name must be unique.

13. Click **OK**. The **Insert the selected Item(s) into the Database** dialog box disappears, but the **Footprint edit mode** screen stays visible.

   The component can be now placed from the database. For details, see "6.4 Placing Parts from the Database" on page 6-39.

➢ To return to the design screen:

   1. Click the design in the **Projects** tab of the **Design Toolbox**.

6.9   Managing the Database

The **Database Manager** allows you to add, organize, view, create, and manage all the parts and components that Ultiboard stores in its database.

➤ To open the **Database Manager**, choose **Tools/Database/Database Manager**.

Database panel buttons

Parts filter

Parts panel buttons

Click to display selected part's dimensions.

Database panel

Click to show subtree contents

Parts panel

Preview panel
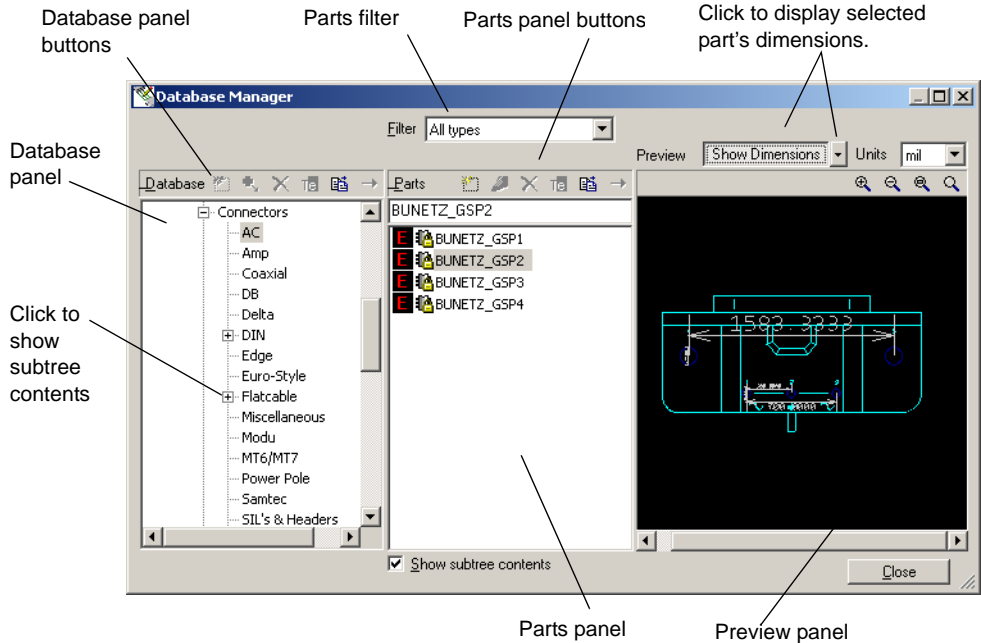
There are three panels in the **Database Manager**:

• The **Database** panel, which lists the databases and their sub-categories. The **Database** panel contains the following buttons. For more information on these functions, see "6.9.1 Working with Database Categories" on page 6-61.

	New	Create a new database category or sub-category.
	Delete	Delete a database category or sub-category.
	Rename	Rename a database category or sub-category.
	Copy	Copy a database category or sub-category.
	Move	Move a database category or sub-category.

Additionionally, the **Add** button allows you to add parts to either the **User** or **Corporate** database. For details, see "6.9.2.1 Adding Parts using the Database Manager" on page 6-63.

• The **Parts** panel, which lists the parts in the selected sub-category. The **Parts** panel contains the following buttons to help you work with the parts:

	New	Create a new part. See "6.8.1 Using the Database Manager to Create a Part" on page 6-52.
	Edit	Edit a part. See "6.5.1 Editing a Placed Part (In-Place Edit)" on page 6-40.
	Delete	Delete a part.
	Rename	Rename a part.
	Copy	Copy a part.
	Move	Move a part.

• The **Preview** panel, which allows you to preview the part you selected in the **Parts** panel. To manipulate the view of the part, click in the **Preview** area and use any of the following:
  • **Show Dimensions** button — displays selected dimensions of the part (unit of measure is set from the **Units** drop-down list). To change the displayed dimensions, click on the down-arrow beside the **Show Dimensions** button.
  • **Zoom In** button — click to zoom in on the component for more detail. You can also press the F8 key.
  • **Zoom Out** button — click to zoom out. Shows less detail and more of the whole component. You can also press the F9 key.
  • **Zoom Window** button — click (or press F6) and then drag a rectangle around the portion of the part you want to enlarge. The area inside the rectangle enlarges to fill the **Preview** panel.
  • **Zoom Full** button — click to view the entire part. You can also press CTRL + F7.

- **Mouse Wheel** — if your mouse has a center wheel, you can use it to zoom in and out on the part.
- **Scroll bars** — when the part has been enlarged beyond the borders of the **Preview** area, scroll bars appear that you can move in the usual manner to locate the desired section of a component.

In addition, there are two functions that allow you to control the information that appears in the **Database** and **Parts** panels:

- The **Filter** drop-down list allows you to display only PCB parts, custom PAD shapes, net bridges or mechanical CAD parts in the **Parts** panel.
- The **Show subtree contents** checkbox lets you control the display of the subtree contents from the **Database** panel.

6.9.1 Working with Database Categories

To make parts in the database easier to locate, the database organizes parts into categories and sub-categories. You can expand and collapse the categories and sub-categories in the **Database Manager** just as you would in similar Windows applications: click the plus (+) to expand a category or subcategory, and click the minus (-) to collapse it.

While there are default categories and sub-categories, you can create new ones if necessary. These are stored in the **User Database**.

You can copy any database category and its contents to another category or sub-category. The sub-categories that appear by default cannot be deleted, renamed, or moved, but you can delete, rename, or move the ones that you have added.

➢ To create a new database category:

1. Open the **Database Manager**.

2. In the **Database** panel, click the root or a subcategory to indicate where the new category belongs. The new category will be created as a sub-category of the item you select.

3. Click the **New** button above the **Database** panel. A new category is created with **New Object-Group** as the name.

4. Type a name for the new category and press ENTER.

➢ To copy a database category:

1. In the **Database** panel, select the category to be copied.

2. Click the **Copy** button above the **Database** panel. The **Select Destination in Database** dialog box appears.
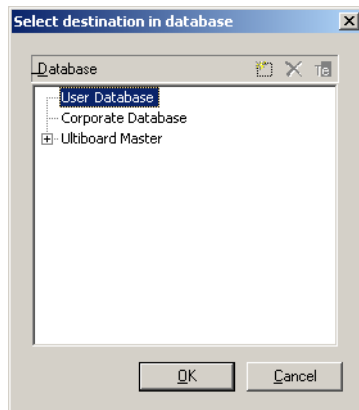
3. Select the category or sub-category where the category is to be copied. The copied category will go under the item you select.

4. Click **OK**. The category is copied to the location you specified, and the **Select destination in database** dialog box disappears.

➢ To delete a database sub-category:

1. In the **Database** panel, select the sub-category to be deleted.

2. Click the **Delete** button above the **Database** panel. You are prompted to confirm the deletion. The sub-category disappears from the list.

➢ To rename a database sub-category:

1. In the **Database** panel, select the sub-category to be renamed.

2. Click the **Rename** button above the **Database** panel. The name of the sub-category is highlighted, and you can change the name the same way you would in Windows Explorer.

➢ To move a database category or sub-category:

1. In the **Database** panel, select the category or sub-category to be moved.

2. Click the **Move** button above the **Database** panel. The **Select destination in database** dialog box appears.

3. Select the category or sub-category where the category is to be moved. The moved category will go under the item you select.

4. Click **OK**. The category is moved to the location you specified, and the **Select destination in database** dialog box disappears.

6.9.2   Adding Parts to the Database

Parts that appear on a design but do not exist in the database can be added to the database two ways:

- By using the **Database Manager**.
- By using **Tools/Database/Add Selection to Database**.

Note   You can only add parts to the **User Database** (or a sub-category of the **User Database**).

6.9.2.1   Adding Parts using the Database Manager

➢ To add parts to the database using the **Database Manager**:

1. In the **Database** panel, select the category (within the **User** or **Corporate Database**) you want to add the part to.

2. Click the **Add** button above the **Database** panel. The **Add new parts to database** dialog box appears.
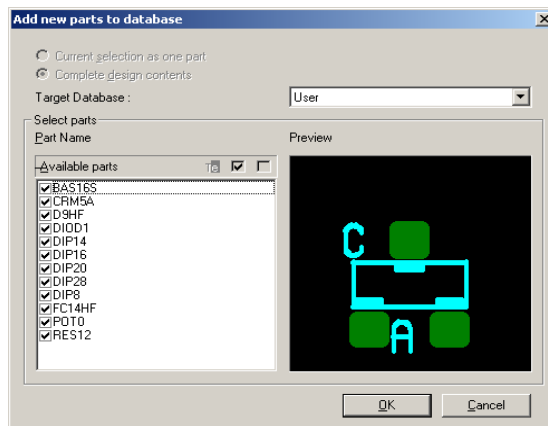
If you selected one or more parts on the design before opening the **Database Manager**, the part(s) are shown in the **Preview** panel and the **Current selection as one part** option is selected at the top of the dialog box.

If you want to add the parts individually, select the **Complete design contents** option (this is the default if no parts were chosen on the design prior to opening the **Database Manager**). The dialog box lists the parts in the design along with a preview of each.

Select the desired **Target Database** (where the parts will be saved).

3. Select the part to be added. To select all parts, click the **Select All Items** button (or press CTRL + A). To deselect all parts, click the **Unselect All Items** button.

4. To rename a part, highlight it and click the **Rename** button. The part name must be unique regardless of the database category you want to place it in.

5. Click **OK**. The part is added to the database.

6.9.2.2   Adding Parts using the Add Selection to Database Command

➢ To add a part to the database using the **Add Selection to Database** command:

1. In the design, select the part to be added. You can also select multiple parts that will be added as a group.

2. Choose **Tools/Database/Add Selection to Database**. The **Insert the Selected Items into the Database** dialog box opens, with the part illustrated in the **Preview** panel, and the part's name in the **Existing parts** field (unless multiple parts were selected).
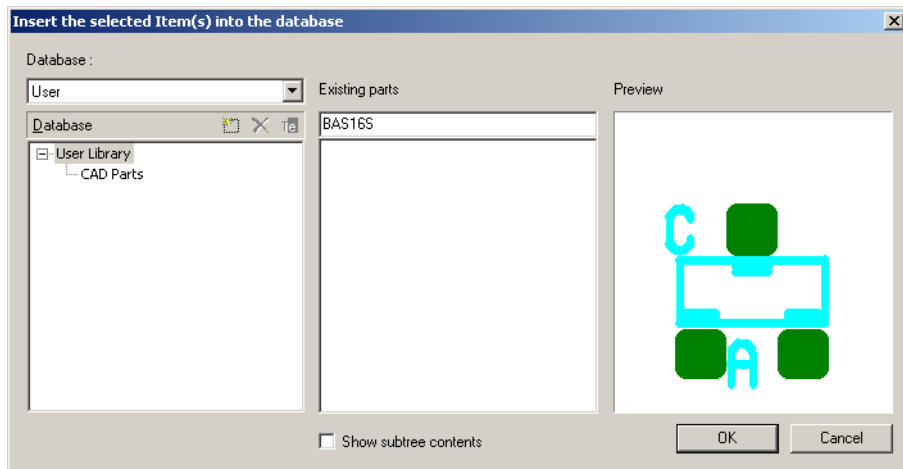
3. If necessary, enter or edit the part's name. The part name must be unique for the selected database, regardless of the category it will be stored in.

4. Select the database category or sub-category where the selected part is to be kept.

5. Click **OK**.

Note   If you selected multiple parts, you can save them to the database as one item. When a part that has been saved to the database in this manner is placed on the workspace, it will become separate items again, including any parts and traces that were in the original selection.

6.10   Merging and Converting Databases

You can merge components from one database into another or convert components that you created in your User Database in Ultiboard 2001 or Ultiboard 7 to Ultiboard 8 format.

Details follow.

6.10.1 Merging Databases

You can merge components from one database into another. For example, you may have created a number of components on your home computer that you wish to merge into a component database on your office computer.

➢ To merge databases:

1. Select **Tools/Database/Merge Database**. The **Database Merge** dialog box appears.

2. Click on **Select a Component Database Name** in the **Source Database** area. The **Select a Component Database Name** dialog box displays.

3. Navigate to the location of the database that you wish to merge (your source database) and select the type of database that you wish to merge in the **Files of type** drop-down list:

   ◈
   - **User (Usrcomp_s_*.usr)** — user database.
   - **Corporate (Cpcomp_s.prj)** — corporate database.

4. Highlight the source database file (the one you wish to merge) and click **Open**. You are returned to the **Database Merge** dialog box.

5. Select the **Target Database**. This is the database into which you will merge the components from your **Source Database**.

   ◈
   Corporate Database — components from the source database will be merged into the **Corporate Database**.

   User Database — components from the source database will be merged into the **User Database**.

6. Click **Start**. The following dialog box displays.
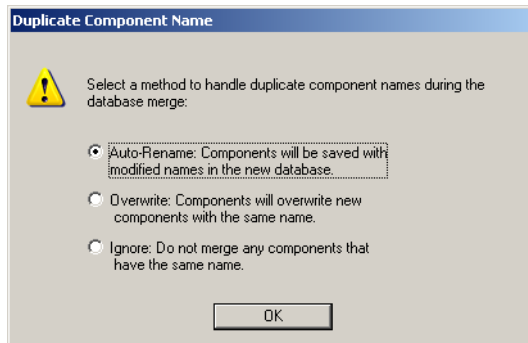
7. Select the desired option and click **OK**. The components from the source database are merged into the target database, based on the options selected above.

8. Click **Close** to close the **Database Merge** dialog box.

6.10.2 Converting 2001 or V7 Databases

If you are a user of Ultiboard 6, Ultiboard 2001 or Ultiboard 7, your **User Database** and **Corporate Database** must be converted to Ultiboard 9 format if you wish to use those components in Ultiboard 9.

Note The option to convert the **Ultiboard Master** database is not available, as a new **Ultiboard Master** database is loaded when you install Ultiboard 9.

Note If you are upgrading from Ultiboard 8, there is no need to convert these databases, as the formatting of the V8 and V9 databases is the same. However, it is recommended that you merge your existing V8 **User Database** and **Corporate Database** into Ultiboard 9 so that you have access to the components from these V8 databases. For details, see "6.10.1 Merging Databases" on page 6-65.

➢ To update your Ultiboard 6, Ultiboard 2001 or Ultiboard 7 databases to Ultiboard 8 format:

1. Select **Tools/Database/Convert V6/V7 database**. The **Convert Database** dialog box appears.

2. Click on **Select a Database File Name** in the **Source Database** area. The **Select a Database File Name** dialog box displays.

3. Navigate to the location of the database that you wish to convert (your source database), select the database type in the **Files of type** drop-down list, select the database and click **Open**. You are returned to the **Convert Database** dialog box.

4. Select the **Target Database**. This is the database into which you will merge the converted components from your **Source Database**.

🔷 **Corporate Database** — components from the source database will be converted and added to the **Corporate Database**.

User Database — components from the source database will be converted and added to the **User Database**.

5. Click **Start**. The following dialog box displays.
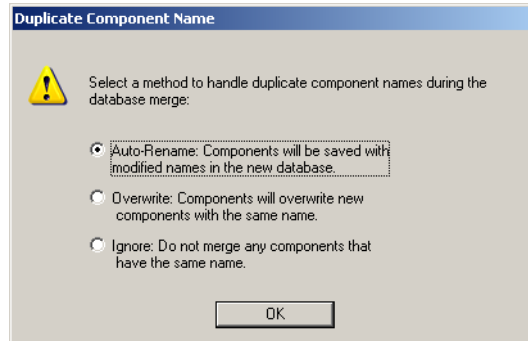
6. Select the desired option and click **OK**. The database is converted based on the option selected.

   - **Auto-Rename...** — Import and automatically rename the duplicate components.
   - **Overwrite...** — Replace the Ultiboard 9 components with Ultiboard 2001/Ultiboard 7 components.
   - **Ignore...** — Do not import Ultiboard 2001/Ultiboard 7 components with duplicate names.

7. Click **Close** to close the **Convert Database** dialog box.

Chapter 7
Working with Traces and Copper

The following are described in this chapter.

7.1   Placing Traces

You place the traces on the board by using one of the manual methods explained here, or by using the internal router or Ultiroute. For more information, see " Internal Router" on page 9-1.

The manual methods used to place traces are:

- **Manual trace**. This method allows you the maximum control in trace placement. You select the pads and vias to be connected, and dictate the path the trace takes to the next connection.
- **Follow Me router**. This method allows you to select the next pad or via to be connected by having Ultiboard follow your mouse movement as it places a trace. Ultiboard routes the resulting trace around most obstacles.
- **Connection Machine**. This method connects two pads or vias with a trace that is automatically routed around obstacles.

All methods of placing traces support the ratsnest selection method. This method starts the trace when the ratsnest is selected and auto-connects the trace to the nearest destination pad.

All methods except **Connection Machine** support continuous trace placement; when a trace is placed between two pads, the trace placement will continue from the last pad connected.

As you place a trace, and before you click to place it, you can always remove a segment by backing up over it. Each time you click while placing a manual trace, or each time a **Connection Machine** trace changes direction, a separate segment of that trace is created. When performing operations on traces, be sure to select either the appropriate segment or, if you wish, the whole trace.

Clearance is the distance from the edge of the board and around pads and traces that is to be kept free of any other elements. Trying to run a trace through a clearance, or trying to place a part so that a pad is put within a clearance, for example, results in an error. The board outline clearance is defined in the **PCB Properties** dialog box. Clearances for other copper elements are defined in the **General** tab of the element's properties. For more details, see "7.3 Viewing and Editing Copper Properties" on page 7-14.

➢ To view clearances, choose **View/Clearances**. The clearances are shown as fine blue lines around pads and traces.

7.1.1   Working with Traces

Ultiboard's default trace measures 10 mil wide and has a clearance of 10 mil. Clearances are measured from the outside edge of an object: a 10 mil trace with a 5 mil clearance would measure 20 mil across from edge to edge (5 mil clearance on one side, the 10 mil trace, and 5 mil clearance on the other side).

7.1.2   Placing a Trace: Manual Method

When you place a trace manually, you click pads and vias, and you must also click the trace's pivot points. This means that you have the most control over where the trace lies, but you must avoid placing the trace through parts and over other traces. If you try to place a manual trace through a part or over another trace, an error is generated in the **DRC** tab of the **Spreadsheet View**.

➢ To place a trace manually:

1. Choose a copper layer.

2. Select or enter the desired trace size in the **Draw Settings** toolbar.

3. Choose **Place**/**Line**.

4. Click a pad on the board. The net the pad is a part of is highlighted, and the pads in the net are each marked with an X.

5. Make your way to the next pad in the net. Remember that you have to avoid parts and other traces. Click to "glue" the trace to the board each time you change direction.

6. Click the next pad in the net. Continue in this way from pad to pad, clicking the points on the board where you must route the trace around obstacles.

7. When you place the final trace in the net, cancel trace placement by either pressing Esc twice or by right-clicking and selecting Esc from the pop-up menu that appears twice.

Narrowing/Widening Trace Width On-the-Fly

On occasion, it may be necessary to change the width of a trace segment (e.g., when placing it in a narrow area between pins). You can change the width of a trace segment on-the-fly as detailed below.

➢ To change the trace width on-the-fly:

1. Begin placing the trace as detailed above.

2. Click to place a trace segment, right-click and select either **Widen** or **Narrow** from the pop-up that appears.

3. Continue placing the trace and left-clicking to anchor trace segments. The width of the trace segments will either increase or decrease by 10% of the width of the previous trace segment, depending on whether you selected **Widen** or **Narrow**.

Tip   If you want to increase or decrease the next segment by more than 10%, right-click and select either **Widen** or **Narrow** multiple times.

Note   You will not be able to exceed the **Max Width** value for the net as set in the **Nets** tab of the **Spreadsheet View**, or set a width lower than the **Min Width** value.

7.1.3   Placing a Trace: "Follow Me" Method

While you must avoid obstacles such as other traces when placing a manual trace, a"Follow Me" trace avoids most of the obstacles that lie along a trace's route.

Note   For details on manual trace placement, see "7.1.2 Placing a Trace: Manual Method" on page 7-3.

➤ To place a "Follow Me" trace:

1. Choose a copper layer.

2. Choose **Place/Follow-me**.

3. Click a pad on the board. The net the pad is a part of is highlighted, and the pads in the net are each marked with an X.

4. Make your way to the next pad in the net. The trace follows the pointer, routing itself around most obstacles.

5. When you place the final trace in the net, cancel trace placement by either pressing ESC or by right-clicking.

7.1.4   Placing a Trace: Connection Machine Method

The **Connection Machine** is the simplest and fastest method of connecting two pads, but it cannot be used to connect more than two pads at a time.

➤ To place a "Connection Machine" trace with default routing:

1. Choose a copper layer.

2. Choose **Place/Connection Machine**.

3. Click a segment of a ratsnest on the board. The two pads connected by the segment of the ratsnest are connected with a trace that has been routed around obstacles.

4. Press ESC twice to end trace placement.

➢ To place a "Connection Machine" trace with custom routing:

1. Choose a copper layer.

2. Choose **Place/Connection Machine**.

3. Click a segment of a ratsnest on the board. The two pads connected by the segment of the ratsnest are connected with a trace that has been routed around obstacles.

4. Drag the trace segment to change the default routing around obstacles. The middle of the trace will move, although the trace remains anchored to the two specified pads.

5. Click to lock the moved trace segment in place.

6. Press ESC twice to end trace placement.

7.1.5   Placing a Bus

Use to connect multiple traces between multi-pinned devices such as ICs.

The procedure below uses the following example.
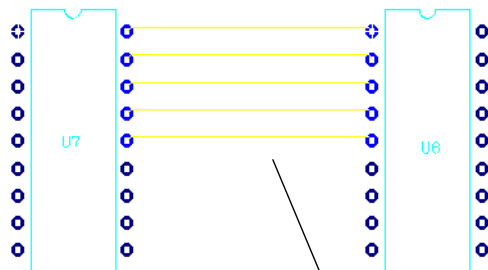
Bus 1 consists of these five nets
and is set up in the Edit Groups
dialog or the Spreadsheet View

➢ To place the copper for the bus in the above example:

1. Be sure that you have not selected any pins.

2. Select **Place/Bus**. The cursor changes to a bus symbol. Click on a pin on each of the nets that you wish to place in the bus and then move the cursor towards the target IC.

Click on a pin
from each of
the nets

The traces in the
bus are drawn as
you move the
cursor.
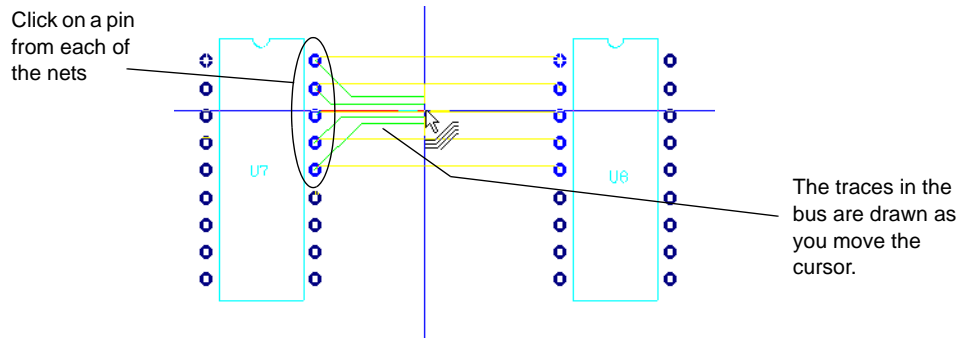
3. Move the cursor toward the bus's destination pins and double-click to complete copper placement.

Double-click to finish....

... connection of copper in bus
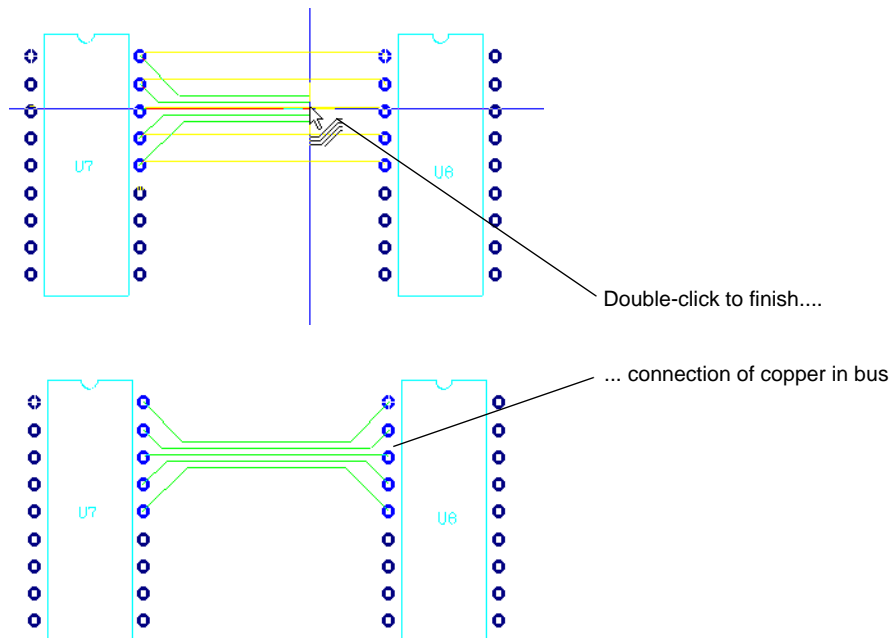
7.1.6   Working with Density Bars

Density bars use color to indicate the density of pins and pads at cross-sections of your board. The higher their density at any given cross-section, the more difficulty you will have routing traces through that section of the board and the more copper is used in that area. When placing parts you should strive to achieve relatively equal density distributions to avoid difficult-to-route areas.

As the design's pin/pad density gets heavier, the color of the density bars changes from green to red.

When you start Ultiboard, the density bars are toggled off by default. If you toggle the density bars on, they appear on the right side and underneath the design.

➢ To toggle the density bars on, or off again, choose **View/Density bars**.

7.1.7   Working with Keep-in/Keep-out Areas

7.1.7.1   Placing Keep-in/Keep-out Areas

The **Keepin/out Properties** dialog box lets you place a polygon that will act as either a keepin or a keepout area. By default, a keepout is placed. To change to a keepin area, or change any other properties, use the **Keepin/out** tab.

➢ To add a restricted area:

1. Choose **Place/ Keep-in/Keep-out Area**.

2. Left-click all points that are to define the polygon, ending with the starting point.

7.1.7.2   Viewing and Editing Keep-in/Keep-out Properties

➢ To display the **Keepin/out Properties** dialog box, select the desired keepin/out and select **Edit/Properties**. (The **Keepin/out** layer must be active).

Keepout area properties consist of two tabs: **Attributes** and **Keepin/out**.

The **Attributes** tab allows you to edit the properties of the selected keepout area. For more information on editing properties in the **Attributes** tab, see "6.2.1 Attributes" on page 6-18.

The **Keepin/out** tab lets you set specific parameters.

Select keepout or keepin

Check layers to which the Keepin/out applies

Set advanced options by clicking checkbox and then active button

Optionally, enter name of keepin/out area selected
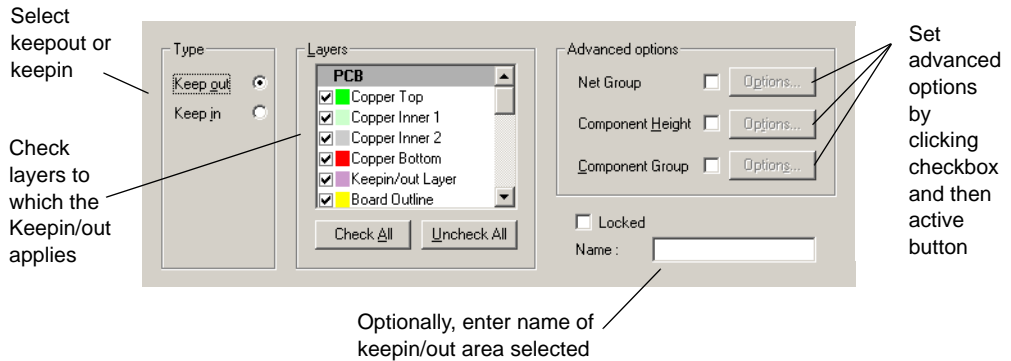
➢ To set advanced options:

1. Click on one of the following checkboxes in the **Advanced options** area and then click the **Options** button when it becomes active:

   Net group — displays the **Select Groups** dialog box where you select the net groups to which you wish to apply the keepin/out.

   Component Height — displays the **Component Height Ranges** dialog box where you enter the height of components to which you wish the keepin/out area to apply.
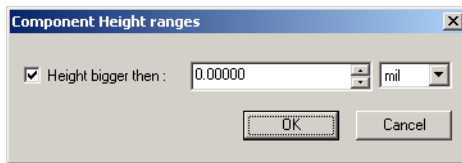
   Component Group — displays the **Select Groups** dialog box where you select the component groups to which you wish to apply the keepin/out area.

2. Click **OK** in the **Keepin/out Properties** dialog box.

If no **Advanced options** are set:

- A keepin area will keep all objects within the keepin area or report a DRC error if an object is outside the area.
- A keepout area will keep all objects outside the keepout area or report a DRC error if an object is inside the area.
- If there are multiple keepins or keepouts, the objects may be divided amongst the different areas at your discretion. Multiple areas then behave as a single disjointed area.

If any **Advanced options** are set:

- A keepin area will keep all specified objects within the keepin area or report a DRC

error if an object is outside the area.
- A keepout area will keep all specified objects outside the keepout area or report a DRC error if an object is inside the area.

7.1.8 Equi-Spacing Traces

◈ This option lets you set the spacing between traces to be equal.

➢ To equi-space traces:

1. Select two traces that surround at least one other trace.
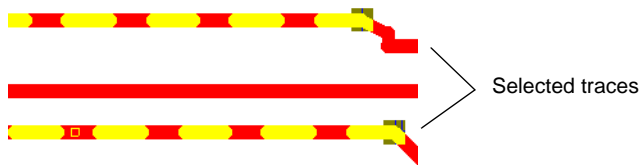
Selected traces

2. Select **Tools/Equi-space traces**. The spaces of the traces is made equal.

Note The selected traces must be between two objects (i.e., two pins, two traces); initial spacing between traces must not be equal; traces must belong to a net in the same layer. Rubber-banding does not work with this feature, hence some traces may become disconnected.

7.1.9 Deleting a Trace

Traces can be deleted like other objects. When you delete a trace, you are not asked to confirm the deletion, but you can undo the deletion by choosing **Edit/Undo** immediately after making the deletion.

Note Depending on your setting in the **PCB Design** tab of **Options/Global Preferences**, vias associated with a trace will be deleted when the trace is deleted.

➢ To delete a trace that you have just placed:

↺ Choose **Edit/Undo Place Trace Segment**.

➢ To delete an existing trace:

1. Select the trace.

✕ 2. Choose **Edit/Delete**.

   Or

1. Select the trace.

2. Press the DELETE key.

7.2 Working with Other Copper Elements

This section contains the following subjects:

❒ "7.2.1 Placing Copper Areas" on page 7-10

❒ "7.2.2 Placing Powerplanes" on page 7-11

❒ "7.2.3 Splitting Copper" on page 7-11

❒ "7.2.4 Converting a Copper Shape to an Area" on page 7-11

❒ "7.2.5 Deleting All Copper" on page 7-12

❒ "7.2.6 Adding Teardrops" on page 7-13

7.2.1 Placing Copper Areas

Use the **Place/Copper Area** command to define copper polygons.

➢ To place a copper area:

1. Choose a copper layer.

2. Choose **Place/Copper Area**. The pointer has a polygon shape attached.

3. Left-click all points that are to define the copper area, ending with the starting point.

4. Right-click to cancel the **Place** command.

➢ To delete a copper area, choose **Edit/Copper Delete/Copper Island** and then click on the copper area you want to delete.

7.2.2   Placing Powerplanes

Powerplanes are copper areas that cover the entire plane.

➢ To place a powerplane:

1. In the **Layers** tab, select the layer to be used as a powerplane.

2. Choose **Place/Powerplane**. The **Choose Net and Layer for Powerplane** dialog box appears.

Choose Net and Layer for PowerPlane	☒
Net	[▼]
Layer	[Copper Top ▼]
	OK Cancel
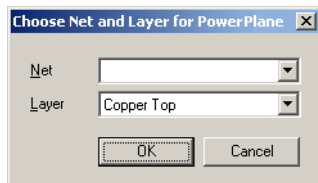
3. Specify the net and the layer for the powerplane.

4. Click **OK**. The **Choose Net and Layer for Powerplane** dialog box disappears and the powerplane is placed on the layer that you specified.

7.2.3   Splitting Copper

The **Polygon Splitter** is used to split copper areas or powerplanes.

➢ To split a copper area or powerplane:

1. Choose **Design/Polygon Splitter**.

2. Move the pointer to the polygon you want to split.

3. Click at the point where you want to begin the split.

4. Move the pointer over the polygon. A line will appear to indicate where the split will occur. When it's in the place you want, click to finish the split.

5. Right-click to cancel the **Polygon Splitter** function.

7.2.4   Converting a Copper Shape to an Area

Use to convert to a polygon that supports voiding around unconnected nets.

➢ To shape a copper shape to an area:

   1. Select a copper shape in your workspace.

Selected copper area
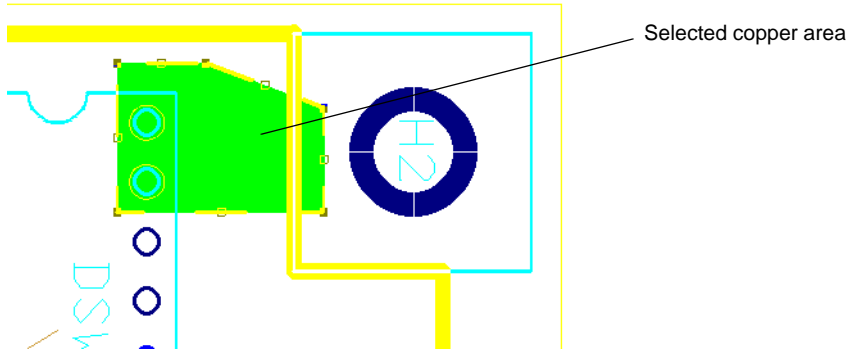
   2. Select **Design/Shape to Area**.

Copper is cut out to
avoid other elements
within its area
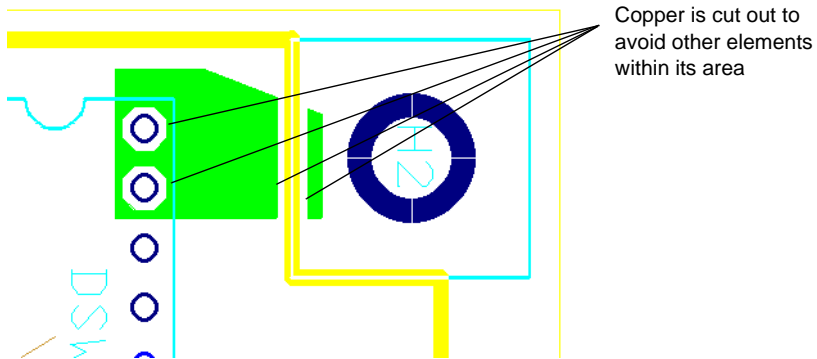
7.2.5 Deleting All Copper

If you want to delete all copper elements (traces, copper areas, and powerplanes) and start over, make sure the design is open and choose **Edit/Copper Delete/All Copper.** This deletes all copper elements in the design.

7.2.6   Adding Teardrops

A teardrop is a flair that you can add to a trace where the trace connects to a pad. This is typically used with very small sized traces, to prevent possible breakage in the copper between the trace and the pad.

➢ To add teardrops:

1. Select **Design/Add teardrops**. The **Teardrops** dialog box displays.

Select units for teardrop length

Enter preferred and minimum length of teardrop

Display changes as lengths are changed

Click to ignore pads that are not connected to a net

Select pad types to apply teardrop to

Select options for action to take when a DRC error occurs
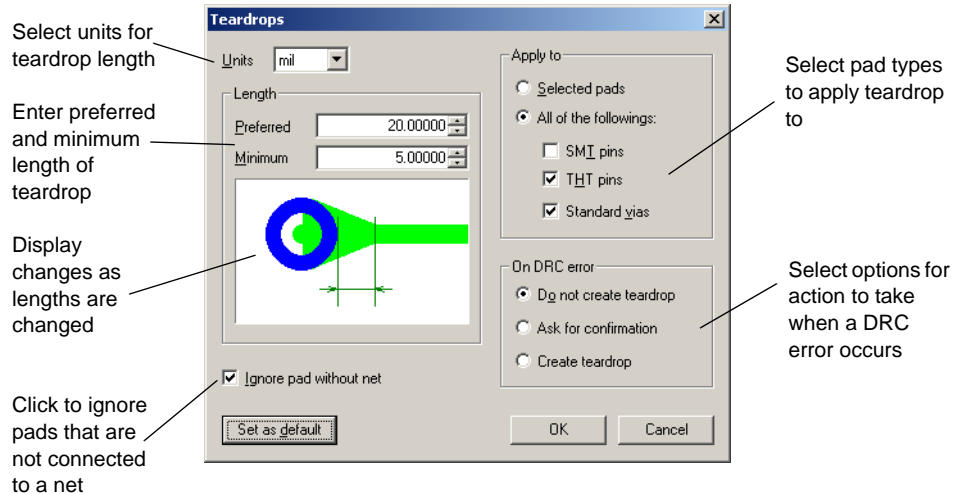
2. Set lengths and options as desired and click **OK**. Teardrops are added.

7.2.6.1   Removing Teardrops

➢ To remove all teardrops select **Edit/Copper Delete/All Teardrops**.

7.3   Viewing and Editing Copper Properties

All copper elements (traces, copper areas, and powerplanes) share the same three properties tabs (**Attributes**, **General**, and **Position**). The fourth tab for traces is **Line**, and the fourth tab for copper areas and powerplanes is **Copper Area**.

The **General** tab is the default, and appears when you choose **Edit/Properties**. It lets you edit the properties of the selected copper element as shown below:

Trace width

The net the
trace is a part of

Trace type
(fixed or variable)

Units of
measurement

Set the clearance
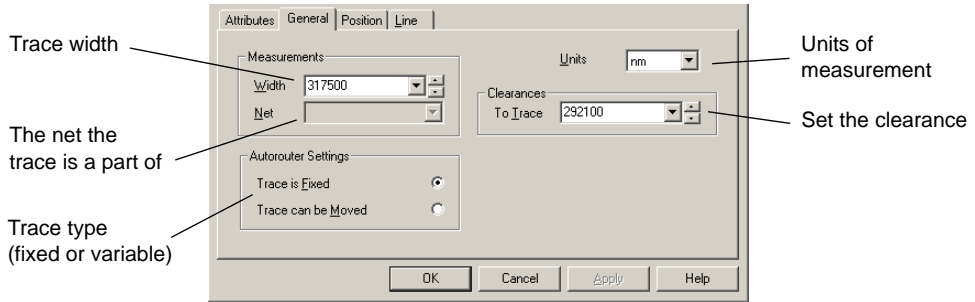
User-placed traces are fixed (default setting) so that they will not be moved when autorouting is performed.

The **Attributes** tab is where you edit the attributes of the selected copper element.

For more information on editing properties in the **Attributes** tab, see "6.2.1 Attributes" on page 6-18.

The **Position** tab allows you to change the layer the selected copper element is on. You can also use this tab to lock the copper element on the layer:

Select the layer from
the drop-down list

Click to lock
the trace to
the layer
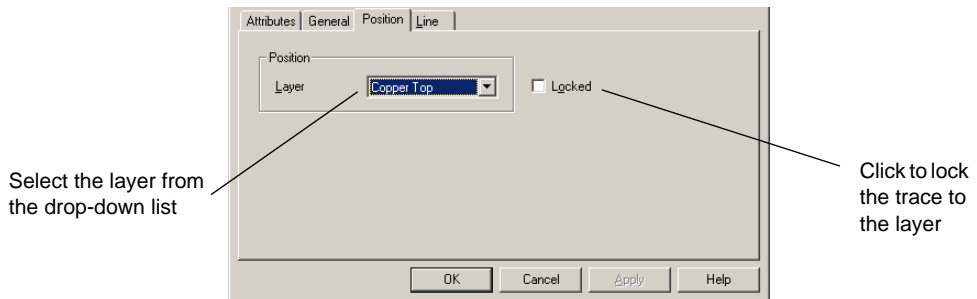
The **Line** tab is the default when the selected copper element is a trace. It allows you to control the coordinates for the trace's starting and ending points:

The X coordinate of the trace's starting point

The X coordinate of the trace's ending point

The Y coordinate of the trace's starting point

The Y coordinate of the trace's ending point

When a copper area is selected, the **Copper Properties** dialog box includes a **Copper Area** tab. To set a copper area's net and parameters, use this tab:

Select the net

Select to have the area void over traces

Select the thermal reliefs and styles allowed

Select the parameters to remove islands. An island is a section of copper that is not connected to any other copper.

Select to replace all islands that you removed manually (i.e., by selecting and deleting).

7.4   Working with Vias

This section contains the following topics:

❐ "7.4.1 Placing Vias" on page 7-16

❐ "7.4.2 Viewing and Editing Via Properties" on page 7-17

7.4.1   Placing Vias

A via is a plated through-hole in a printed circuit board used to connect two or more layers, as well as the top and bottom surfaces of the board.

Once placed, a via can be moved like a part. For information on moving and adjusting objects, see "6.1.3 Tools to Assist Part Placement" on page 6-5.

Note   See also "5.1.1 Defining Copper Layers" on page 5-2.

➢   To place a via:

1.   Choose **Place/Via** and click on the board where you want to place the via. A dialog box appears listing all copper layers available on the board.

Select the lamination that is to be used for this via	✕

From Layer		To Layer	
From layer	Object information	To layer	Object information
Copper Top		Copper Bottom	
Copper Bottom			

Create a via from : Copper Top () to : Copper Bottom ()
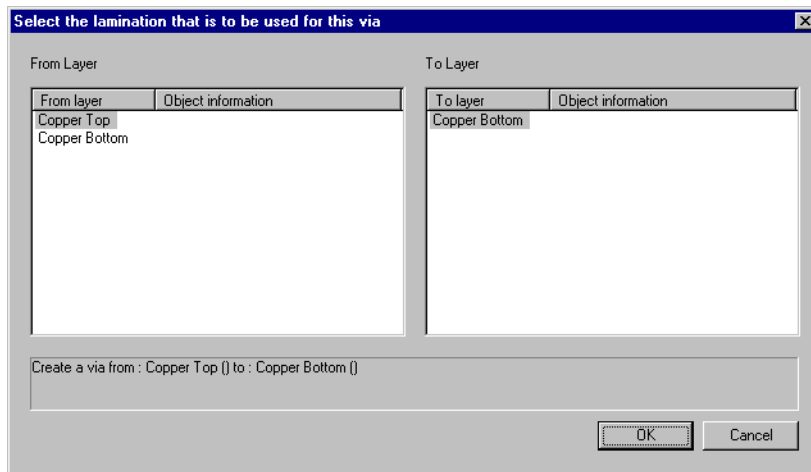
[OK]   [Cancel]

2.   Select the layers that the via is to run between.

3.   Click **OK**. The dialog box disappears.

4.   Right-click to cancel the **Place Via** command, or click in another location to place another via.

Note   Depending on your setting in the **PCB Design** tab of the **Preferences** dialog box, vias associated with a trace will be deleted when the trace is deleted.

7.4.2   Viewing and Editing Via Properties

Via properties consist of five tabs: **Attributes**, **General**, **Via, Autorouting Layers** and **Thermal Relief**.

The **Attributes** tab allows you to edit the properties of the selected via.

For more information on editing properties in the **Attributes** tab, see "6.2.1 Attributes" on page 6-18.

The **General** tab is the default, and appears when you choose **Edit/Properties**. It allows you to change the X/Y coordinates, the size of the clearance, the via angle, the side of the board the via is on, and to define the units of measurement.
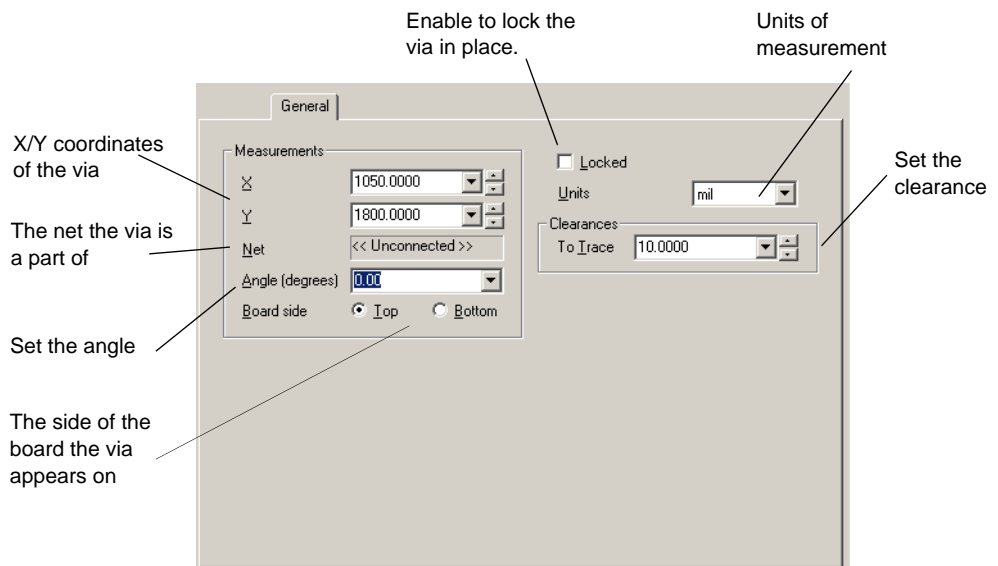
The **Via** tab is where you set up the parameters shown below.

Enable to assign a specific net to the via, then select the net from this drop-down list.

Enable to use settings in Pads/Vias tab of PCB Properties dialog.

Enable to enter Pad and Drill diameter for the selected via in the dropop-down lists.

Select whether via is fixed, or can be moved during autorouting.

Select solder masks as desired (default behavior is enabled).
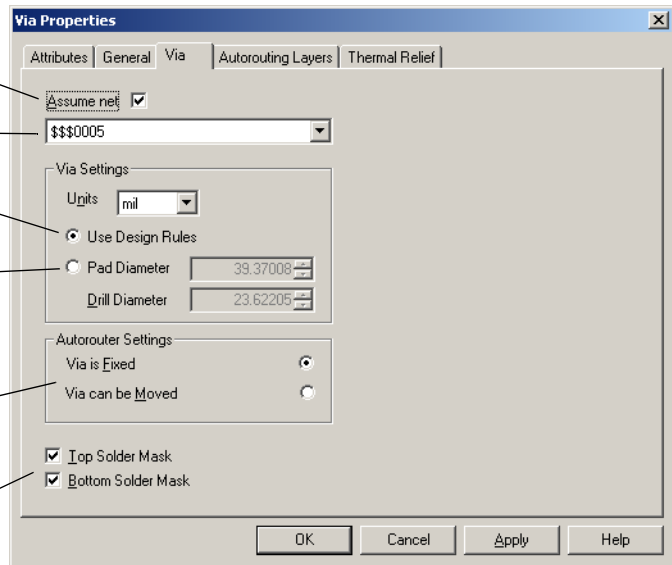
➢ Use the **Autorouting Layers** tab to choose which layers the via connects. .
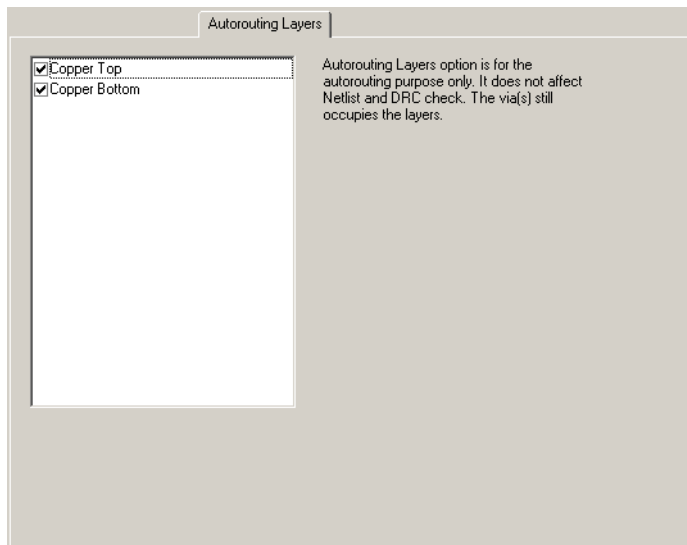
The **Thermal Relief** tab lets you choose if the via uses a thermal relief and, if so, what kind.
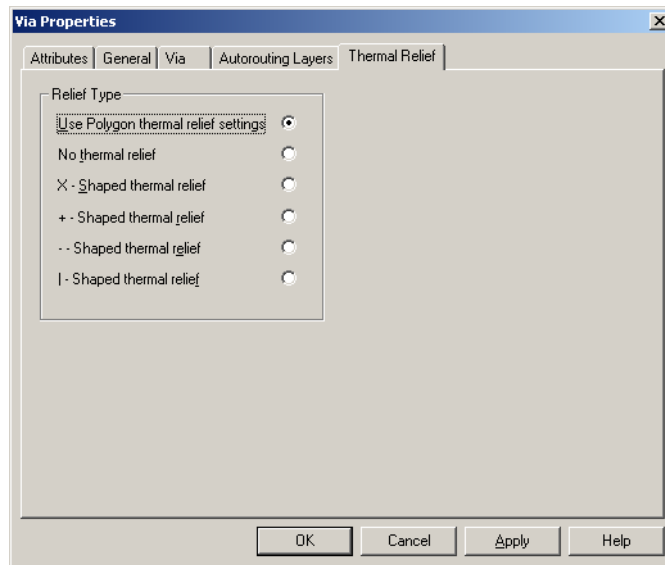
7.5 Placing SMD Fanouts

The **Fanout SMD** command attaches vias to each pin of either a selected SMD (Surface Mount Device) or all SMDs on the board.

➢ To place a fanout from an SMD component:

1. Optionally, select the component(s) to which you wish to apply fanouts.

Selected component
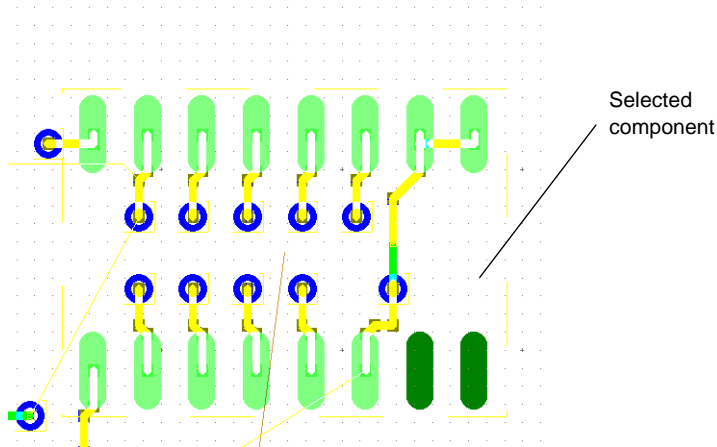
2. Select **Design/Fanout SMD**. The **Fanout Options** dialog box appears.

Select whether fanout applies to entire board or selected component
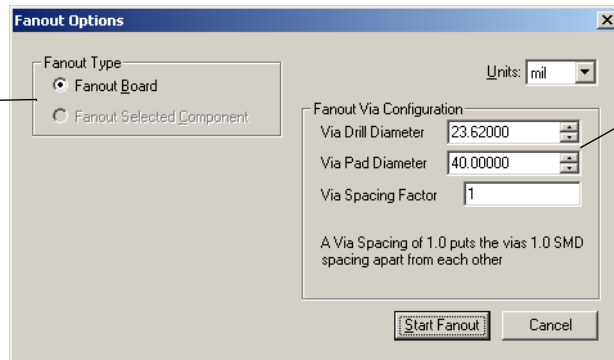
Enter via parameters

3. Enter options as desired and click **Start Fanout**. The **Fanout Options** dialog box disappears and the fanout vias are placed on the design.

Placed SMD via fanout
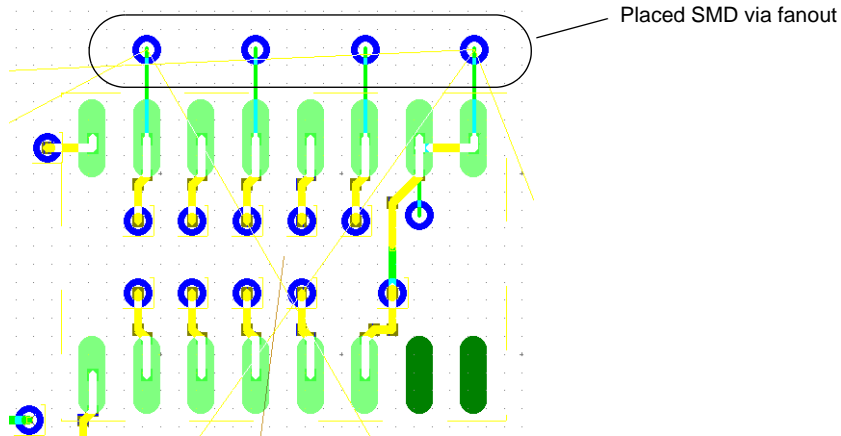
7.6   Working with Nets

This section contains the following subjects:

7.6.1 Using the Nets Tab

The **Nets** tab buttons in the **Spreadsheet View** allow you to preview a net in relation to the board, find a net, highlight a selected net, lock and unlock a selected net, set the width and clearance of a selected net, and remove the copper of a selected net.
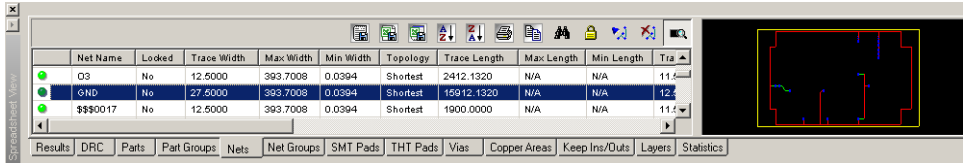
➤ To preview a net:

1. Click the **Show or Hide the Preview** button to toggle the **Preview** function on if it is not already.

2. Click the net in the list. A picture of the path the net makes on the board displays in the preview area.

➤ To find a net in the design:

1. Click the net in the list to select it.

2. Click the **Find the selected net** button. The view zooms in on the net and selects it.

➤ To highlight a selected net:

1. Click the net in the list to select it.

2. Click the **Highlight selected nets** button. The selected net is highlighted on the design. (You can change the highlight color from the **Color Element** drop-down list in the **Colors** tab of the **Preferences** dialog box).

➤ To lock and unlock any copper placed for a net:

1. Click the net in the list to select it.

2. Click the **Lock the selected net** button to lock an unlocked net, or to unlock a locked net.

➤ To remove the copper of a selected net:

1. Click the net in the list.

2. Click **Remove Copper**. You are prompted to confirm the removal.

3. Click **Yes** to remove the copper. The copper is deleted and the ratsnest reappears.

Note For details on the other buttons in this tab, see "3.6 Spreadsheet View" on page 3-32.

7.6.2   Using the Netlist Editor

The **Netlist Editor** can be used to view the nets in the design and to view the pins in the nets. You can also use it to add and remove nets from the design, add/delete pins from an existing net, adjust trace widths in a net, set high speed parameters and adjust a net's via diameters and via drill hole size.

➢ To open the **Netlist Editor**, choose **Tools/Netlist Editor**. The **Net Edit** dialog box opens.
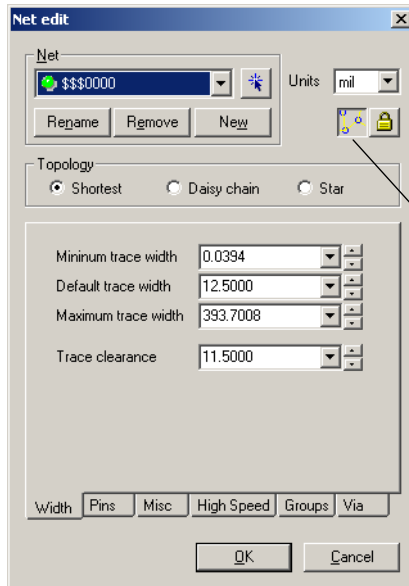
When depressed, shows ratsnest for the selected net. For details on ratsnests, see "6.1.3.1 Working with Ratsnests" on page 6-5.

7.6.2.1 Adding a Net

➢ To add a net:

1. Select **Tools/Netlist Editor** and click the **Pins** tab.

Click to add a net
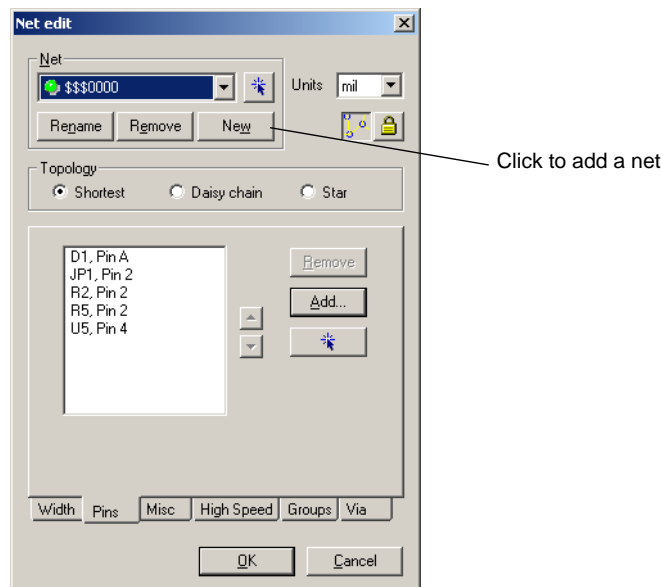
2. Click **New**. The **Add Net** dialog box displays.
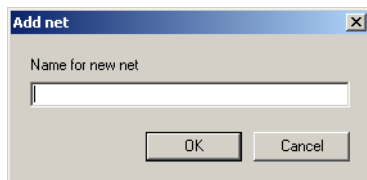
3. Enter a name and click **OK**.

New net name displays here.
The light green "lamp"
indicates that there are no
pins connected to the net.

New net has no pins connected.

See below to add pins to net.
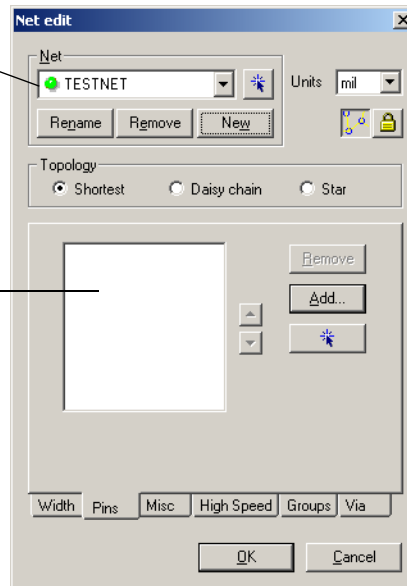
> To add a pin to a new or previously existing net:
>   1. Select **Tools/Netlist Editor** and click the **Pins** tab.
>   2. Select the desired net from the **Net** drop-down list in the **Net edit** dialog box.

You can also select the net by clicking on the button to the right of the drop-down list and then clicking on one of the pins from that net in the workspace. To do this, there must already be pins connected to the net.
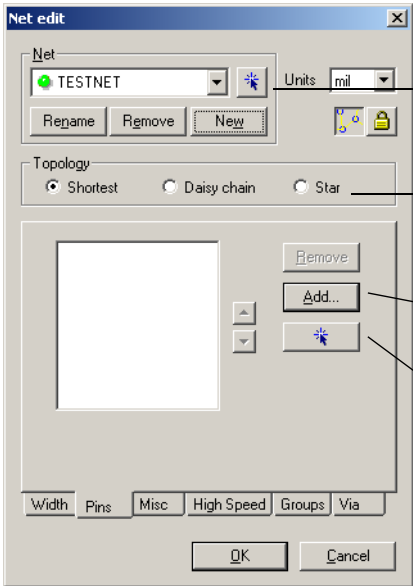
Select net by using drop-down list or button as described above.

Select topology of net. This feature is not available in all versions of Ultiboard. See page 7-28 for details.

Click to add pins to net using "Add Pins to Net" dialog box. See next step.

Add Pins button. Click to add pins to net by clicking on them in the workspace. See next step.

The remainder of this section uses the following sample components:
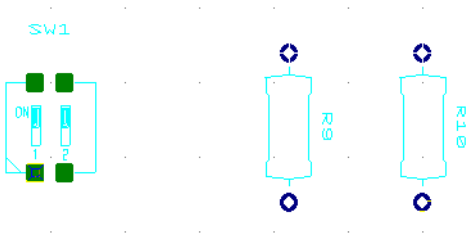
These components are not connected to any net.

3. Click the **Add pins** button and click the desired pin in the workspace. Continue until all pins for the net are listed in the **Pins** area.
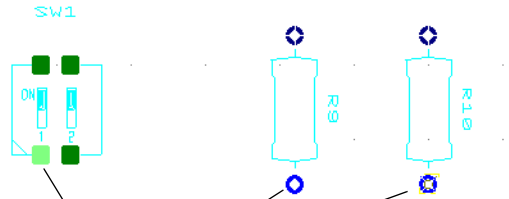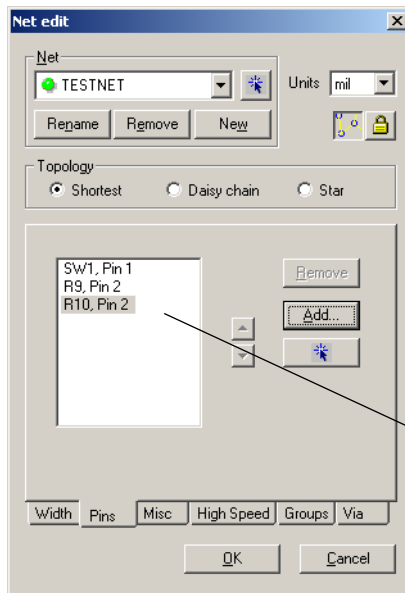
As pins are clicked, they are highlighted in the workspace.

At the same time, their identifiers appear here.

Or

Click **Add**. The **Add Pins to the Net** dialog box displays.
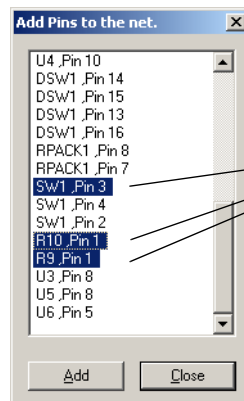
Pins can be selected using a combination of the CTRL and SHIFT keys and the left mouse button.

Highlight the pins to be added and click **Add**. The dialog box closes and the **Net edit** dialog box shows the added pins in the list of pads for the displayed net.

4. Click **OK** in the **Net edit** dialog box. The dialog box closes and the net information is added in the workspace.

Force vectors for new net

Ratsnest for new net
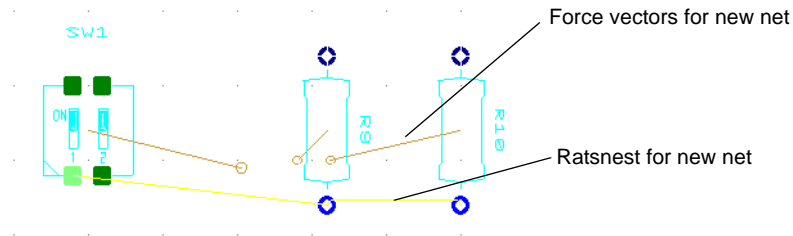
➢ To change a net's topology.

1. Select **Tools/Netlist Editor** and select the net from the **Net** drop-down list in the **Net edit** dialog box.

2. Click either **Shortest**, **Daisy chain** or **Star** in the **Topology** area and click **OK**. The ratsnest placement on the workspace changes to reflect the new topology.
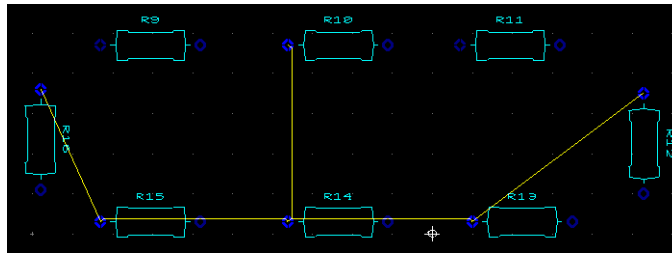
Shortest
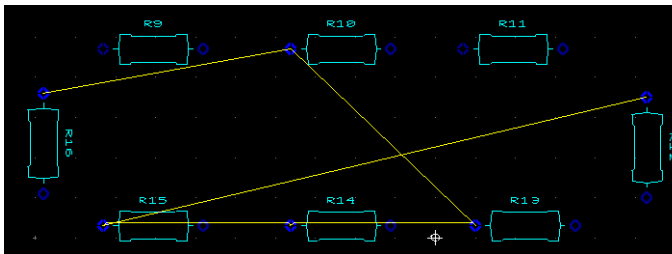
Daisy chain
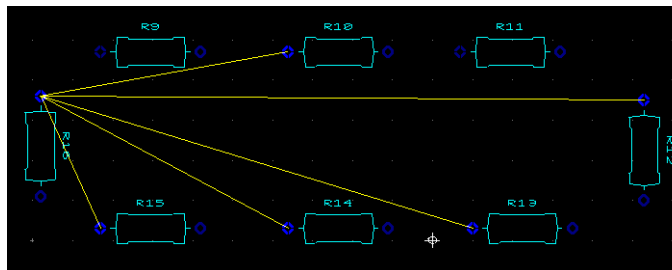
Star

Shortest — When connections are made, the shortest distance possible will be maintained. The order for the connection is not considered.

Daisy chain — The connection between pins is based on the order in which the pins are selected. The connection distance between pins is not considered.

Star — Pins are connected with a reference point, which is the first selected pin. Other pins will only be connected to the reference source. The effect is like a star, with connections "shooting out" to other pins. Orders in which the pins are selected or the distance between pins are not considered. If the first pin is deleted from the net, then the pin below the reference point will become the source.

7.6.2.2 Renaming a Net

➢ To rename a net:

1. Select **Tools/Netlist Editor** and select the desired net from the **Net** drop-down list in the **Net edit** dialog box.

2. Click **Rename** in the **Net edit** dialog box. You are prompted for the new net name.

3. Type a name for the net.

4. Click **OK** to save the new name or **Cancel** to cancel the operation.

   The new net name appears in the **Net** drop-down list in the **Net edit** dialog box, and in the **Nets** tab in the **Spreadsheet View**.

7.6.2.3 Removing a Net

To remove a net:

1. Select **Tools/Netlist Editor** and click **Remove** in the **Net edit** dialog box.
   The **Select the net to delete** dialog box opens.
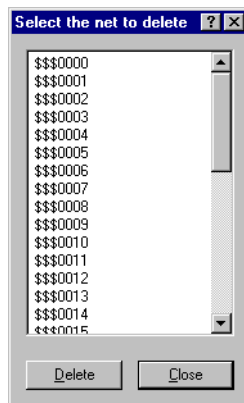
2. Select the net(s) to be deleted.

3. Click **Delete** to delete the net(s). The **Select the net to delete** dialog box closes, and the net no longer appears in the **Net** drop-down list of the **Net edit** dialog box, or the **Nets** tab in the **Spreadsheet View**. The net is also removed from the board, but while the net disappears, the traces stay on the board.

Note   There is no deletion confirmation. To cancel the deletion before you click **Delete**, click **Close**.

7.6.2.4   Deleting a Pin from a Net

➢ To delete a pin from a net:

1. Select **Tools/Netlist Editor**, click the **Pins** tab and select the desired net from the **Net** drop-down list in the **Net edit** dialog box.

2. Highlight the pin you wish to delete and click **Remove**.

Highlight the pin to be deleted and click Remove.
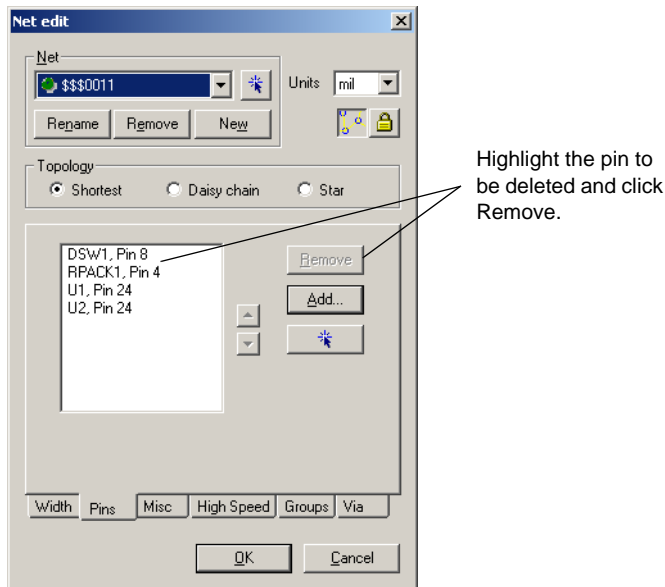
The pin disappears from the list of pins for the net displayed.

Note   There is no deletion confirmation.

3. Click **OK** to close the **Net edit** dialog box.

7.6.2.5 Setting Net Widths

➢ To set net widths:

1. Select **Tools/Netlist Editor** and select the desired net from the **Net** drop-down list in the **Net edit** dialog box.

2. Click on the **Width** tab.
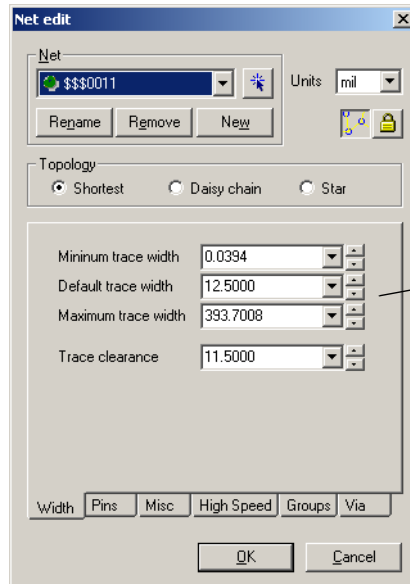
Enter parameters or choose "Use Group Settings" from the drop-down lists

3. Click **OK** to save changes.

7.6.2.6 Setting High Speed Parameters

➢ To set high speed parameters for a net:

1. Select **Tools/Netlist Editor** and select the desired net from the **Net** drop-down list in the **Net edit** dialog box.

2. Click the **High Speed** tab.

Checkbox is only active if "Daisy chain" or "Star" topology is selected. This feature is not available in all versions of Ultiboard.
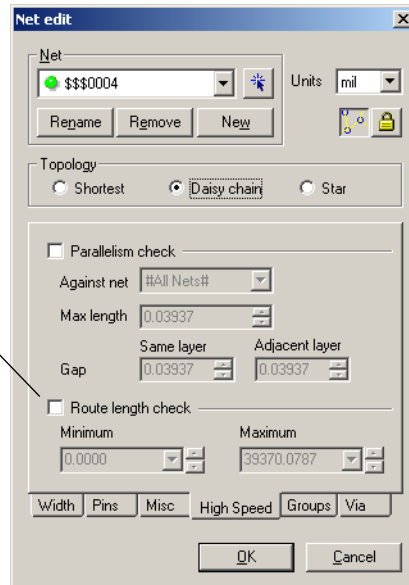
Enter the following as required:

Parallelism check — click checkbox and enter **Maximum length** that traces can run in parallel. Enter size of **Gap** (allowable distance) between parallel traces on **Same layer** and **Adjacent layer**. In the **Against net** field, select the net against which the parallelism check is done.

Route length check — click checkbox and enter **Minimum** and **Maximum** lengths for the length of copper between connected pins for **Daisy chain** and **Star** topologies.

3. Click **OK** to close dialog box and accept changes.

7.6.2.7 Setting Miscellaneous Net Parameters

➢ To set miscellaneous parameters for a net:

1. Select **Tools/Netlist Editor** and select the desired net from the **Net** drop-down list in the **Net edit** dialog box.

2. Click on the **Misc** tab.
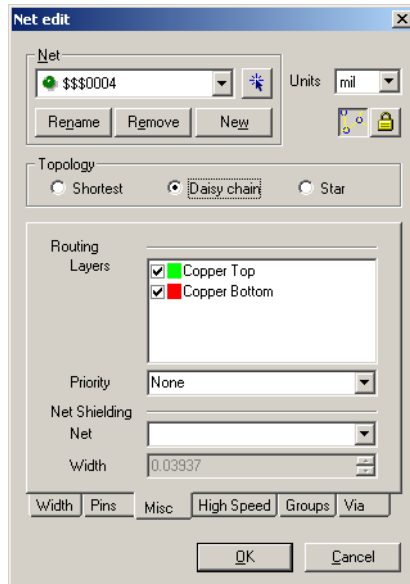
3. In the **Layers** area:

   • select layers to use for routing copper for the selected net.

4. In the **Priority** field:

   • enter the routing priority for the selected net. "1" is the highest priority, "2" the second highest, etc. Leave as None if priority routing is not required.

5. In the **Net Shielding** area:

   • select which **Net** is used for the shield. **Width** becomes active.

   Note Shields are used to place copper around a selected net to act as a buffer or shield the net from signal interferance.

6. Enter desired parameters and click **OK**.

7.6.2.8 Setting Group Parameters

➢ To enter group information for a net:

1. Select **Tools/Netlist Editor** and select the desired net from the **Net** drop-down list in the **Net edit** dialog box.

2. Click on the **Groups** tab.
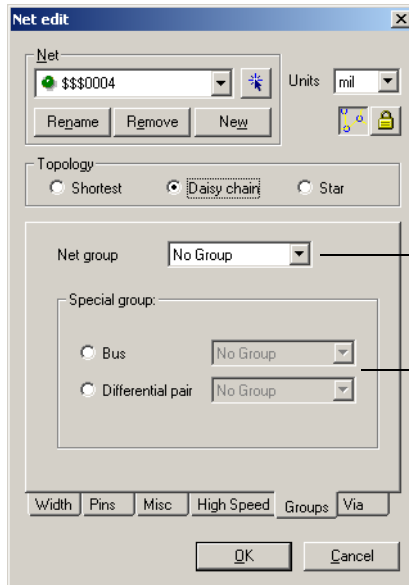
Select net group from drop-down.

To enter a group for a bus and/or differential pair for the selected net, click the "Bus" and/or "Differential pair" radio button and select the group from the drop-down list. (These groups are created in the Edit Groups dialog box).

3. Enter desired group parameters and click **OK**.

7.6.2.9 Setting Via Parameters

➢ To edit via information for a net:

1. Select **Tools/Netlist Editor** and select the desired net from the **Net** drop-down list in the **Net edit** dialog box.

2. Click on the **Via** tab.
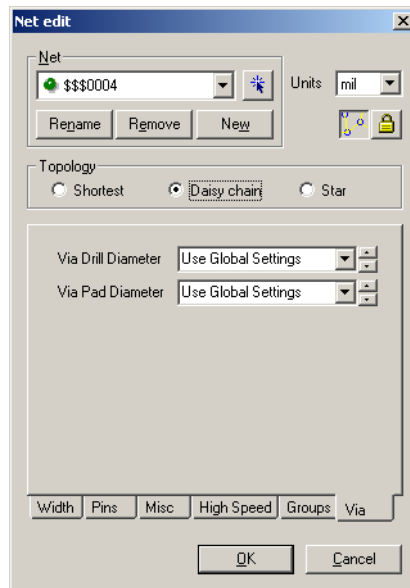
3. Set desired **Via Drill Diameter** and **Via Pad Diameter** and click **OK**.

7.6.3   Highlighting a Net

➢ To highlight a net:

1. Click on a segment of the net that you wish to highlight.

2. Select **Design/Highlight Selected Net**. The entire net is highlighted on the workspace and also in the **Preview** area of the **Nets** tab of the **Spreadsheet View**.

Tip   You can change the highlight color from the **Color Element** drop-down list in the **Colors** tab of the **Preferences** dialog box.

7.6.4   Net Bridges

The net bridge functionality permits connections between different nets (e.g., digital and analog grounds) without losing the properties of either net.

7.6.4.1   Creating a Net Bridge

➢ To create a net bridge:

1. Select **Tools/Database/Database Manager**.

Click here
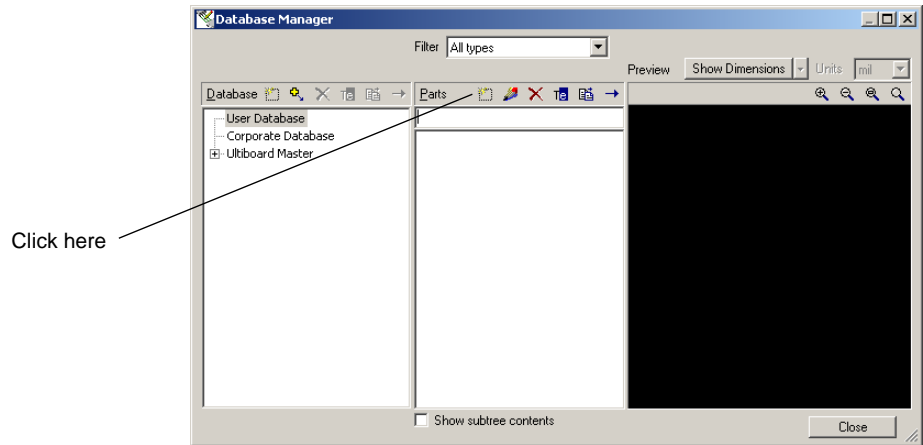
2. Click the **New** button in the **Parts** area to display the following dialog.
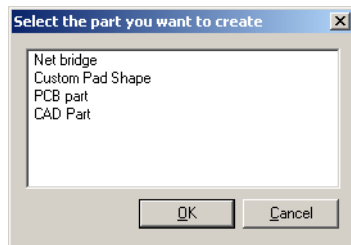
Select **Net bridge** and click **OK**. The **Net Bridge Edit Layer** displays on your workspace.

3. Select **Place/Net Bridge Pins**.
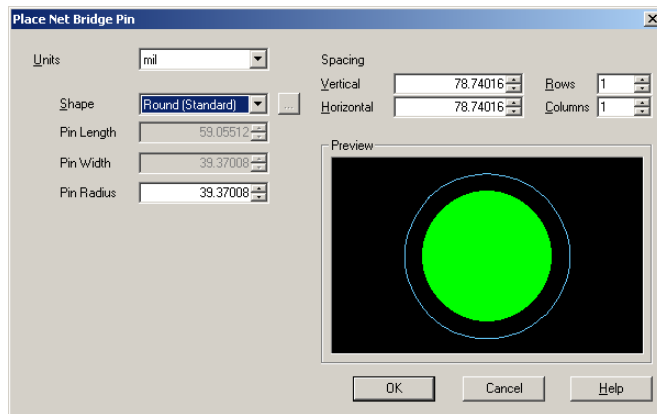
4. Enter the desired parameters for the first pin of the net bridge, click **OK** and place the pin on the workspace.

5. Select **Place/Net Bridge Pins** again, enter the parameters for the second pin of the net bridge, click **OK** and place the pin on the workspace in the desired location.

Net bridge edit layer
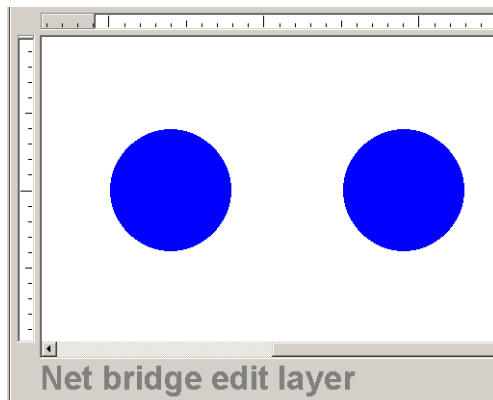
6. Connect the two pins by selecting **Place/Shape/Rectangle** and drawing a rectangle between them, being sure to overlap the two pins.

Net bridge edit layer
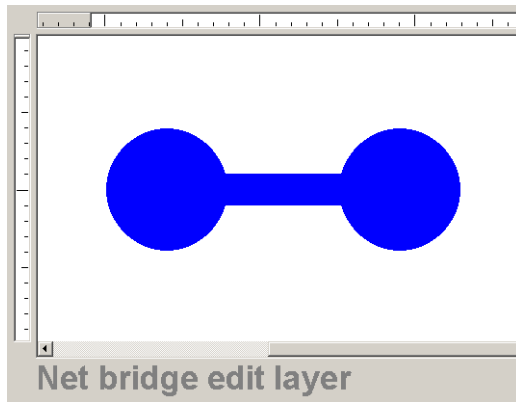
7. Close the **Net Bridge Edit Layer**, and when prompted, save the new net bridge as shown below.
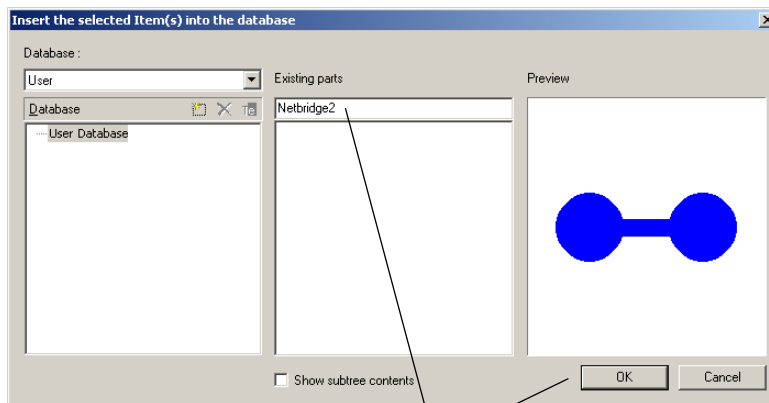
Enter name for new net bridge and click OK.

7.6.4.2 Placing a Net Bridge

This example connects two traces - one is on net "Ground" and the other is on net "GND".

➢ To place a net bridge:

1. Select **Place/Net Bridge**. The **Place Net Bridge** dialog box appears.
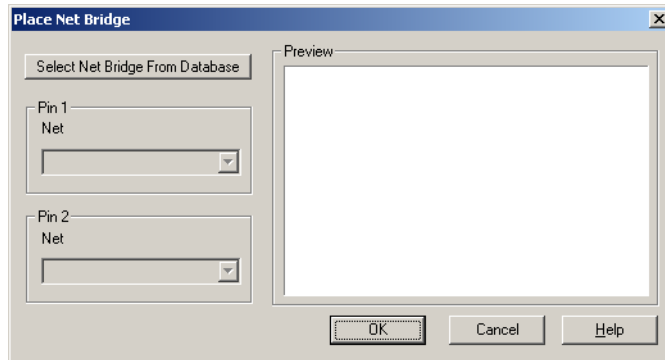
2. Click **Select Net Bridge From Database**. The **Get a part from the database** dialog box appears.

3. Select the desired net bridge in the **Available Parts** area and click **OK**. You are returned to the **Place Net Bridge** dialog box.
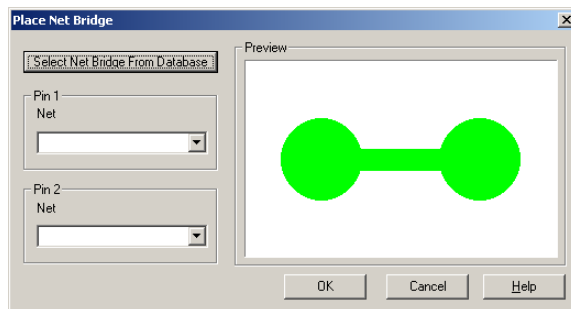
4. Map the pins of the net bridge to the desired nets in the **Pin 1** and **Pin 2** area.
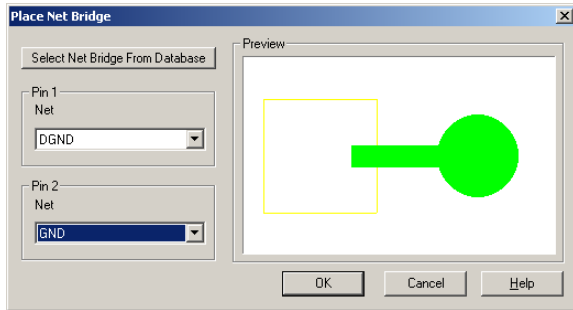
5. Click **OK** and place the netbridge across the two nets as in the example below.
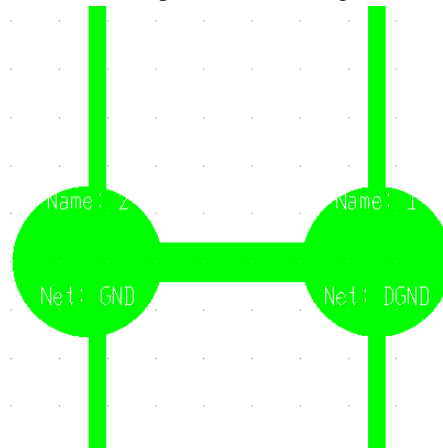
7.6.5   Copying a Copper Route

You can copy the routing of traces between two identical parts groups that have been set up using the **Group Replica** command.

➢ To copy copper routing between groups:

1. Create two identical part groups as detailed in "6.1.3.10 Replicating a Group" on page 6-15.

2. Route the traces for one of the groups.

PartGroupA

PartGroupB
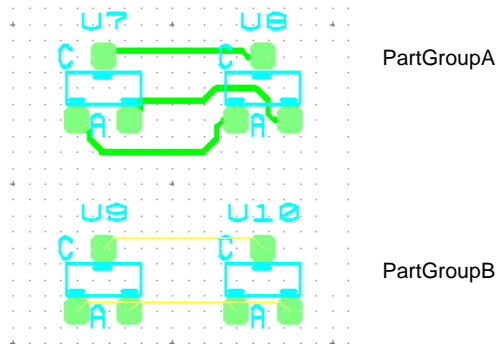
3. Select **Design/Copy route**. The **Copy Route** dialog box appears.
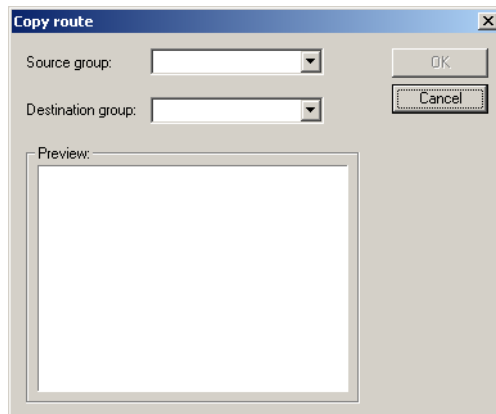
4. Select the group you have already routed in the **Source group** field, and the group you wish to have the same routing in the **Destination group** field and click **OK**. The routing is duplicated for the destination group.

7.7 Swapping Pins and Gates

Pin and gate swapping are done between like pins and gates to reduce the amount of copper needed to route a given net.

The following sections document manual pin swapping, manual gate swapping and automatic pin/gate swapping. For these functions to work, the pin groups must be set up in the **Footprint** tab of the **Component Properties** dialog box in Multisim or Multicap, before the circuit is exported to Ultiboard. Refer to the *Multisim 9 User Guide* or the *Multicap 9 User Guide* for details.

7.7.1   Swapping Pins

This feature works between allowed pins in the same gate (section of an IC). Swapping of pins between gates in the same IC or between similar ICs is not allowed.

The following design is used in this example.

➢ To swap pins between gates:

1. Select **Design/Swap pins**.

2. Click on the first pin that you wish to swap.
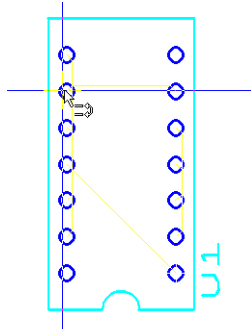
3. Click on the second pin to complete the action.

Note Error messages will display if the selected pins cannot be swapped, or if there is no PINGROUP information for a pin.

7.7.2   Swapping Gates

This feature allows you to swap similar gates, and works for the following which are set in the **Pin & Gate Swapping Settings** area of the **Design Rules** tab of the **PCB properties** dialog box:

•   **Internal Gates Only** — Allows gate swapping in the same IC only.
•   **Advanced Swapping** — Allows gate swapping intenally and between similar ICs.

Group settings for swappable gates can be set into component groups in Ultiboard. Each new component group will have its own swapping information, which Ultiboard will follow.

The following design is used in this example:
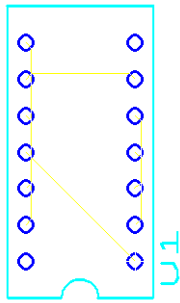
➢   To swap gates between components:

1.   Select **Design/Swap Gates**. The workspace changes to reflect the gates.
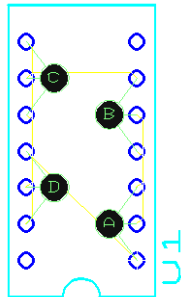
2. Select the first gate that you wish to swap by clicking on the corresponding letter.
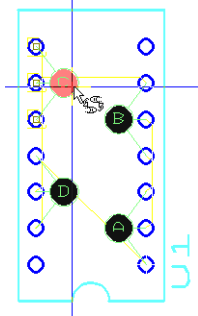
3. Click on the letter corresponding to the gate with which you want to exchange the gate you selected above. The ratsnest changes to reflect the swap.
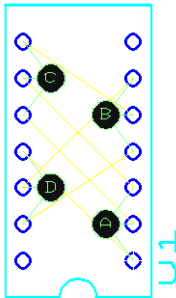
7.7.3   Automatic Pin/Gate Swapping

This feature lets you swap pins and/or gates after moving component(s) on the workspace.

Note   For this feature to function, you must allow pin/gate swapping in the **Spreadsheet View**, and in the **Design Rules** tab of the **PCB properties** dialog box.

➢ To swap pins and gates automatically after a component move:

1. Move desired components on the workspace.

2. Select **Design/Automatic Pin/Gate Swap**. Pins and gates are swapped to achieve the most efficient routing of nets possible.
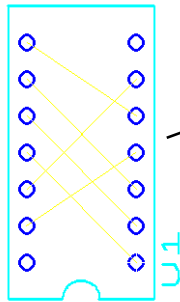
Before the automatic pin/gate swap is done.

Note the position of the ratsnests.
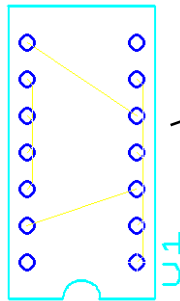
After the automatic pin/gate swap is done.

Note the new position of the ratsnests.

7.7.4   Real-Time Pin/Gate Swapping

This feature allows Ultiboard to swap pins and/or gates automatically in real-time as you move components on the workspace.

Note   For this feature to function, you must allow pin/gate swapping in the **Spreadsheet View**, and allow real-time swapping in the **Design Rules** tab of the **PCB properties** dialog box.

Chapter 8
PCB Calculators

The following are described in this chapter.

8.1   PCB Transmission Line Calculator

To control reflections on high-speed PCBs, it is necessary to make the traces appear as if they are transmission lines. This is done by calculating the characteristic impedance of the trace (Zo) and then terminating it with its characteristic impedance. This makes the trace appear like an infinitely long transmission line, and it will therefore have no reflections, even though in reality it has a finite length. (What actually occurs is that all of the energy that travels down the trace is absorbed, and there is no energy left to reflect back). Once you have calculated Zo, you can use it to design the trace's termination.

Note There are a number of methods used to terminate transmission lines, for example, series termination, diode termination, which are beyond the scope of this guide. We recommend that you refer to any number of available texts on the subject.

In addition to Characteristic Impedance (Zo), you can use the **PCB Transmission Line Calculator** to calculate the following parameters for typical printed circuit board trace geometries:

- Per unit length Capacitance (Co)
- Per unit length Inductance (Lo)
- Propogation Delay (tpd).

The **PCB Transmission Line Calculator** supports:

- Microstrip Trace Calculations
- Embedded Microstrip Trace Calculations
- Centered Stripline Trace Calculations
- Asymmetric Stripline Trace Calculations
- Dual Stripline Trace Calculations.

Note Formulas used are based on the IPC-D-317A document from the IPC organization (www.ipc.org).

8.1.1 Microstrip Trace Calculations

➢ To perform microstrip trace calculations:

1. Select **Tools/PCB Transmission Line Calculator**.

2. Select Microstrip in the **Type** drop-down list.

Trace configuration diagram

3. In the **Input Data** area, edit the following fields as desired.

- **Input Length Unit** — select mils or millimeters

- **Dielectric Thickness (H)** — see the trace configuration diagram.
- **Trace Thickness (T)** — see the trace configuration diagram.
- **Trace Width (W)** — see the trace configuration diagram.
- **Relative Permittivity (epsilon r)** — see the trace configuration diagram.

4. Click **Calculate**. Results of the calculation appear in the **Calculation Results** area.

They also appear in the **Results** tab of the **Spreadsheet View**.

5. Click **Close** to close the **PCB Transmission Line Calculator**.

Microstrip Formulas

The formulas used to perform the microstrip calculations are:

$Z0 = 87/(sqrt(Er + 1.41)) * ln(5.98*H/(0.8*W+T))$

$Tpd = 58.35247*sqrt(Er+1.41)$

$C0 = Tpd/Z0$

$L0 = C0*Z0*Z0$

8.1.2   Embedded Microstrip Trace Calculations

➢ To perform embedded microstrip trace calculations:

1. Select **Tools/PCB Transmission Line Calculator**.

2. Select Embedded Microstrip in the **Type** drop-down list.

Trace configuration diagram

3. In the **Input Data** area, edit the following fields as desired.

- **Input Length Unit** — select mils or millimeters
- **Dielectric Height (H1)** — see the trace configuration diagram.
- **Dielectric Thickness (H)** — see the trace configuration diagram.
- **Trace Thickness (T)** — see the trace configuration diagram.
- **Trace Width (W)** — see the trace configuration diagram.
- **Relative Permittivity (epsilon r)** — see the trace configuration diagram.

4. Click **Calculate**. Results of the calculation appear in the **Calculation Results** area.

They also appear in the **Results** tab of the **Spreadsheet View**.

5. Click **Close** to close the **PCB Transmission Line Calculator**.

Embedded Microstrip Formulas

The formulas used to perform the embedded microstrip calculations are:

Z0 = 56*ln(5.98*H/(0.8*W+T))/sqrt(Er*(1-exp(-1.55*H1/H)))

Tpd = 84.66667*sqrt(Er*(1-exp(-1.55*H1/H)))

C0 = Tpd/Z0

L0 = C0*Z0*Z0

8.1.3 Centered Stripline Trace Calculations

➢ To perform centered stripline trace calculations:

1. Select **Tools/PCB Transmission Line Calculator**.

2. Select Centered Stripline in the **Type** drop-down list.

Trace configuration diagram

3. In the **Input Data** area, edit the following fields as desired.

- **Input Length Unit** — select mils or millimeters
- **Dielectric Thickness (H)** — see the trace configuration diagram.
- **Trace Thickness (T)** — see the trace configuration diagram.
- **Trace Width (W)** — see the trace configuration diagram.
- **Relative Permittivity (epsilon r)** — see the trace configuration diagram.

4. Click **Calculate**. Results of the calculation appear in the **Calculation Results** area.

They also appear in the **Results** tab of the **Spreadsheet View**.

5. Click **Close** to close the **PCB Transmission Line Calculator**.

Centered Stripline Formulas

The formulas used to perform the centered stripline calculations are:

Z0 = 60*ln(4*(2*H+T)/(0.67*3.1415926*(0.8*W+T)))/sqrt(Er)

Tpd = 84.66667*sqrt(Er)

C0 = Tpd/Z0

L0 = C0*Z0*Z0

8.1.4 Asymmetric Stripline Trace Calculations

➢ To perform asymmetric stripline trace calculations:

1. Select **Tools/PCB Transmission Line Calculator**.

2. Select Asymmetric Stripline in the **Type** drop-down list.

Trace configuration diagram

3. In the **Input Data** area, edit the following fields as desired.

   • **Input Length Unit** — select mils or millimeters
   • **Dielectric Height (H1)** — see the trace configuration diagram.
   • **Dielectric Height (H)** — see the trace configuration diagram.
   • **Trace Thickness (T)** — see the trace configuration diagram.
   • **Trace Width (W)** — see the trace configuration diagram.
   • **Relative Permittivity (epsilon r)** — see the trace configuration diagram.

4. Click **Calculate**. Results of the calculation appear in the **Calculation Results** area.

They also appear in the **Results** tab of the **Spreadsheet View**.

5. Click **Close** to close the **PCB Transmission Line Calculator**.

Assymetric Stripline Formulas

The formulas used to perform the asymmetric stripline calculations are:

Z0 = (1-H/(4*H1))*80*ln(4*(2*H+T)/(0.67*3.1415926*(0.8*W+T)))/sqrt(Er)

Tpd = 84.66667*sqrt(Er)

C0 = Tpd/Z0

L0 = C0*Z0*Z0

8.1.5   Dual Stripline Trace Calculations

➢ To perform centered stripline trace calculations:

1. Select **Tools/PCB Transmission Line Calculator**.

2. Select Dual Stripline in the **Type** drop-down list.

Trace configuration diagram

3. In the **Input Data** area, edit the following fields as desired.

- **Input Length Unit** — select mils or millimeters
- **Trace Separation (C)** — see the trace configuration diagram.
- **Dielectric Thickness (H)** — see the trace configuration diagram.
- **Trace Thickness (T)** — see the trace configuration diagram.
- **Trace Width (W)** — see the trace configuration diagram.
- **Relative Permittivity (epsilon r)** — see the trace configuration diagram.

4. Click **Calculate**. Results of the calculation appear in the **Calculation Results** area.

They also appear in the **Results** tab of the **Spreadsheet View**.

5. Click **Close** to close the **PCB Transmission Line Calculator**.

Dual Stripline Formulas

The formulas used to perform the dual stripline calculations are:

Z0 = 30*(ln(8*H/(0.67*3.1415926*(0.8*W+T)))+
     ln(8*(H+C)/(0.67*3.1415926*(0.8*W+T))))/sqrt(Er)

Tpd = 84.66667*sqrt(Er)

C0 = Tpd/Z0

L0 = C0*Z0*Z0

8.2 PCB Differential Impedance Calculator

To control reflections on high-speed PCBs, it is necessary to make the traces appear as if they are transmission lines. This is done by calculating the characteristic impedance of the trace (Zo) and then terminating it with its characteristic impedance. This makes the trace appear like an infinitely long transmission line, and it will therefore have no reflections, even though in reality it has a finite length. (What actually occurs is that all of the energy that travels down the trace is absorbed, and there is no energy left to reflect back). Once you have calculated Zo, you can use it to design the trace's termination.

If two traces in a differential pair are placed closely together, the differential impedance (Zdiff) of the pair must be calculated for proper trace termination. (This is the Differential Impedance Rule).

Note There are a number of methods used to terminate transmission lines, for example, series termination, diode termination, which are beyond the scope of this guide. We recommend that you refer to any number of available texts on the subject.

The **PCB Differential Impedance Calculator** performs calculations for two traces that carry signals that are exactly equal and opposite (a differential pair).

You can use the **PCB Differential Impedance Calculator** to calculate the following parameters for differential pairs:

- Characteristic Impedance (Zo)
- Per unit length Capacitance (Co)
- Per unit length Inductance (Lo)
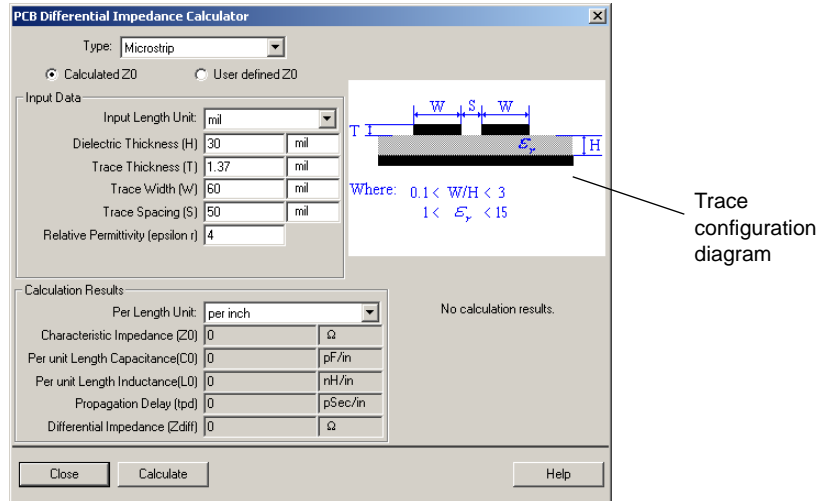- Propogation Delay (tpd)
- Differential Impedance (Zdiff).

The **PCB Differential Impedance Calculator** supports:

- Microstrip Calculations
- Embedded Microstrip Calculations
- Centered Stripline Calculations
- Asymmetric Stripline Calculations.

Note Formulas used are based on the IPC-D-317A document from the IPC organization (www.ipc.org).

8.2.1   Microstrip Calculations

➢ To perform microstrip differential impedance calculations:

1. Select **Tools/PCB Differential Impedance Calculator**.

2. Select Microstrip in the **Type** drop-down list.

Trace configuration diagram

3. In the **Input Data** area, edit the following fields as desired.

- **Input Length Unit** — select mils or millimeters
- **Dielectric Thickness (H)** — see the trace configuration diagram.
- **Trace Thickness (T)** — see the trace configuration diagram.
- **Trace Width (W)** — see the trace configuration diagram.
- **Trace Spacing (S)** — see the trace configuration diagram.
- **Relative Permittivity (epsilon r)** — see the trace configuration diagram.

Or

If you wish to define the Characteristic Impedance (Zo) yourself, click **User Defined Zo** and edit the following fields as desired:

- **Input Length Unit** — select mils or millimeters
- **Dielectric Thickness (H)** — see the trace configuration diagram.
- **Trace Spacing (S)** — see the trace configuration diagram.
- **Characteristic Impedance (Zo)** — see the trace configuration diagram.

4. Click **Calculate**. Results of the calculation appear in the **Calculation Results** area.

They also appear in the **Results** tab of the **Spreadsheet View**.

Note If you chose **User Defined Zo** in the previous step, the **Per Length Unit** and the **Differential Impedance** are the only values that appear in the **Calculation Results** area of the **PCB Differential Impedance Calculator** dialog and the **Results** tab when you click **Calculate**.

5. Click **Close** to close the **PCB Differential Impedance Calculator**.

Microstrip Differential Impedance Formulas

The formulas used to perform the microstrip differential impedance calculations are:

Z0 = 87/(sqrt(Er + 1.41)) * ln(5.98*H/(0.8*W+T))

Tpd = 58.35247*sqrt(Er+1.41)

C0 = Tpd/Z0

L0 = C0*Z0*Z0

Zdiff = 2*Z0*(1-0.48*exp(-0.96*S/H))

8.2.2   Embedded Microstrip Calculations

➢ To perform embedded microstrip differential impedance calculations:

1. Select **Tools/PCB Differential Impedance Calculator**.

2. Select Embedded Microstrip in the **Type** drop-down list.

Trace configuration diagram

3. In the **Input Data** area, edit the following fields as desired.

- **Input Length Unit** — select mils or millimeters
- **Dielectric Height (H1)** — see the trace configuration diagram.
- **Dielectric Thickness (H)** — see the trace configuration diagram.
- **Trace Thickness (T)** — see the trace configuration diagram.
- **Trace Width (W)** — see the trace configuration diagram.
- **Trace Spacing (S)** — see the trace configuration diagram.
- **Relative Permittivity (epsilon r)** — see the trace configuration diagram.

Or

If you wish to define the Characteristic Impedance (Zo) yourself, click **User Defined Zo** and edit the following fields as desired:

- **Input Length Unit** — select mils or millimeters
- **Dielectric Thickness (H)** — see the trace configuration diagram.
- **Trace Spacing (S)** — see the trace configuration diagram.
- **Characteristic Impedance (Zo)** — see the trace configuration diagram.

4. Click **Calculate**. Results of the calculation appear in the **Calculation Results** area.

They also appear in the **Results** tab of the **Spreadsheet View**.

Note If you chose **User Defined Zo** in the previous step, the **Per Length Unit** and the **Differential Impedance** are the only values that appear in the **Calculation Results** area of the **PCB Differential Impedance Calculator** dialog and the **Results** tab when you click **Calculate**.

5. Click **Close** to close the **PCB Differential Impedance Calculator**.

Embedded Microstrip Differential Impedance Formulas

The formulas used to perform the embedded microstrip differential impedance calculations are:

Z0 = 56*ln(5.98*H/(0.8*W+T))/sqrt(Er*(1-exp(-1.55*H1/H)))

Tpd = 84.66667*sqrt(Er*(1-exp(-1.55*H1/H)))

C0 = Tpd/Z0

L0 = C0*Z0*Z0

Zdiff = 2*Z0*(1-0.48*exp(-0.96*S/H1))

8.2.3 Centered Stripline Calculations

➢ To perform centered stripline differential impedance calculations:

1. Select **Tools/PCB Differential Impedance Calculator**.

2. Select Centered Stripline in the **Type** drop-down list.

Trace configuration diagram

3. In the **Input Data** area, edit the following fields as desired.

- **Input Length Unit** — select mils or millimeters.
- **Dielectric Thickness (H)** — see the trace configuration diagram.
- **Trace Thickness (T)** — see the trace configuration diagram.
- **Trace Width (W)** — see the trace configuration diagram.

- **Trace Spacing (S)** — see the trace configuration diagram.
- **Relative Permittivity (epsilon r)** — see the trace configuration diagram.

Or

If you wish to define the Characteristic Impedance (Zo) yourself, click **User Defined Zo** and edit the following fields as desired:

- **Input Length Unit** — select mils or millimeters
- **Dielectric Thickness (H)** — see the trace configuration diagram.
- **Trace Spacing (S)** — see the trace configuration diagram.
- **Characteristic Impedance (Zo)** — see the trace configuration diagram.

4. Click **Calculate**. Results of the calculation appear in the **Calculation Results** area.

They also appear in the **Results** tab of the **Spreadsheet View**.

Note If you chose **User Defined Zo** in the previous step, the **Per Length Unit** and the **Differential Impedance** are the only values that appear in the **Calculation Results** area of

the **PCB Differential Impedance Calculator** dialog and the **Results** tab when you click **Calculate**.

5. Click **Close** to close the **PCB Differential Impedance Calculator**.

Centered Stripline Differential Impedance Formulas

The formulas used to perform the centered stripline differential impedance calculations are:

$Z0 = 60*\ln(4*(2*H+T)/(0.67*3.1415926*(0.8*W+T)))/\text{sqrt}(Er)$

$Tpd = 84.66667*\text{sqrt}(Er)$

$C0 = Tpd/Z0$

$L0 = C0*Z0*Z0$

$Zdiff = 2*Z0*(1-0.347*\exp(-2.9*S/(2*H+T)))$

8.2.4 Asymmetric Stripline Calculations

➢ To perform asymmetric stripline differential impedance calculations:

1. Select **Tools/PCB Differential Impedance Calculator**.

2. Select Asymmetric Stripline in the **Type** drop-down list.

Trace configuration diagram

3. In the **Input Data** area, edit the following fields as desired.

- **Input Length Unit** — select mils or millimeters.
- **Dielectric Height (H1)** — see the trace configuration diagram.
- **Dielectric Height (H)** — see the trace configuration diagram.

- **Trace Thickness (T)** — see the trace configuration diagram.
- **Trace Width (W)** — see the trace configuration diagram.
- **Trace Spacing (S)** — see the trace configuration diagram.
- **Relative Permittivity (epsilon r)** — see the trace configuration diagram.

Or

If you wish to define the Characteristic Impedance (Zo) yourself, click **User Defined Zo** and edit the following fields as desired:

- **Input Length Unit** — select mils or millimeters
- **Dielectric Thickness (H)** — see the trace configuration diagram.
- **Trace Spacing (S)** — see the trace configuration diagram.
- **Characteristic Impedance (Zo)** — see the trace configuration diagram.

4. Click **Calculate**. Results of the calculation appear in the **Calculation Results** area.

They also appear in the **Results** tab of the **Spreadsheet View**.

Note If you chose **User Defined Zo** in the previous step, the **Per Length Unit** and the **Differential Impedance** are the only values that appear in the **Calculation Results** area of the **PCB Differential Impedance Calculator** dialog and the **Results** tab when you click **Calculate**.

5. Click **Close** to close the **PCB Differential Impedance Calculator**.

Asymmetric Stripline Differential Impedance Formulas

The formulas used to perform the asymmetric stripline differential impedance calculations are:

Z0 = (1-H/(4*H1))*80*ln(4*(2*H+T)/(0.67*3.1415926*(0.8*W+T)))/sqrt(Er)

Tpd = 84.66667*sqrt(Er)

C0 = Tpd/Z0

L0 = C0*Z0*Z0

Zdiff = 2*Z0*(1-0.347*exp(-2.9*S/(H+H1+T)))

Chapter 9
Internal Router

This chapter describes how to automatically route traces by using the internal router included with Ultiboard.

The following are described in this chapter.

Subject	Page No.
Using the Internal Router	9-1

9.1 Using the Internal Router

If you use thicker traces for power and ground, we suggest that you manually pre-place and lock these traces before using the internal router.

➢ To autoroute the traces:

1. Select **Autoroute/Start Internal Router**.

   Internal Router Mode displays and the traces are routed. When routing is complete, **Internal Router Mode** closes and the routed design appears on the Ultiboard workspace.

   Routing statistics appear in the **Results** tab of the **Spreadsheet View**.

   ```
   Ultiboard - 2005-10-14 12:39:09

   Internal router [Quick3] 2005-10-14 12:39:13
   Routing completed successfully.
   49 of 49 connections (100%) routed with 21 vias.  Time: 0:01
   ```
   Results
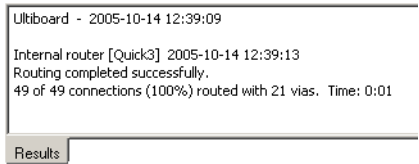
Chapter   10
Preparing for Manufacturing/Assembly

This chapter explains the basic functions you need to perform to prepare your board for manufacturing.

The following are described in this chapter.

10.1   Placing and Editing Text

Text can be placed anywhere on the design and on any layer, regardless of what element is selected.

➢   To place text on the design:

1.   Choose **Place/Graphics/Text**. The **Text** dialog box opens:

Type the text to be placed

Select what you wish to display

Preview the text and adjust its alignment

Select the height of the text

Enable to mirror the text (show it backwards

Set the thickness of the text

Enable if you wish to use a Windows font. See note below

Select a layer for the text

Select a color for the text
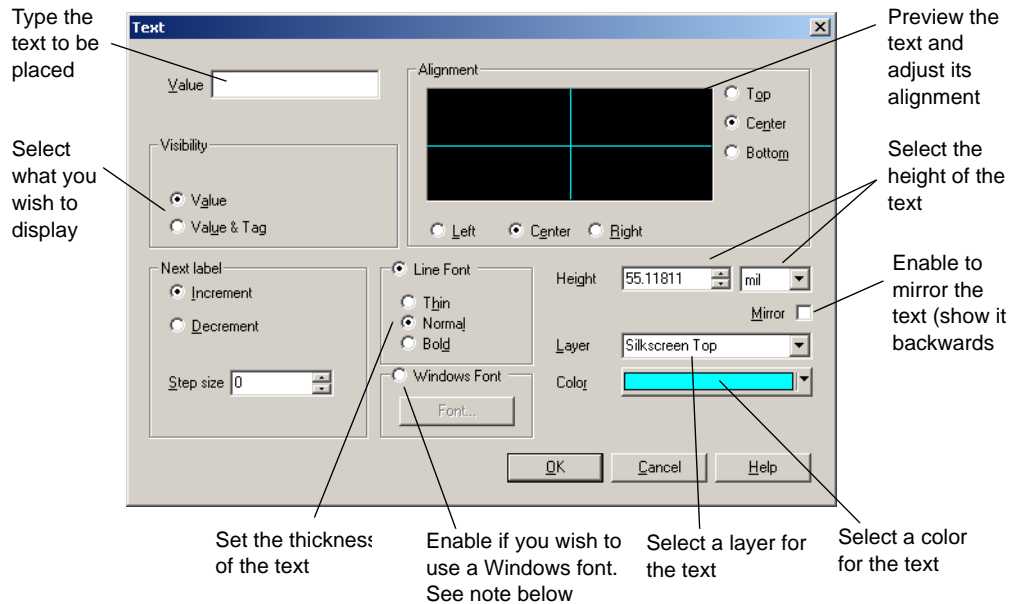
Note   Ultiboard supports multi-byte fonts so that Chinese, Japanese, Korean and other users can make full use of Microsoft fonts for special characters.

2.   Type the text in the **Value** field. As you type, the text appears in the preview dialog box.

3.   Define the other parameters for the text as shown in the diagram in step 1. Your choices are reflected in the preview dialog box.

4.   Optionally, in the **Next Label** area:

- **Increment** — enable to increase a number that you placed at the end of the text with each successive placement of that text. You must also enter a value in **Step Size**. For example, if you enable this checkbox and enter "1" in **Step Size**, and the **Value** you entered was "Resistor1", the first placement of the text will say "Resistor1", the second will say "Resistor2", the third will say "Resistor3", and so on.
- **Decrement** — similar to the **Increment** setting, except that the number will decrease by the **Step Size** with each successive placement. (The number will not go below 0).

5. Click **OK**. The **Text** dialog box disappears; the cursor now has your text on it.

6. Move the cursor where you want the text placed, and left-click to drop the text.

7. Right-click to cancel the **Place/Graphics/Text** command.

➢ To edit text:

1. Double-click the text. The **Copper Attribute Properties** dialog box opens.

2. Click on the **Attribute** tab.

3. Edit the text.

4. Click **OK**. The **Copper Attribute Properties** dialog box disappears and your changes are applied to the text in the design.

10.2 Capturing Screen Area

You can capture an area of the screen and then manipulate the image as you would any other screen capture contained in the system clipboard.

➢ To copy a section of your screen to the clipboard:

1. Select **Tools/Capture Screen Area**. A selection frame appears on your workspace.
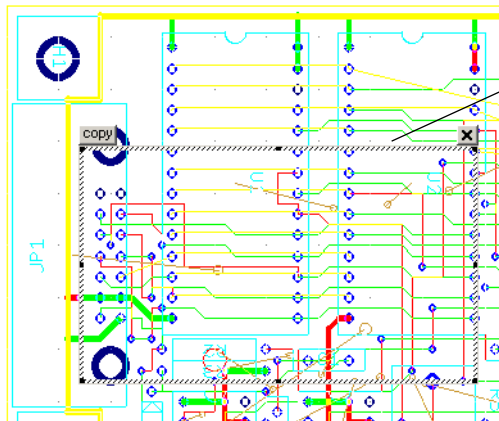
Area within the selection frame will be copied to clipboard when **copy** is clicked.

2. To move the frame to a different location:

- Move your cursor to the border of the frame. A crosshair is added to the cursor.

Do *not* select sizing handles to move the selection frame.

A crosshair is added to the cursor, indicating that the selection frame can be moved to a different location.
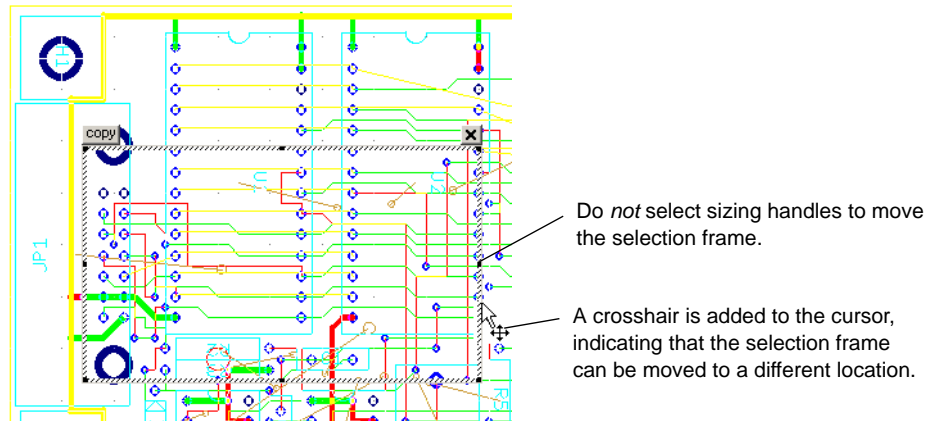
- Drag the selection frame to the desired location.

3. To re-size the selection frame:

- Move the cursor to one of the sizing handles.

Sizing handles

Cursor switches to line with arrows.
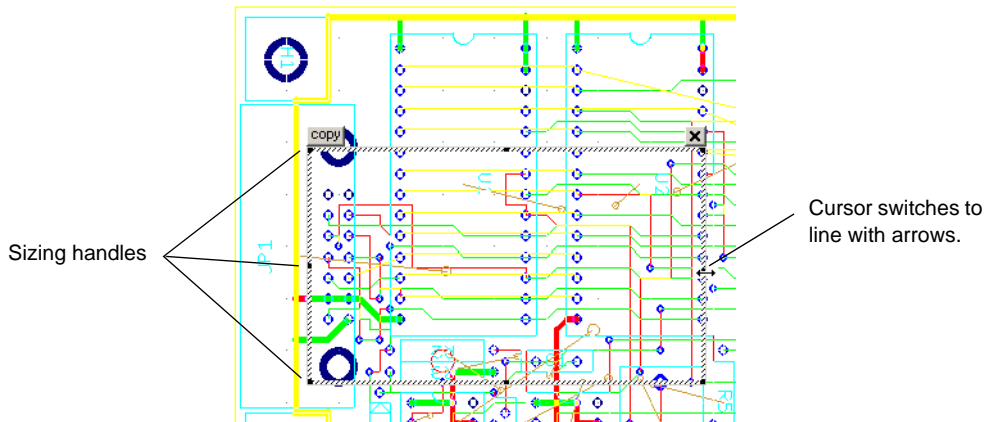
- Drag the cursor to re-size the selection frame.

4. Click on the **copy** button at the top left corner of the selection frame. The image inside the selection frame is copied to the system clipboard.

5. Click on the **x** at the top right corner of the selection frame to close it.

6. Open desired application, for example, Word, and click on Paste to paste the image.

10.3 Placing a Comment

Adding a comment permits "redlining", which can be used to show engineering change orders, to facilitate collaborative work among team members, or to allow background information to be attached to a design.

You can "pin" a comment to the workspace, or directly to a component. When a component with an attached comment is moved, the comment also moves.

Comment pinned to a component with its contents displayed.

John (2005-05-14):
There are 2 occurences
of this component.
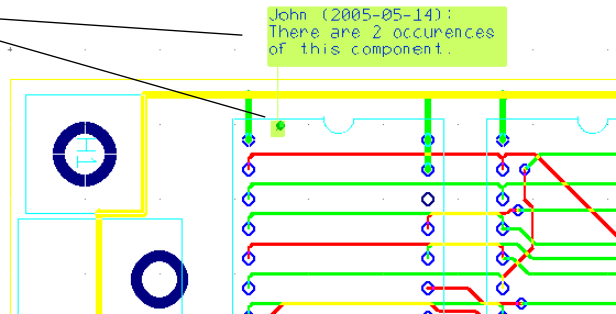
➢ To pin a comment to a component or the workspace:

1. Double-click on the **Comment** layer in the **Design Toolbox** to make it the active layer.

2. Select **Place/Comment**. The **Comment** dialog box appears.

Enable to show the contents of the comment on the design.

Type the comment in this area.
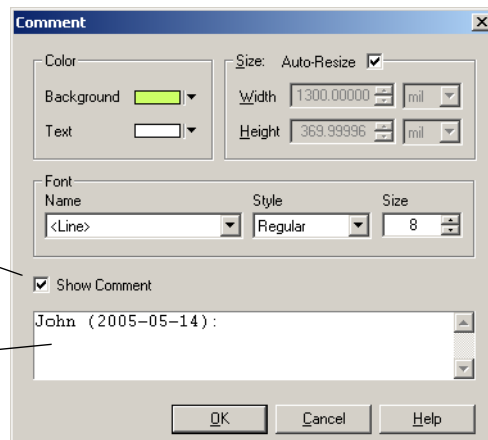
3. Type the comment in the area indicated above.

4. Optionally, modify the information in the **Color** area:

   • **Background** — click to change the color of the placed comment's background.

- **Text** — click to change the color of the placed comment's text.

When you click either **Background** or **Text**, the **Select color** dialog box displays. Change the colors as shown below and click **OK** to return to the **Comment** dialog box.
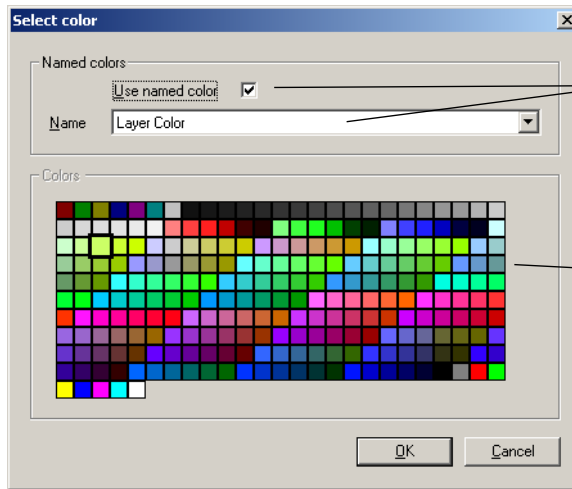
Enable checkbox if you want the color to be the same as the element selected in the Name drop-down list.

Or

Disable the checkbox and select the color from this palette.

5. In the **Size** area:
   - **Auto-Resize** checkbox — enable to have the size of the displayed comment automatically resized to fit the text. Disable if you wish to set the size of the displayed comment by typing values into the **Width and Height** fields.
6. In the **Font** area, set the font **Name**, **Style** and **Size** as desired.
7. Click **OK** to close the dialog and double-click at the desired location on the design to place the comment.

➢ To change a placed comment's properties, select the comment (you must be in the **Comment** layer), and select **Edit/Properties**.

➢ To delete a placed comment, select the comment and press DELETE on your keyboard.

10.4   Renumbering Parts

As parts are added to, moved, and deleted from the design, their numbering changes. Renumbering components automatically renames all components in the order that you specify. It is easier to produce, service, and troubleshoot boards when components are ordered in a logical manner. Renumbering is primarily for assembly, to help locate all the elements.

Using the **Renumber Components** dialog box, you can select the corner of the board in which you want the renumbering to start, and whether you want the components to be renumbered

incrementally in a vertical or horizontal fashion. You can preview your renumbering strategy and change it before applying it.

➢ To renumber the parts in the design:

1. Optionally, select the parts you want renumbered.

2. Choose **Tools/Renumber Footprints**. The **Renumber Components** dialog box appears.
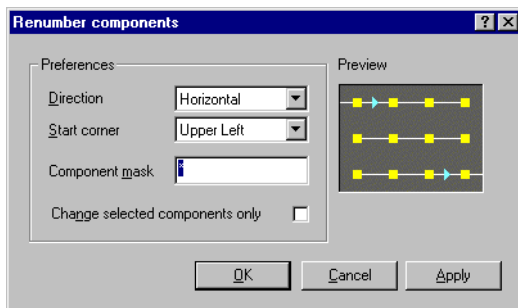
3. Use the drop-down lists to select the **Direction** and **Start corner** for the renumbering and a value in **Component mask** if you want to renumber only certain elements. For example, you can put R* or C* in that box if you only want to renumber resistors or only capacitors; the default = * so everything gets renumbered.

   The **Preview** panel illustrates the numbering's direction and start corner as you define the settings.

4. If you only want to renumber parts you selected prior to opening this dialog box, select the **Change selected components only** option.

5. To apply your changes and keep the dialog box open, click **Apply**. To apply your changes and close the dialog box, click **OK**. In either case, you are prompted to save the design file with the changes. To cancel your changes, click **Cancel**.

10.5 Backannotation to Multisim/Multicap

Backannotation is a highly automated process which ensures that modifications made to an Ultiboard design are transferred to the board's schematic in Multisim. This process helps keep your schematics and board layouts consistent with one another.

Backannotation is an important feature of CAD software. Component renaming or removing cause inconsistencies between the schematic and the PCB design. Backannotation can overcome these inconsistencies. To backannotate, Multisim reads the log file in which Ultiboard reports all the changes that are made to a PCB. The log file has the same name as the project, but with the extension `.log`.

Not all changes that are made to the PCB can be backannotated to Multisim. The following changes can be backannotated:

- component removing
- component renaming
- netlist renaming

➢ To backannotate your revisions:

1. Save and close your design in Ultiboard.

2. Open Multisim.

3. Follow the Back Annotation procedure documented in the *Multisim 9 User Guide*.

10.6   Mitering Corners

Corner mitering is used to reduce or remove sharp angles for placed traces by creating 135° angles in their place. This is important for manufacturing purposes. You can apply corner mitering to the entire design or just the currently selected traces.

➢ To miter the corners of traces prior to manufacturing:

1. Optionally, select the traces to which you want mitering to apply.

2. Choose **Design/Corner Mitering**. The **Corner Mitering** dialog box appears.
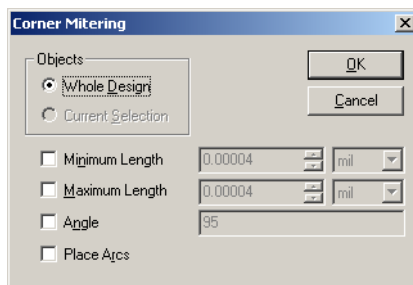
3. To apply the changes to just the selected traces, enable the **Current Selection** option. To apply the changes to the whole design, enable the **Whole Design** option.

4. To set a minimum length for the mitering, enable the **Minimum Length** option and enter a length and units of measurement.

   Any corner that is to be mitered will have two trace segments forming the corner. The **Minimum Length** refers to the shortest length of a segment that should be mitered. For example, say a corner is formed from a 6 mm segment and an 8 mm segment. A minimum length setting greater than 2 mm will cause the trace to not be mitered.

The default is 0, that is, all traces will be mitered regardless of the minimum length of any one segment.

5. To set a maximum length for the mitering, enable the **Maximum Length** option and enter a length and units of measurement.

The **Maximum Length** setting refers to the maximum length of the mitered segment. The longest length of the component of the mitered segment in the horizontal or vertical direction will not exceed the **Maximum Length** or one third of the shortest segment. Using the example of a corner formed from a 6 mm segment and an 8 mm segment, a **Maximum Length** setting of 3 mm will cause the trace to be mitered to 2 mm (i.e. one third of 6 mm).

The default is 0, that is, all traces will be mitered regardless of the maximum length of any one segment.

6. To set the angle of corners to be affected by mitering, enable the **Angle** option and enter a value. For example, a setting of 95° will mean that all angles less than 95° will be mitered to 135°.

7. To have an arc placed when doing the mitering, enable the **Place Arcs** option.

8. To save your changes and miter the corners, click **OK**. To cancel them, click **Cancel**.

10.7 Manually Re-Running the Design Rules and Netlist Check

The design rules and netlist check normally runs automatically, but you may want to force a final check of the board's integrity prior to saving or exporting the design.

➢ To do this, select **Design/Netlist and DRC Check**.

10.8 Cleaning up the Board

Before sending the board for manufacturing, you should remove any open trace ends and unused vias that have been left on the board.

10.8.1 Deleting Open Trace Ends

Open trace ends are trace segments that do not have any terminating connections in the design.

➢ To delete open trace ends, make sure the design is open and choose **Edit/Copper Delete/Open Trace Ends.** This deletes all open trace ends in the design.

10.8.2 Deleting Unused Vias

Normally, you would delete unused vias after deleting any open trace ends.

➢ To delete any unused vias, make sure the design is open and choose **Design/Clean Unused Vias** to delete all vias that do not have any trace segments or copper areas connected to them.

10.9   Exporting a File

Exporting a file refers to producing an output from Ultiboard in a format that can be understood by the equipment at the board manufacturer. An exported file contains complete information describing how a finished board is to be manufactured. There are many different manufacturing techniques used to produce printed circuit boards and Ultiboard can produce a wide variety of outputs to meet these needs.

It is important to talk to your production house and identify all the files and formatting information they need to support their manufacturing process.

You can export a file in the following formats.
- Gerber photoplotter 274X or 274D
- DXF
- 3D DXF (see "11.3 Exporting to 3D DXF" on page 11-7)
- 3D IGES (see "11.4 Exporting to 3D IGES" on page 11-8)
- IPC-D-356A Netlist (see "10.9.3 Exporting the Desired File" on page 10-19)
- NC drill
- SVG (Scalable Vector Graphics)

You can also export text files that contain:
- Board Statistics
- Part Centroids
- Bill of Materials

You can also create reports on:
- Copper Amounts
- Test Points
- Layer Stackup

Exporting a file begins by opening the **Export** dialog box. You can also use the **Export** dialog box to create and delete export settings, and to view and edit the properties of the export settings.

10.9.1 Using Export Settings

Export settings are useful for establishing different parameters as required by different manufacturers or for different purposes (e.g. to export only copper layers, or mechanical drawings).

➢ To create a new export setting:

1. Choose **File/Export**. The **Export** dialog box appears.
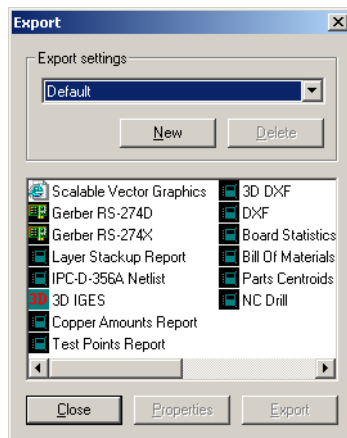
2. Click **New**. The **New settings** dialog box appears.

3. Enter the new name and click **OK**. The **New settings** dialog box disappears and the new setting is displayed in the **Export settings** drop-down list.

The new setting uses the same properties as the Default setting, or the setting that was last loaded. To change the properties stored in the new setting, see "10.9.2 Viewing and Editing Export Properties" on page 10-12.

➢ To delete an export setting:

1. Choose **File/Export**. The **Export** dialog box appears.

2. Select the setting to be deleted from the **Export settings** drop-down list.

3. Click **Delete**. The setting is deleted from the **Export settings** drop-down list.

10.9.2 Viewing and Editing Export Properties

Properties can be viewed and edited for each device or type of export. This is done through the property dialog boxes that correspond to the devices or types listed in the **Export** dialog box:

• Gerber RS-274D
• Gerber RS-274X
• DXF
• Board Statistics
• Bill of Materials
• Parts Centroids
• NC Drill

➢ To display the properties dialog box for a type of export:

1. Select the item in the list displayed in the **Export** dialog box.

2. Click **Properties**. The item's property dialog box appears.

10.9.2.1 Setting Gerber Properties

The **Gerber properties** (RS-274X or RS-274D) dialog box allows you to select the layers to be exported, the number of digits in numerals, and the kind of measurements:

Click to move the selected layer to the Export Layers side

Export and Merged Layers

Select one or more export layers and click to merge/unmerge

Layers available to be exported

Choose the kind of units of measurement

Define the number of integers and decimal places

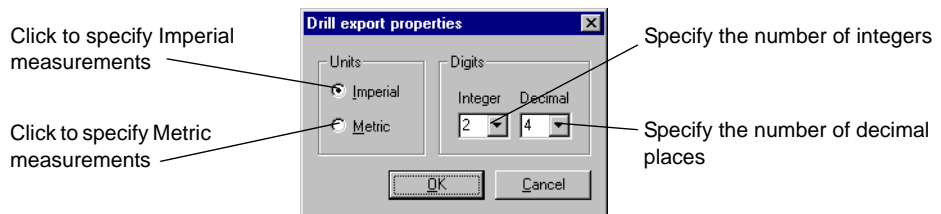Specify oversize parameters for solder mask and paste.

Specify export options
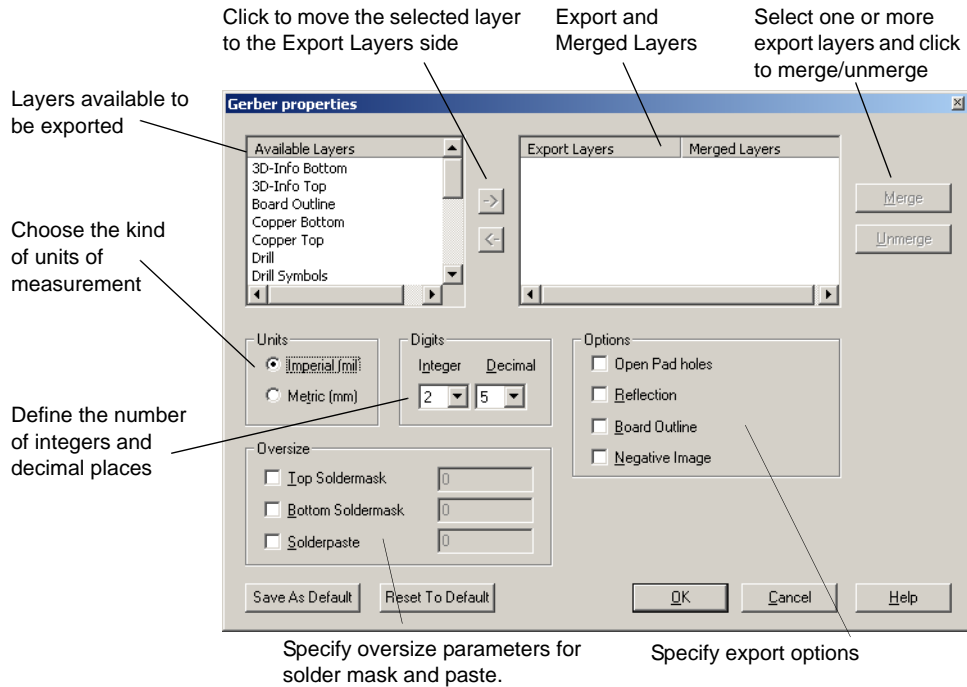
10.9.2.2 Setting DXF Properties

The **DXF export settings** dialog box allows you to select the layers to be exported and the units of measurement to be used.

Layers available to be exported

Choose the units of measurement

Click to move the selected layer to the Layers to Process side

Layers to process appear here
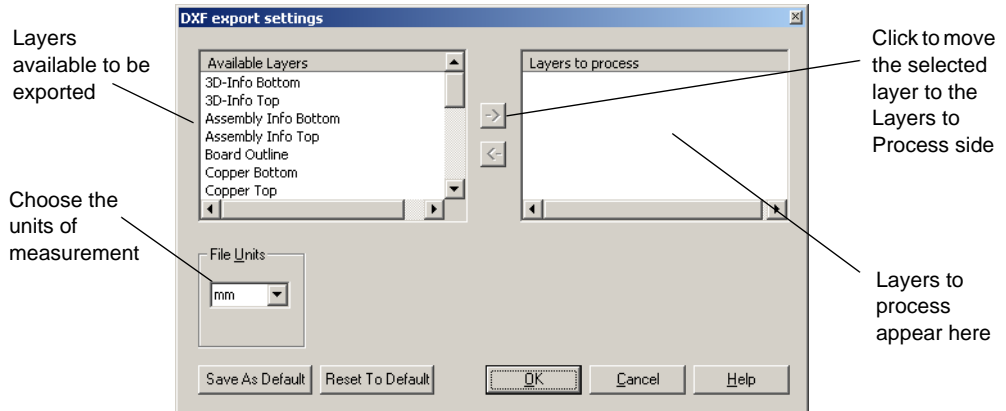
10.9.2.3 Setting NC Drill Properties

The **Drill export properties** dialog box allows you to specify measurement units, and to specify the number of digits for integers and decimals:

Click to specify Imperial measurements

Click to specify Metric measurements

Specify the number of integers

Specify the number of decimal places

10.9.2.4 Working with SVG Properties

The **SVG Export Properteis** dialog box allows you to specifiy whether Scaleable Vector Graphic (SVG) files will be compressed or not when they are exported.
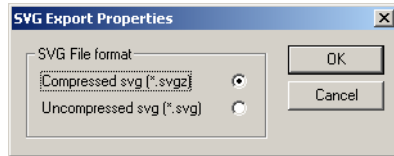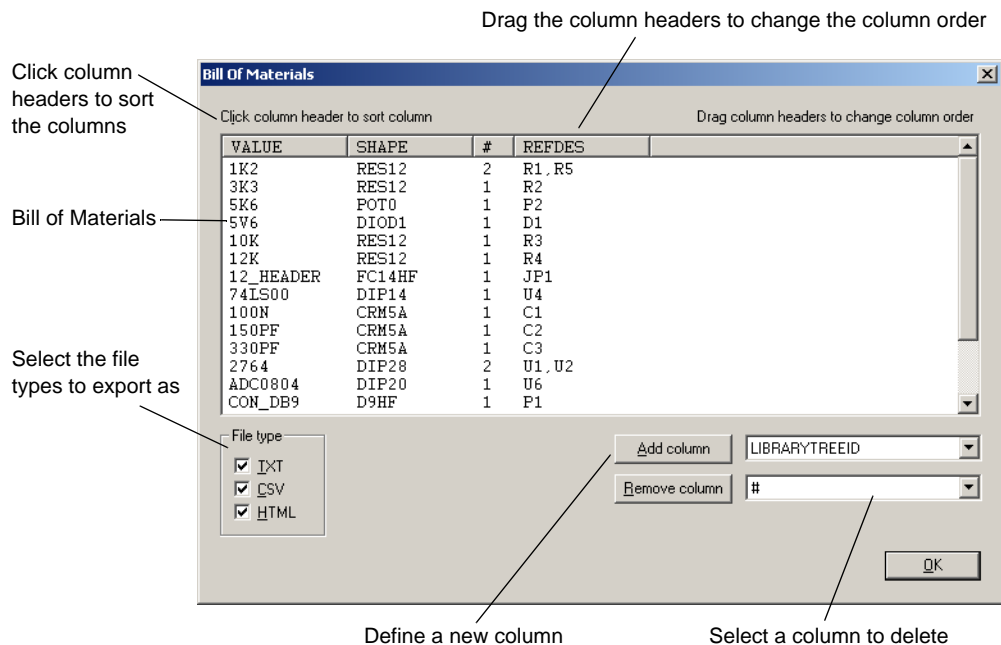

10.9.2.5 Working with other Properties

Working with Board Statistics Properties

The **Board Statistics** dialog box allows you to view the statistics on the board being exported as well as to filter the file types to be exported and to define the units of measurement in the statistics:
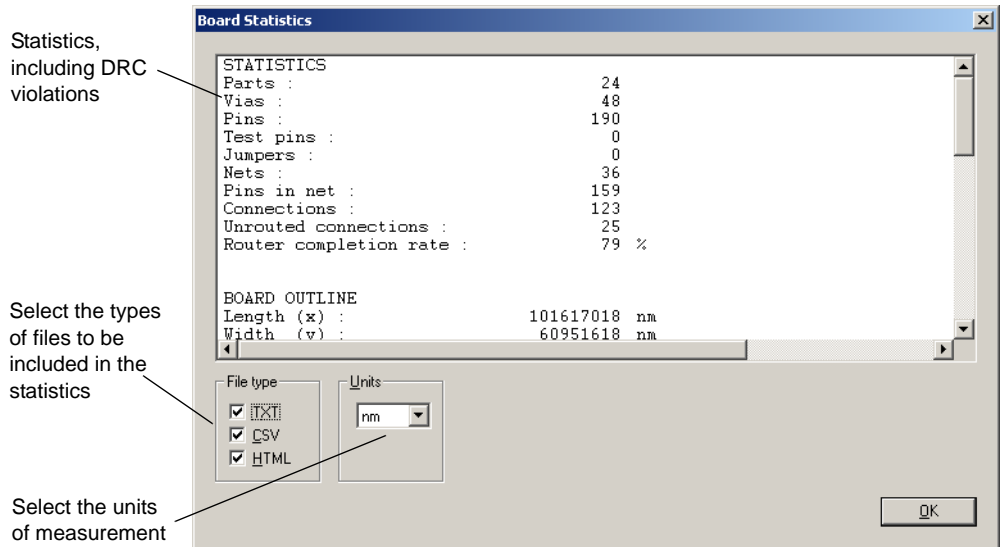
Working with Bill of Materials Properties

The **Bill of Materials** dialog box displays the bill of materials, and includes facilities for sorting the information displayed:

Drag the column headers to change the column order

Click column headers to sort the columns

Bill of Materials

Select the file types to export as

Define a new column   Select a column to delete

Other attributes that have been defined for objects can be assigned to columns for reporting purposes (e.g. preferred component supplier, etc.).

Working with Parts Centroids Properties

The **Parts Centroids** dialog box provides a list of all parts on the boards and their coordinates, and includes facilities for sorting the information displayed:

Drag the column headers to change the column order

Click column headers to sort the columns

Properties

Select the file types to export as

Select the units of measurement

Define a new column

Select a column to delete
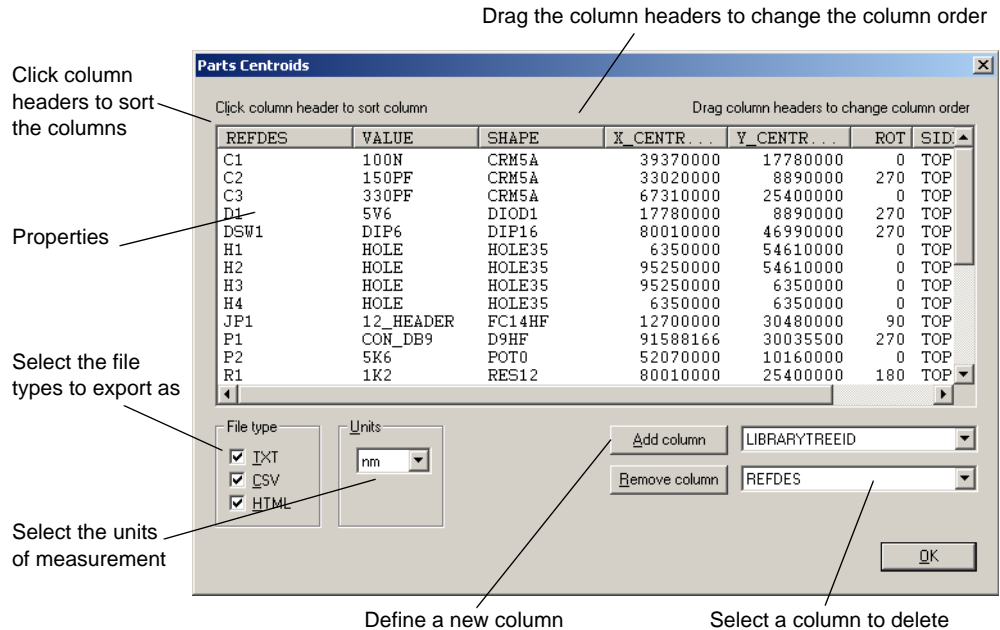
Working with Test Point Properties

The **Test Points Report** dialog box provides a list of all test points on the boards and their coordinates, etc.

REFDES	Coordinate[nm]	Diameter[nm]	Board Side	Rot
TP1	X:74054970,Y:10076180	177800	Top	0
TP2	X:74054970,Y:7942580	177800	Top	0
TP3	X:37783770,Y:42080180	177800	Top	0
TP4	X:16447770,Y:39946580	177800	Top	0
TP5	X:16447770,Y:38524180	177800	Top	0
TP6	X:16447770,Y:34290000	177800	Top	0
TP7	X:59830970,Y:52070000	177800	Top	0
TP8	X:61253370,Y:45636180	177800	Top	0
TP9	X:61253370,Y:44450000	177800	Top	0
TP10	X:73343770,Y:42080180	177800	Top	0
TP11	X:73343770,Y:39235380	177800	Top	0
TP12	X:73343770,Y:32834580	177800	Top	0
TP13	X:70498970,Y:27856180	177800	Top	0

Test Points Report

Click column header to sort column — Drag column headers to change column order

File type: ☑ TXT ☑ CSV ☑ HTML

Units: nm

Add column — SHAPE
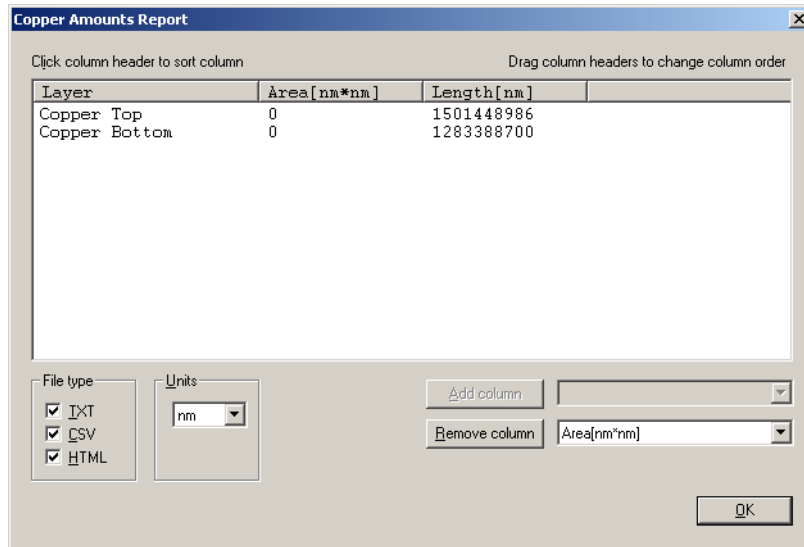
Remove column — Board Side
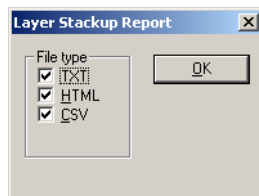
OK

Working with Copper Amount Properties

The **Copper Amounts Report** dialog shows the amount of copper used on the board.

Working with Layer Stackup Properties

The **Layer Stackup Report** dialog lets you set which file types to export when you run a Layer Stackup Report.

Note A Layer Stackup Report shows you a board's layers, the layer type (ground, power, signal or unassigned) and the types of vias that are between layers.

10.9.3 Exporting the Desired File

➢ To export a file:

1. Select **File/Export** to display the **Export** dialog box.
2. Select the export setting to be used.

3. If desired, change any properties stored in the export setting.

Note When exporting an IPC-D-356A netlist, there are no user-settable properties.

4. Select the type of export from the list.

5. Click **Export**. A dialog box opens where you can define the export file's name and path. If you are exporting more than one file, you must define the name and path of each file.

6. Select the path that defines the export file's location and type the file's name.
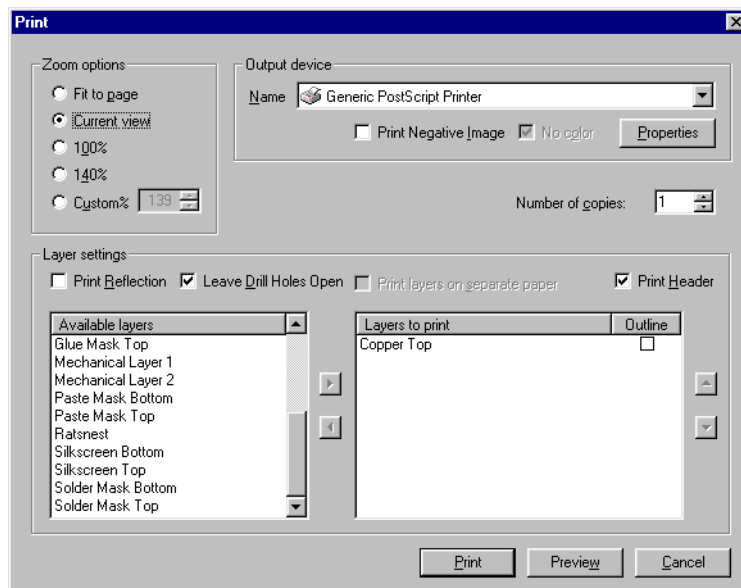
7. Click **Save**.

The file has been exported.

10.10 Printing your Design

➢ To print a design file:

1. Choose **File/Print**. The **Print** dialog box appears.

2. Choose from the list of available printers and, if necessary, set the printer's properties appropriately.

3. To print a negative image, enable the **Print Negative Image** option.

4. To print the reflection (mirror image), enable the **Print Reflection** option.

5. To leave drill holes open, enable the **Leave Drill Holes Open** option.

6. Select the layers you want to print in the **Available layers** list and click the arrow to move them to the **Layers to print** list.

7. For each layer you choose to print, you can enable or disable the **Outline** option to specify whether or not to include the board outline with the layer.
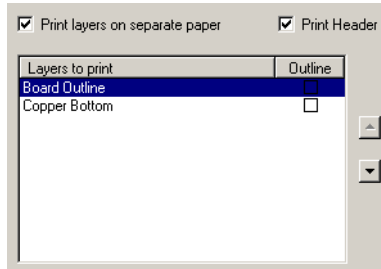
Once you have more than one layer selected, you can choose to print layers on separate sheets. You can also choose to print a header at the top of each page, containing the design name, date, and layer name. Finally, you can use the arrows to change the order in which layers will be printed.
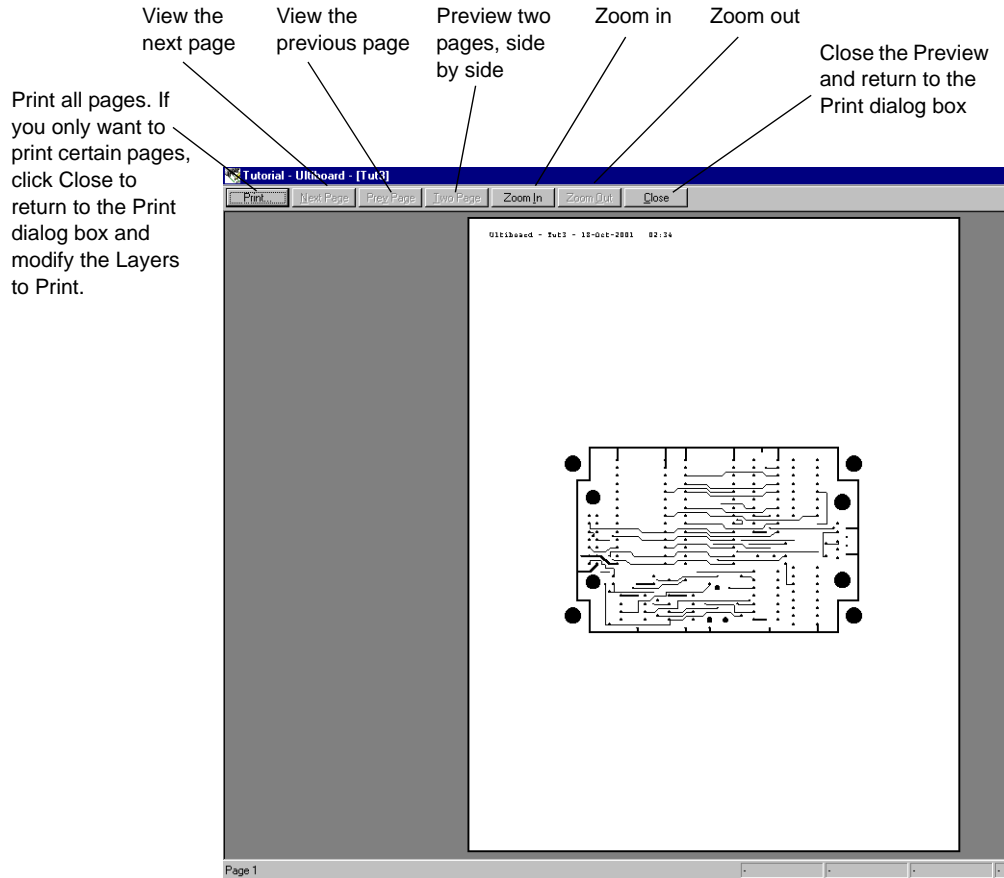
8. When you have finished setting the print parameters, click **Print**.

➢ To print a 3D image of your design:

1. Select **Tools/3D Viewer**. The 3D view of the design appears.

2. Scale the image to the desired size and orientation. For details, see " Viewing Designs in 3D" on page 11-1.

3. To set up the printing of the 3D image, select **File/Print Setup**.

4. To preview the printing of the 3D image, select **File/Print Preview**.

5. Select **File/Print** and click **OK**.

10.11 Previewing the Printed Design

➢ To preview the way a design will look when printed:

1. Choose **File/Print.** The **Print** dialog box appears.

2. Click **Preview**. (At least one layer must be in the **Layers to print** column to enable the **Preview** button). The **Preview** screen appears, displaying an image of what the printed file will look like.

Note A blank page in a preview dialog box usually indicates an empty layer is being previewed. If the preview displays a blank page, go back and make sure that you are previewing a layer that has something on it.

View the next page
View the previous page
Preview two pages, side by side
Zoom in
Zoom out
Close the Preview and return to the Print dialog box

Print all pages. If you only want to print certain pages, click Close to return to the Print dialog box and modify the Layers to Print.

If you elect to enlarge the size of your printout in the **Zoom Options** area, each layer will be tiled onto as many pages as required to print the whole layer.
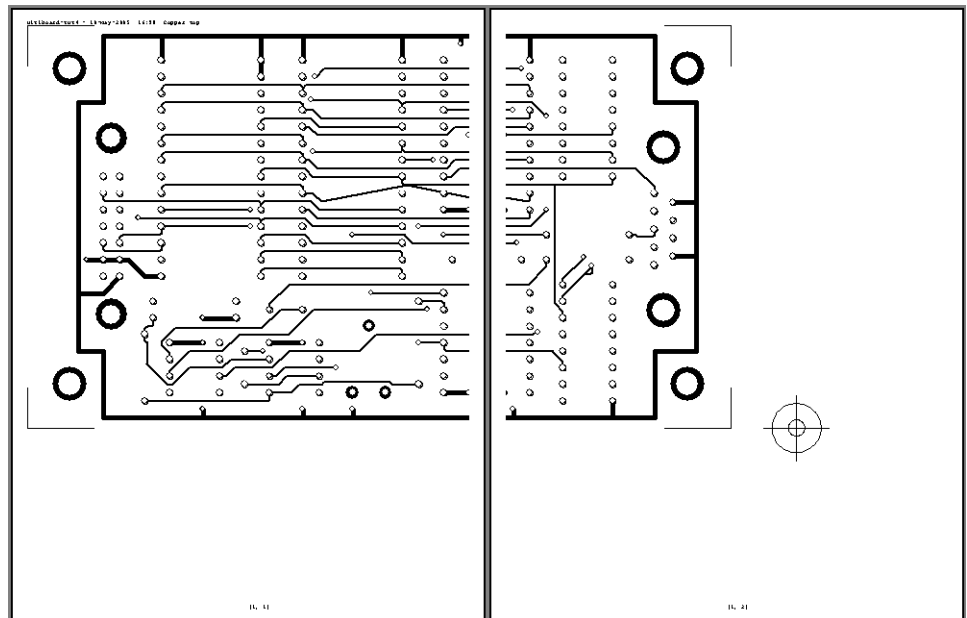
Chapter   11
Viewing Designs in 3D

Ultiboard allows you to see what the board looks like in three dimensions (3D) at any time during the design.

This chapter explains how to set up the options for 3D viewing, how to view the board in 3D, and how to manipulate the view.
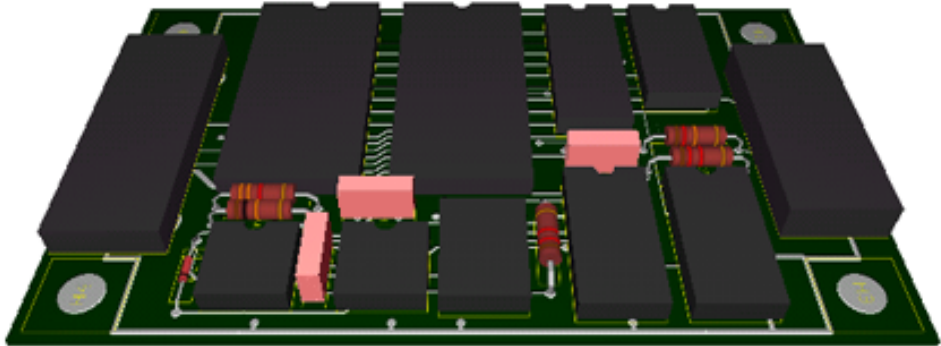
The following are described in this chapter.
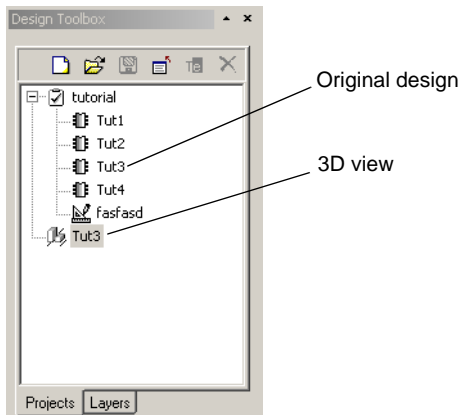
11.1   Viewing the Board in 3D

➤ To view the board in three dimensions, choose **Tools/3D Viewer**.

A new window opens displaying a 3D representation of the board:

The **Projects** tab in the **Design Toolbox** indicates that you have a 3D view open, in addition to the design from which the 3D view was taken:

➤ To close the 3D view, right-click on the 3D view in the **Projects** tab and choose **Close Window** from the context menu.

> **Note**   If you loaded a file from Ultiboard 2001, before you can use the 3D view you must use **Tools/Update Shapes**.

The 3D appearance of individual parts is controlled by the 3D properties of those parts. For details, see "6.2.2 Viewing and Editing Component Properties" on page 6-20.

11.2   Manipulating the 3D View

When you choose **Tools/3D Viewer**, the view appears so that you are looking at a three dimensional rendition of the top of the board on an angle. You can manipulate this view to show you all points on the board, top and bottom, at any angle you choose.

You manipulate the 3D view with the mouse pointer, which represents your viewing position and the light source that illuminates the component. You can click and drag the mouse to rotate the board, and you can change the angle at which it displays, allowing you to flip it over to see the bottom. The view rotates around the center of the board. When you let go of the mouse button, the board stays in its most recent view.

➢ To manipulate the 3D view:

1. Select **Tools/View 3D Position**. This is the default setting when you enter the 3D view.

2. Click and hold the mouse button down on the board in the 3D view.

3. Still holding the button, move the pointer as follows:

   • Down to the bottom of the screen to view more of the top of the board.
   • Up to the top of the screen to view the edge and then the underside of the board.
   • To the left or right to view the board from either end.

➢ To pan the 3D view:

1. Hold down your mouse wheel. The pointer becomes a four-headed arrow.

2. Move the pointer in any direction.

➢ To turn the board over:

1. Click and hold on the board.

2. Move the pointer up, towards the top of the screen. As the view of the edge passes, release the mouse button and catch the underside of the board.

3. Continue until you can see the underside of the board fully.

➢ To zoom in a 3D design do one of the following:

• Click and hold the right mouse button. Roll the mouse up to zoom in and down to zoom out.
   Or

• Use **View/Zoom In** and **View/Zoom Out**. You can also use your mouse wheel, if available.

11.2.1 Controlling the Elements Viewed in 3D

The **Design Toolbox** in the 3D view also has a **Layers** tab. The layers on this tab do not necessarily correspond to the layers in the design, but they work the same way as the **Layers** tab does in designing: the 3D **Layers** tab allows you to dim or remove elements from the board. For details on dimming and removing layers, see "5.1.2 Accessing Layers" on page 5-3.
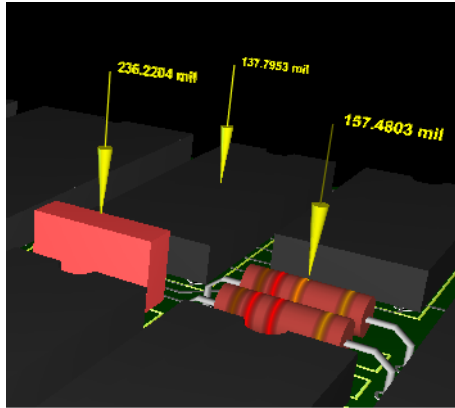
The following is a typical 3D **Layers** tab:
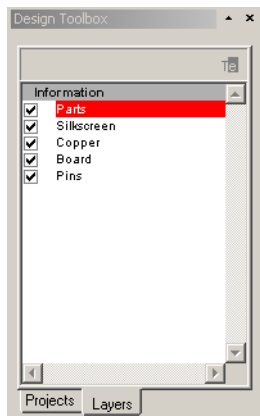
11.2.2 Showing an Object's Height

While in the 3D view, you can show an object's height.

➢ To show an object's height:

1. Select **Tools/Show or Hide Height**.

2. Click the cursor on a 3D component. A callout with the component's height appears.

3. Repeat on as many components as desired.

➢ To hide a component's height, click on the component. Its the callout with the height disappears.

➢ To rotate or magnify the board, select **Tools/View 3D Position**. For details on this command, see "11.2 Manipulating the 3D View" on page 11-3.
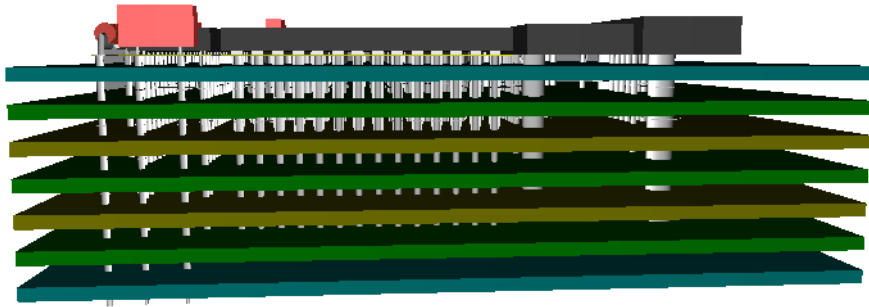
11.2.3 Exploded View

You can use the **Exploded View** to look between the layers of a multi-layer PCB.

➢ To view the 3D exploded view of a PCB:

1. From the 3D view, select **View/Internal Layers**.

Normal 3D View

Exploded 3D View

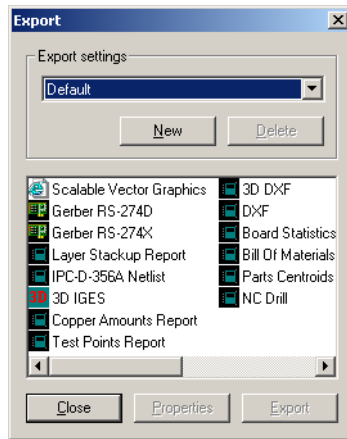2. Zoom in and out on the **Exploded View** as desired.

Note **Exploded View** options are set in the **3D Options** tab of the **Preferences** dialog box. For details, see "3.3.6 3D Options Tab" on page 3-21.

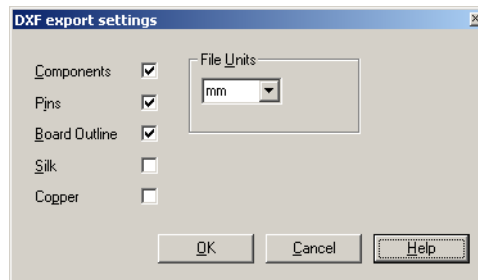# 11.3   Exporting to 3D DXF

Ultiboard can export a 3D DXF file for your design.

➢ To export a design's 3D properties:

1. Select **File/Export**.

2. Highlight **3D DXF** and click **Properties**. The **DXF export settings** dialog box appears.

3. Select the desired parameters to export and click **OK**. You are returned to the **Export** dialog box.

4. Click **Export**. A standard Windows Save dialog appears.

5. Select the desired filepath and enter the 3D DXF filename (the file extension must be .DXF).
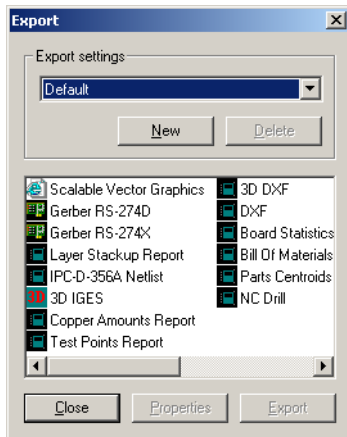
6. Click **Save**.
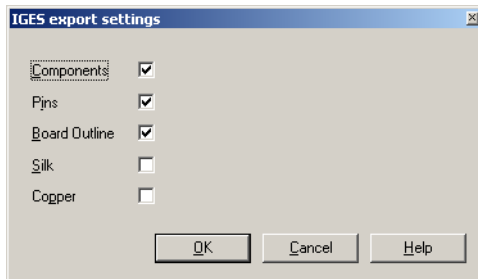
11.4   Exporting to 3D IGES

3D IGES (Initial Graphics Exchange Specification) is a file format for the exchange of CAD information (both 2D and 3D). A 3D IGES file contains surface information and details of a part.

➢ To export a design's 3D IGES properties:

1. Select **File/Export**.

2. Highlight **3D IGES** and click **Properties**. The **IGES export settings** dialog box appears.

3. Select the desired parameters to export and click **OK**. You are returned to the **Export** dialog box.

4. Click **Export**. A standard Windows Save dialog appears.

5. Selected the desired filepath and enter the 3D IGES filename (the file extension must be .igs).

6. Click **Save**.

Chapter 12
Using Mechanical CAD

Ultiboard's mechanical CAD function can be used to design enclosure boxes, front panels, or any other mechanical part associated with your PCB design. Mechanical CAD functionality in Ultiboard takes two forms:

• You can create multi-layer mechanical CAD design files.
• You can have mechanical CAD layers as part of your PCB design files. Design files can include up to 10 mechanical CAD layers. These are accessed through the **Layers** tab of the **Design Toolbox** and can be used in the same way as other layers in your design. For details, see "5.1.2 Accessing Layers" on page 5-3.

For the most part, you work with mechanical CAD designs the same way that you do with PCB designs, although you cannot place traces or use the Autoroute function in mechanical CAD designs.
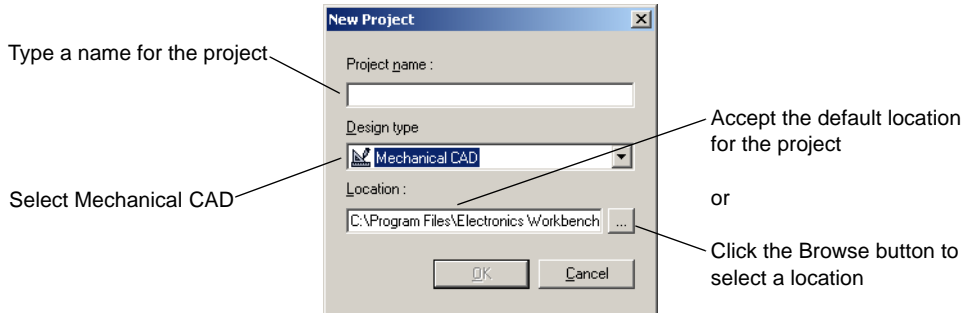
The following are described in this chapter.

12.1   Creating Mechanical CAD Projects

➢ To create a new project :

1. Choose **File/New Project**. The **New Project** dialog box appears.

Type a name for the project

Select Mechanical CAD

Accept the default location
for the project

or

Click the Browse button to
select a location

2. Type the project name in the **Project name** field.

3. From the **Design type** drop-down list, select **Mechanical CAD**.

4. Indicate the folder where the file is to be kept. Either accept the default shown in the **Location** field, or click the Browse button to select a different location.

5. Click **OK**. The **New Project** dialog box disappears, and a blank unnamed mechanical CAD design is opened in that project. The project you just created is also shown in the **Projects** tab.

12.2   Creating Mechanical CAD Design Files

➢ To create a new mechanical CAD design, you can either use the new design that appears when you create a mechanical CAD file, or you can create a new design and assign it to an existing file. To create a new design and assign it to an existing file:

1. Open the file that the new design is to be added to.

2. Choose **File/New Design**. The **New Design** dialog box appears.
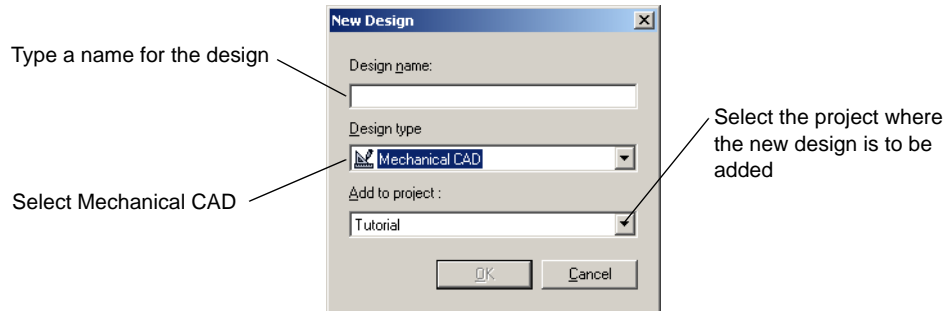
Type a name for the design

Select Mechanical CAD

Select the project where the new design is to be added

3. Type the design name in the **Design name** field.

4. From the **Design type** drop-down list, select **Mechanical CAD**.

5. Indicate the file where the design is to added. The drop-down list shows only the open files. To include this design in another project, open the file so that it appears in the **Add to file** drop-down list.

6. Click **OK**. The **New Design** dialog box disappears, and a blank mechanical CAD design with the name that you specified is opened as a separate window.

   The **Projects** tab shows the mechanical CAD design is a part of the open project that you specified during the design's creation the dialog box:

Files

PCB design

Mechanical CAD design
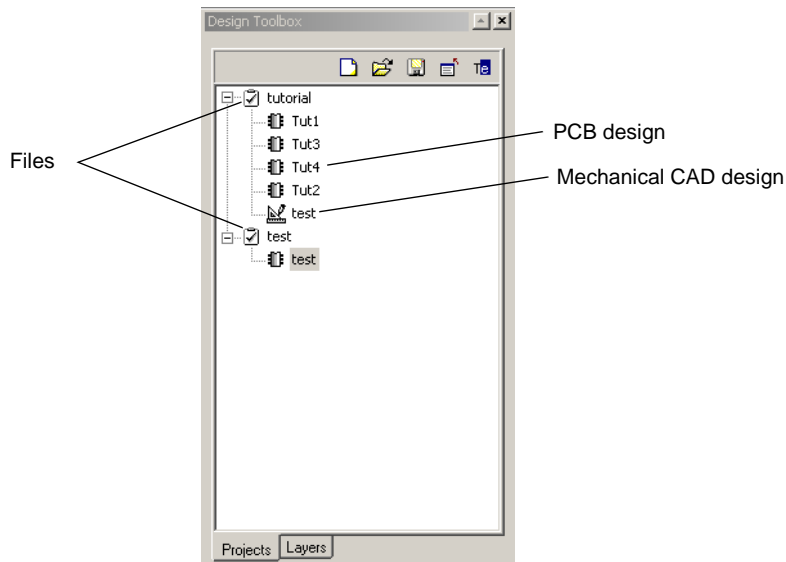
Note Mechanical CAD designs can be part of a project containing PCB designs. Project files are independent of design file types.

12.2.1 Importing a DXF File

➢ To import a DXF file into a mechanical CAD design:

1. Choose **File/Import/DXF**. A standard file selector appears.

2. Navigate to the correct location for the .dxf file, select it and click **OK**.

12.3   Setting Mechanical CAD Properties and Options

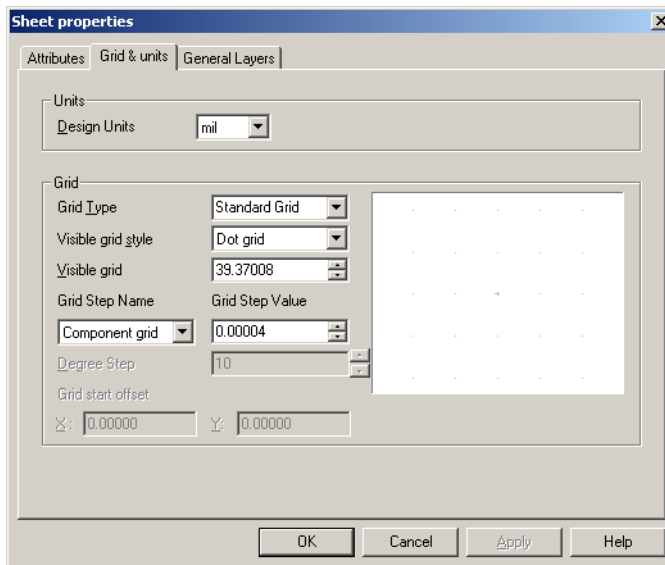## 12.3.1 Setting Mechanical CAD Properties

➢ To set mechanical CAD properties:

Choose **Edit/Properties**.

Or

Right-click on an empty area of the design and choose **Properties** from the context menu.
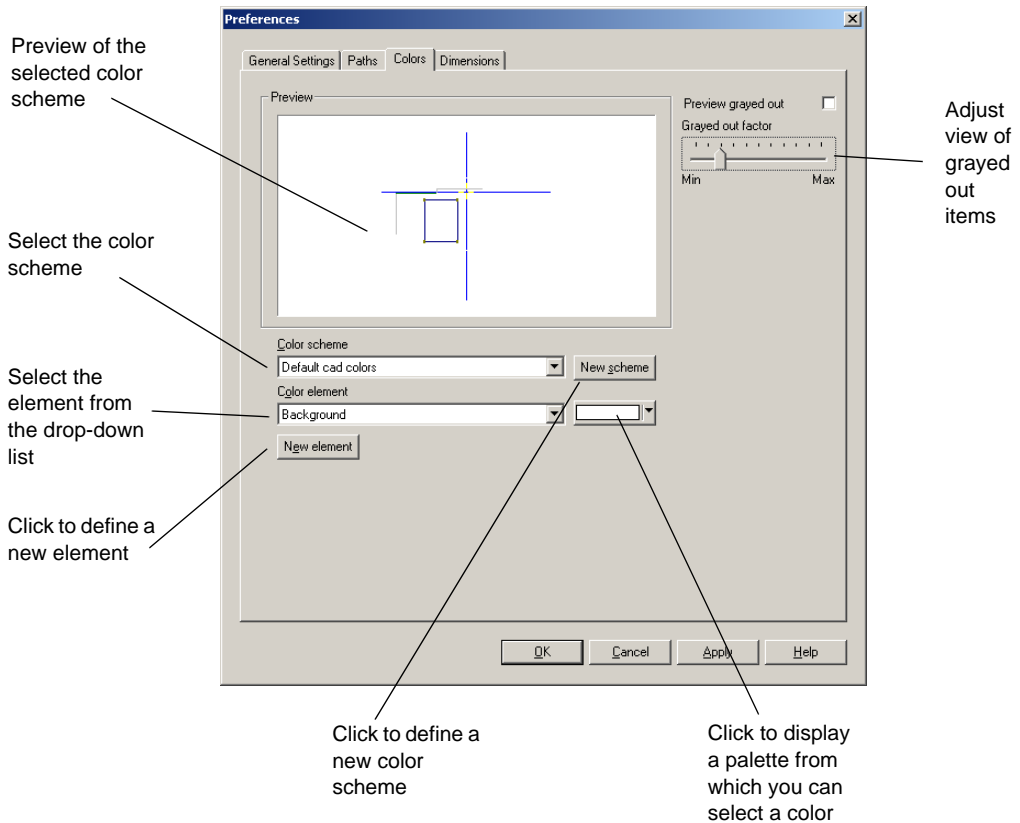
Working with mechanical CAD properties is done the same way as with PCB properties, except that mechanical CAD designs have only two grids.

12.3.2 Setting Mechanical CAD Colors

1. Choose **Options/Global Preferences**. The **Preferences** dialog box appears.

2. Click the **Colors** tab:

Preview of the selected color scheme

Adjust view of grayed out items

Select the color scheme

Select the element from the drop-down list

Click to define a new element

Click to define a new color scheme

Click to display a palette from which you can select a color

Except for default elements being slightly different, this dialog box works the same way as for PCB designs. For details on using the **Colors** tab, see "3.3.3 Colors Tab" on page 3-16.

12.3.3 Controlling Workspace Elements for Mechanical CAD

The **General Settings** tab allows you to control whether or not invisible attributes or cross hairs are shown in normal view, and options for full screen view. This tab also allows you to have Ultiboard load your last project automatically, and to have Ultiboard automatically save your project at specified intervals.

➢ To view or change workspace options:

1. Choose **Options/Global Preferences**. The **Preferences** dialog box appears.

2. Click the **General Settings** tab:
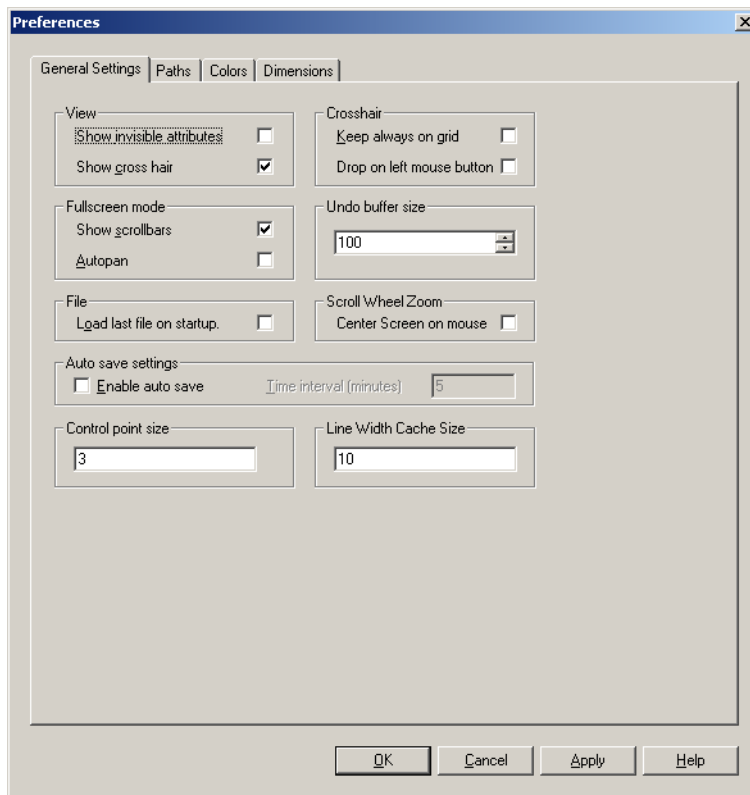
3. Adjust the **View** box settings as desired:

- **Show invisible attributes** — these are set up in the **Attributes** tab (**Visibility** column) of the element's properties dialog.
- **Show crosshair** — displays a large crosshair that is attached to your mouse's pointer.

4. Adjust the **Crosshair** box settings as desired:

   - **Keep always on grid** — makes sure that the crosshair will always snap to grid points even when you are not placing any parts or traces (normally when you are not placing parts or traces the crosshair moves gridless.
   - **Drop on left mouse button** — changes the way moving of objects works. Normally when you start a move, the object will be dropped when you release the left mouse button. When this option is enabled, releasing the mouse button will not drop the component - only pressing the left mouse button will.

5. Adjust the **Fullscreen mode** box settings as desired:

   - **Show scrollbars** — places vertical and horizontal scrollbars in the fullscreen view. Use these to move to the desired area on the workspace.
   - **Autopan** — automatically moves the view of the workspace as you move the cursor offscreen.

6. Increase or decrease the **Undo buffer size** as desired. The number here is the number of undo actions allowed.

7. In the **File** box, enable **Load last file on startup** if you would like to continue working on the last file you had open in your previous Ultiboard session.

8. In the **Scroll Wheel Zoom** box:

   - **Center Screen on mouse** — if your mouse has a scroll wheel, enabling this option will center the part of the workspace that is under the cursor when you turn the scroll wheel.

9. In the **Auto save settings** box:

   - **Enable auto save** — activates the autosave function. When activated, you can also change the time between autosaves in the **Time interval (minutes)** field.

10. Edit the following as desired:

    - **Control point size** — the size of control points on vertices and other objects.
    - **Line width cache size** — The number of recently used line widths Ultiboard keeps in memory. Minimum cache size is 5.
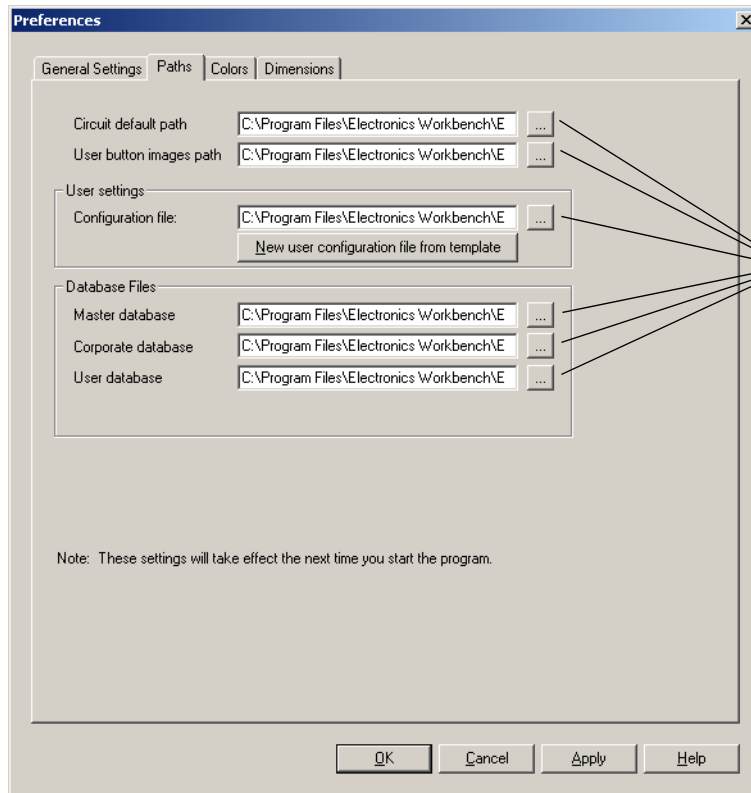
12.3.4 Setting Paths for Mechanical CAD

The Ultiboard installation puts specific files in specific locations. If necessary you can point
Ultiboard to a new location.

➢ To set up file locations for mechanical CAD files:

1. Choose **Options/Global Preferences**. The **Preferences** dialog box appears.

2. From the **Preferences** dialog box, click the **Paths** tab:

This tab works the same way as the one for PCB designs. For details on using the **Paths**
tab, see "3.3.2 Paths Tab" on page 3-15.

12.3.5 Setting Mechanical CAD Dimensions

➢ To set mechanical CAD dimensions:

1. Choose **Options/Global Preferences**. The **Preferences** dialog box appears.

2. Click the **Dimensions** tab and set the parameters for dimensions:

A preview of the results of your choices appears here.
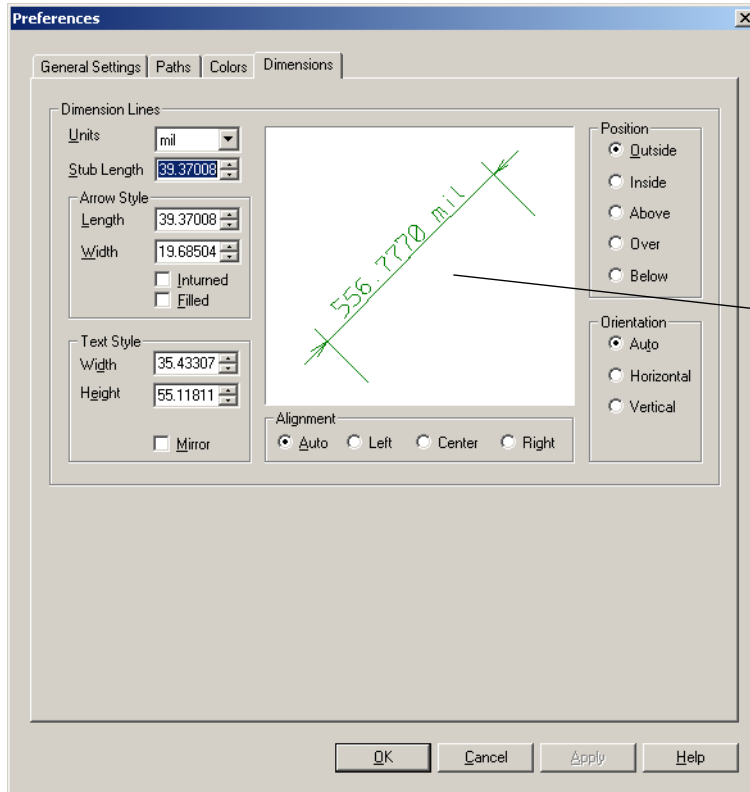
This tab works the same way as the **Dimensions** tab for PCB designs. For details on using the **Dimensions** tab, see "3.3.5 Dimensions Tab" on page 3-19.

Appendix A

This chapter contains brief descriptions for the commands in the Ultiboard menus.

A.1   File Menu

A.1.1   File/New Design

Creates a new design (if a project is open) or project (if no project is open). For details, see "4.1 About Designs and Projects" on page 4-2.

A.1.2   File/New Project

Creates a new project. For details, see "4.1 About Designs and Projects" on page 4-2.

A.1.3   File/Open

Opens an existing project and/or design file. For details, see "4.6 Opening an Existing File" on page 4-6.

A.1.4   File/Open Samples

Displays the Samples folder.

A.1.5   File/Save

Saves the current design file and project.

A.1.6 File/Save As

Saves the current design file with a name and location that you specify in the **Save As** dialog box.

A.1.7 File/Save All

Saves all open design files and projects.

A.1.8 File/Close

Closes the current design file.

A.1.9 File/Close Project

Closes the current project.

A.1.10 File/Close All

Closes all open design files and projects.

A.1.11 File/Import

Use to import an Ultiboard Netlist or DXF file.

A.1.12 File/Export

Exports Ultiboard files for post-processing. For details, see "10.9 Exporting a File" on page 10-10.

A.1.13 File/Save Technology

Lets you save a technology file based on the current design that contains the design rules and configuration options for both Ultiboard and Ultiroute. For details, see "4.8 Saving Technology" on page 4-8.

A.1.14 File/Load Technology

Lets you load a technology file that contains the design rules and configuration options for both Ultiboard and Ultiroute. For details, see "4.8.1 Loading Technology Files" on page 4-9.

A.1.15 File/Print Setup

Sets up the printing paramaters required to print the Ultiboard design file. For details, see "10.10 Printing your Design" on page 10-20. This is found in the **3D view** only.

A.1.16 File/Print Preview

Lets you preview the design file before printing. For details, see "10.10 Printing your Design" on page 10-20. This is found in the **3D view** only.

A.1.17 File/Print

Prints the Ultiboard design file. For details, see "10.10 Printing your Design" on page 10-20.

A.1.18 File/[Recent Files]

Displays a list of the most-recently-opened projects.

A.1.19 File/Exit

Exits Ultiboard. You are prompted to save any unsaved design files and/or projects.

A.2   Edit Menu

A.2.1   Edit/Undo

Undoes your most recent action (for example, undoes the last component move). Command name changes to reflect what will be undone.

A.2.2   Edit/Redo

Redoes your most recent undone action. Command name changes to reflect what will be redone.

A.2.3   Edit/Cut

Removes the selected element(s) from the board. Element is placed on the Windows Clipboard and can be pasted again.

A.2.4   Edit/Copy

Copies the selected elements and stores them on the Windows Clipboard so they can be pasted again.

A.2.5   Edit/Paste

Pastes the item on the Windows Clipboard to its original layer (regardless of what layer is currently active). Creates new duplicate object(s) without the netlist information. New components will have RefDes that increment from the next available for that component type. For example, if the last resistor was R34, and a resistor is pasted, its RefDes will be R35.

A.2.6   Edit/Paste Special/Paste with net

Pastes the item on the Windows Clipboard into the design with the same connectivity as the source. New components will have RefDes that increment from the next available for that component type.

A.2.7   Edit/Paste Special/Paste to active layer

Pastes the item on the Windows Clipboard to the active layer (regardless of what layer it originally resided on).

A.2.8   Edit/Delete

Use to delete the selected design elements. You are prompted to confirm the deletion.

A.2.9   Edit/Copper Delete

Deletes open trace ends, unused vias, or all copper elements.

All Copper	Use to delete all copper in the design, including traces, copper areas, or powerplanes. For details, see "7.2.5 Deleting All Copper" on page 7-12.
All Teardrops	Use to delete all teardrops on your design. For details, see "7.2.6.1 Removing Teardrops" on page 7-13.
Open Trace Ends	Use to delete all open trace ends in the design. Use this command to clean up the design after design completion. For details, see "10.8.1 Deleting Open Trace Ends" on page 10-9.
Copper Island	Use to remove copper islands. For details, see "7.2.1 Placing Copper Areas" on page 7-10

A.2.10 Edit/Find

Use to find an element in the design. For details, see "4.12 Searching for Design Elements" on page 4-11.

A.2.11 Edit/Select All

Use to select everything on a board, no matter what layer the elements are on. For details, see "4.12 Searching for Design Elements" on page 4-11.

A.2.12 Edit/Group Selection

Groups selected elements together. Grouped elements can be moved together.

A.2.13 Edit/Ungroup Selection

Ungroups the individual elements in a group.

A.2.14 Edit/Select Entire Trace

Use to select a whole trace when one or more segments of the same trace are selected.

A.2.15 Edit/Unlock

Unlocks the selected elements.

A.2.16 Edit/Lock

Locks the selected elements in place so they cannot be moved.

A.2.17 Edit/Selection Filter

Use these toggles to prevent accidentally selecting a particular type of element, for example, selecting a component when you meant to select a trace.

Enable Selecting Parts	When enabled, allows parts to be selected.
Enable Selecting Traces	When enabled, allows traces to be selected.

Enable Selecting Polygons	When enabled, allows polygons to be selected.
Enable Selecting Vias	When enabled, allows vias to be selected.
Enable Selecting Pads	When enabled, allows pads to be selected.
Enable Selecting SMD Pads	When enabled, allows Surface Mount Device pads to be selected.
Enable Selecting Attributes	When enabled, allows attributes to be selected.
Enable Selecting other objects	When enabled, allows other objects on the PCB to be selected.

A.2.18 Edit/Orientation

Use these commands to adjust the orientation of parts as they are placed on a design. For more details on the Orientation commands, see "6.1.3.6 Orienting Components" on page 6-11.

Flip Horizontal	Flips the selected elements horizontally.
Flip Vertical	Flips the selected elements vertically.
90 Clockwise (Ctrl+R)	Flips the selected elements 90 degrees clockwise.
90 CounterCW (Ctrl+Shift+R)	Flips the selected elements 90 degrees counter-clockwise.
Angle	Allows you to specify the angle of rotation for the selected elements.
Swap Layer (Alt+S)	Swaps the selected elements from a layer to its mirror layer (e.g., from Copper Top to Copper Bottom).

A.2.19 Edit/Align

Use these commands to align the edges of the elements.

Align Left	Aligns the left edges of the selected elements.
Align Right	Aligns the right edges of the selected elements.
Align Top	Aligns the top edges of the selected elements.

Align Bottom	Aligns the bottom edges of the selected elements.
Align Center Horizontal	Shifts the selected elements horizontally so their centers are aligned.
Align Center Vertical	Shifts the selected elements vertically so their centers are aligned.
Space Across	Spaces three or more objects beside each other evenly.
Space Across Plus	Increases horizontal space between two or more objects.
Space Across Min	Decreases horizontal space between two or more objects.
Space Down	Spaces three or more objects above each other evenly.
Space Down Plus	Increases vertical space between two or more objects.
Space Down Min	Decreases vertical space between two or more objects.

A.2.20 Edit/Vertex

You can add or remove vertices from all polygons, whether copper or non-copper.

Add Vertex	Add a vertex to a polygon segment.
Remove Vertex	Remove a vertex from a polygon.

See also, "6.5.2 Editing a Polygon" on page 6-42.

A.2.21 Edit/In-Place Part Edit

Use to edit a part that has already been defined and placed on the board. For details, see "6.5.1 Editing a Placed Part (In-Place Edit)" on page 6-40.

A.2.22 Edit/Properties

Displays the properties of the selected element. Results will differ, depending on what is selected. If no element is selected, displays the board properties. Properties are described throughout this guide.

A.3 View Menu

A.3.1 View/Full Screen

Use to fill the screen with the design only (hide menus, toolbars, other windows). Click the **Close Full Screen** button to return to normal view.

A.3.2 View/Redraw Screen

Use to refresh the screen.

A.3.3 View/Zoom In

Use to zoom in on the design so you see more details.

A.3.4 View/Zoom Out

Use to zoom out of the design so you see more of the design.

A.3.5 View/Zoom Window

Use to magnify a selected part of the board. Use this command when you want precise control over the workspace view.

A.3.6 View/Zoom Full

Use to return to the full view of the design after zooming in or out. CTRL+F7 zooms out so that the entire design is visible and centered. This includes the board outline and any objects that may be either inside or outside the board. F7 zooms out so that the board is visible and centered. Any objects outside the board outline will be outside the visible region.

A.3.7 View/Grid

Use to toggle the visual grid on and off.

A.3.8 View/Ruler Bars

Use to toggle the ruler bars on and off. For more details, see "6.1.3.5 Using Ruler Bars" on page 6-11.

A.3.9 View/Clearances

Use to toggle the clearances around pads, traces and other objects on and off. For more details, see "7.1 Placing Traces" on page 7-2.

A.3.10 View/Status Bar

Use to toggle the status bar on and off.

A.3.11 View/Density Bars

Use to toggle the density bars on and off. For more details, see "7.1.6 Working with Density Bars" on page 7-7.

A.3.12 View/Design Toolbox

Use to toggle the **Design Toolbox** on and off. For more details, see "3.5 Design Toolbox" on page 3-31.

A.3.13 View/Birds Eye

Use to toggle the **Birds Eye View** on and off.

A.3.14 View/Spreadsheet View

Use to toggle the **Spreadsheet View** on and off. For more details, see "3.6 Spreadsheet View" on page 3-32.

A.3.15 View/3D Preview

Toggles the **3D Preview** pane on and off.

A.3.16 View/Toolbars

Use to turn toolbars on or off.

Standard	Use to toggle the standard toolbar on and off. For more details, see "3.2.1 The Standard Toolbar" on page 3-4.
View	Use to toggle the view toolbar on and off. For more details, see "3.2.2 The View Toolbar" on page 3-5.
Main	Use to toggle the main toolbar on and off. For more details, see "3.2.3 The Main Toolbar" on page 3-5.
Draw Settings	Use to toggle the draw settings toolbar on and off. For more details, see "3.2.6 The Draw Settings Toolbar" on page 3-8.
Edit	Use to toggle the edit toolbar on and off. For more details, see "3.2.7 The Edit Toolbar" on page 3-9.
Align	Use to toggle the align toolbar on and off. For more details, see "3.2.8 The Align Toolbar" on page 3-9.
Place	Use to toggle the place toolbar on and off. For more details, see "3.2.9 The Place Toolbar" on page 3-10.
Select	Use to toggle the select toolbar on and off. For more details, see "3.2.4 The Select Toolbar" on page 3-7.
Wizard Toolbar	Use to toggle the wizard toolbar on and off. For more details, see "3.2.10 The Wizard Toolbar" on page 3-12.
Autoroute	Use to toggle the autoroute toolbar on and off. For more details, see "3.2.11 The Autoroute Toolbar" on page 3-12. This is available with Ultiroute only.
Settings	Use to toggle the settings toolbar on and off. For more details, see "3.2.5 The Settings Toolbar" on page 3-8.

A.4   Place Menu

A.4.1   Place/Select

Use to change from placing elements to selecting elements.

A.4.2   Place/From Database

Use to place parts from the database onto the workspace. For details, see "6.4 Placing Parts from the Database" on page 6-39.

A.4.3   Place/Line

Use to place a line or a trace, depending on the active layer. For more details, see "6.3.3 Placing Shapes and Graphics" on page 6-32 or "7.1.2 Placing a Trace: Manual Method" on page 7-3.

A.4.4   Place/Follow me

Use to place a trace between two selected points. For more details, see "7.1.3 Placing a Trace: "Follow Me" Method" on page 7-4.

A.4.5   Place/Connection Machine

Use to place a trace between two pads. For more details, see "7.1.4 Placing a Trace: Connection Machine Method" on page 7-4.

A.4.6 Place/Shape

Use to place shapes of different types.

Ellipse	Use to place an ellipse.
Rounded Rectangle	Use to place a rounded rectangle.
Circle	Use to place a circle.
Pie	Use to place a pie shape.
Rectangle	Use to place a rectangle.
Polygon	Use to place a polygon.

For more details, see "6.3.3 Placing Shapes and Graphics" on page 6-32.

A.4.7 Place/Dimension Lines

Standard	Use to place a standard dimension (a dimension to be placed at an angle). Dimension parameters (arrow style, text style, position and so on) are set through **Options/Global Preferences**.
Horizontal	Use to place a horizontal dimension. Dimension parameters (arrow style, text style, position and so on) are set through **Options/ Global Preferences**.
Vertical	Use to place a vertical dimension. Dimension parameters (arrow style, text style, position and so on) are set through **Options/ Global Preferences**.

A.4.8 Place/Graphics/Line

Use to place a line or a trace, depending on the active layer. For more details, see "6.3.3 Placing Shapes and Graphics" on page 6-32 or "7.1.2 Placing a Trace: Manual Method" on page 7-3.

A.4.9   Place/Graphics/Arc

Use to place an arc or a trace, depending on the active layer. For more details, see "6.3.3 Placing Shapes and Graphics" on page 6-32 or "7.1.2 Placing a Trace: Manual Method" on page 7-3.

A.4.10 Place/Graphics/Circle

Use to place a circle on the workspace.

A.4.11 Place/Graphics/Bezier

Use to place a bezier or a trace, depending on the active layer. For more details, see "6.3.3 Placing Shapes and Graphics" on page 6-32 or "7.1.2 Placing a Trace: Manual Method" on page 7-3.

A.4.12 Place/Graphics/Text

Use to place text on the design. Useful for annotation purposes. For more details, see "10.1 Placing and Editing Text" on page 10-2.

A.4.13 Place/Copper Area

Use to define a copper area polygon. For more details, see "7.2.1 Placing Copper Areas" on page 7-10.

A.4.14 Place/Powerplane

Use to define layers as Power/Ground planes or to clear layers that were previously defined as Power/Ground planes. For more details, see "7.2.2 Placing Powerplanes" on page 7-11.

A.4.15 Place/Bus

Use to connect multiple traces between multi-pinned devices such as ICs. For details, see "7.1.5 Placing a Bus" on page 7-5.

A.4.16 Place/Keep-in/Keep-out Area

Use to define a polygon to restrict elements from either exiting or entering the area. For more details, see "7.1.7 Working with Keep-in/Keep-out Areas" on page 7-7.

A.4.17 Place/Group Array Box

Use to place components in a grid array. For details, see "6.1.3.9 Placing a Group Array Box" on page 6-13.

A.4.18 Place/Net Bridge

Places a net bridge on the design. For details, see "7.6.4 Net Bridges" on page 7-36.

A.4.19 Place/Hole

Places a hole on the workspace. For details, see "6.3.2 Placing Holes" on page 6-31.

A.4.20 Place/Pins

Used in In-Place Editing of a part to place a footprint. For more details, see "6.5.1 Editing a Placed Part (In-Place Edit)" on page 6-40.

A.4.21 Place/Automatic Test Points

This function is only available when Ultiroute is installed. For details, refer to your copy of the *Ultiroute 9 User Guide*.

A.4.22 Place/Unplace Components

Use to unplace all non-locked components. For details, see "6.1.4 Unplacing Parts" on page 6-17.

A.4.23 Place/Via

Use to place a via. For more on placing vias, see "7.4 Working with Vias" on page 7-15.

A.4.24 Place/Test Point

Use to place a test point. For more on placing test points, see "6.3.5 Working with Test Points" on page 6-35.

A.4.25 Place/Jumper

Use to place a jumper. For more on placing jumpers, see "6.3.4 Working with Jumpers" on page 6-33.

A.4.26 Place/Comment

Places a comment on the design. For details, see "10.3 Placing a Comment" on page 10-5.

A.5   Design Menu

A.5.1   Design/Netlist & DRC Check

Use to run the design rules and netlist check utility. Results are displayed in the **DRC** tab of the **Spreadsheet View**. For details, see "3.6.1 Spreadsheet View: DRC Tab" on page 3-33.

Depending on your settings in the **PCB Design** tab of the **Preferences** dialog box, this utility may run automatically. You can use this command to force a check of the board's integrity, and may want to use it prior to saving or exporting the design.

A.5.2   Design/Connectivity Check

Runs a connectivity check on the design.

When this command is selected, the **Select a Net** dialog box displays. Select the net on which you wish to run the connectivity check and click **OK**.

Results are displayed in the **Results** tab of the **Spreadsheet View**. For details, see "3.6.2 Spreadsheet View: Results Tab" on page 3-34.

A.5.3   Design/Polygon Splitter

Use to split a copper area or powerplane. For details, see "7.2.3 Splitting Copper" on page 7-11.

A.5.4   Design/Shape to Area

Use to cut out areas in a copper shape to avoid other elements within its area. For details, see "7.2.4 Converting a Copper Shape to an Area" on page 7-11.

A.5.5   Design/Swap Pins

For details, see "7.7.1 Swapping Pins" on page 7-42.

A.5.6   Design/Swap Gates

For details, see "7.7.2 Swapping Gates" on page 7-43.

A.5.7   Design/Automatic Pin/Gate Swap

For details, see "7.7.3 Automatic Pin/Gate Swapping" on page 7-44.

A.5.8 Design/Fanout SMD

Use to place a via fanout for a SMD component. For details, see "7.5 Placing SMD Fanouts" on page 7-20.

A.5.9 Design/Add Teardrops

Use to add teardrops to pads. For details, see "7.2.6 Adding Teardrops" on page 7-13.

A.5.10 Design/Shield Nets

This function is only available once you have installed Ultiroute. For details, refer to your copy of the *Ultiroute 9 User Guide*.

A.5.11 Design/Corner Mitering

Use to apply corner mitering before production. For details, see "10.6 Mitering Corners" on page 10-8.

A.5.12 Design/Clean Unused Vias

Use to delete all vias that do not have any trace segments or copper areas connected to them. Use this command after **Edit/Copper Delete/Open Trace Ends** to clean up the design. For details, see "10.8.2 Deleting Unused Vias" on page 10-10.

A.5.13 Design/Group Replicate Place

Use to copy and place a group of components. For details, see "6.1.3.10 Replicating a Group" on page 6-15.

A.5.14 Design/Copy Route

Use to copy copper routing between groups. For details, see "7.6.5 Copying a Copper Route" on page 7-40.

A.5.15 Design/Highlight Selected Net

Use to select and highlight an entire net when you have selected one trace segment of that net. For details, see "7.6.3 Highlighting a Net" on page 7-35.

A.6 Tools Menu

A.6.1 Tools/Board Wizard

Assists you in creating a board outline. For details, see "5.2 Working with the Board Outline" on page 5-5.

A.6.2 Tools/Component Wizard

Assists you in creating a part and adds it directly to the user database. For details, see "6.8.2 Using the Component Wizard to Create a Part" on page 6-54.

A.6.3 Tools/Database/Database Manager

Use to manage the database of parts available to you. For details, see "6.9 Managing the Database" on page 6-58.

A.6.4 Tools/Database/Add Selection to Database

Use to place the selected part in the database. For details, see "6.9.2 Adding Parts to the Database" on page 6-63.

A.6.5   Tools/Database/Set Database Password

Use to set a password for access to database functions that make any changes to a database.

➢ To set a password for a database:

1. Select **Tools/Set Database Password**.
2. Select the desired database from the **Database** drop-down list.
3. Enter the new password in the **Password** field and re-enter it in the **Confirmation** field.
4. Click **OK** to save the password.

A.6.6   Tools/Database/Merge Database

Use to convert existing component databases to Ultiboard 9 format. For details, see "6.10.1 Merging Databases" on page 6-65.

A.6.7   Tools/Database/Convert V6/V7 Database

Use to update your V6 (Ultiboard 2001) or V7 databases to Ultiboard 9 format. For details, see "6.10.2 Converting 2001 or V7 Databases" on page 6-66.

A.6.8   Tools/PCB Transmission Line Calculator

Use to calculate parameters for typical printed circuit board trace geometries. For details, see "2.5.7 PCB Transmission Line Calculator" on page 2-16.

A.6.9   Tools/PCB Differential Impedance Calculator

Use to perform calculations for two traces that carry signals that are exactly equal and opposite (a differential pair). For details, see "2.5.8 PCB Differential Impedance Calculator" on page 2-17.

A.6.10 Tools/Netlist Editor

Use to remove or add nets and pads. For details, see "7.6.2 Using the Netlist Editor" on page 7-23.

A.6.11 Tools/Group Editor

Use to set up and edit various group types. For details, see "5.5 Working with the Group Editor" on page 5-14.

A.6.12 Tools/Renumber Footprints

Use to renumber components for production. For details, see "10.4 Renumbering Parts" on page 10-6.

A.6.13 Tools/Equi-space Traces

Use to equally space selected traces. For details, see "7.1.8 Equi-Spacing Traces" on page 7-9.

A.6.14 Tools/Change Shape

Use to substitute a part from the database for the current selected part. For details, see "6.6.3 Replacing Parts" on page 6-51.

A.6.15 Tools/Update Shapes

Use if you loaded Ultiboard V. 5 files into Ultiboard 9 and plan to use 3D. Adds the necessary 3D information to the footprints in the file.

A.6.16 Tools/Highlight Selection in Multisim

Highlights a component selection in Multisim. For details, see "6.7 Cross-probing" on page 6-52.

A.6.17 Tools/Capture Screen Area

You can take a screen capture of a selected area. For details, see "10.2 Capturing Screen Area" on page 10-3.

A.6.18 Tools/3D Viewer

Displays the design in 3D mode. For details, see Section 11, "Viewing Designs in 3D".

A.6.19 Tools/View 3D Position

This is only available when viewing the design in the 3D mode. For details, see "11.2 Manipulating the 3D View" on page 11-3.

A.6.20 Tools/Show or Hide Height

Shows or hides the height of selected objects in 3D mode. This is only available when viewing the design in the 3D mode. For details, see "11.2.2 Showing an Object's Height" on page 11-4.

A.7   Options Menu

A.7.1   Options/Global Preferences

Displays the **Preferences** dialog box. For details, see "3.3 Setting Preferences" on page 3-13.

A.7.2   Options/PCB Properties

Use to define the general parameters of your PCB design.

A.7.3 Options/Set Reference Point

Use to define the reference point for your board. For details, see "5.3 Setting the Board's Reference Point" on page 5-11.

A.7.4 Options/Part Shoving

◇ Use to toggle shoving on and off. When shoving is on, components shove any others out of the way when being moved or placed. For details, see "6.1.3.4 Shoving Components" on page 6-9.

A.7.5 Options/Customize User Interface

Use to customize your menus, toolbars and context menus. For more details, see "3.7 Customizing the Interface" on page 3-44.

A.8 Autoroute Menu

Note For other Autoroute menu options, which are added when Ultiroute is installed, see the *Ultiroute 9 User Guide*.

A.8.1 Autoroute/Start Internal Router

Starts the internal router.

A.8.2 Autoroute/Stop Internal Router

Stops the internal router.

A.9 Window Menu

Use these commands to arrange multiple designs open at a time.

A.9.1 Window/New Window

Use to display a new window containing a full view of the design currently open.

A.9.2 Window/Cascade

Use to cascade the open windows, so they are arranged one on top of the next while remaining separately selectable.

A.9.3 Window/Tile Horizontal

Use to adjust two or more windows so that they sit one on top of the other.

A.9.4 Window/Tile Vertical

Use to adjust two or more windows so that they sit beside each other.

A.9.5 Window/Close All Windows

Use to close all open windows.

A.9.6 Window/<open designs>

Shows all open design windows.

A.9.7 Window/Windows

Displays the **Windows** dialog box where you can show or close the files that you currently have open

A.10   Help Menu

A.10.1 Help/Ultiboard Help

Use to display the contents of the online help.

A.10.2 Help/Release Notes

Use to display Ultiboard release notes.

A.10.3 Help/Check for Updates

Runs the Support and Upgrade Utility (SUU). For details, see"1.5 Support and Upgrade" on page 1-13 .

A.10.4 Help/File Information

Displays the **File Information** dialog box.

A.10.5 Help/About Ultiboard

Use to display the version numbers of your copy of Ultiboard.

A.11   Context Menus

Depending on the action, the following context sensitve menus display when the right mouse button is clicked:

A.11.1 Select Menu

When you select an object or objects in a design and then right-click your mouse, a context menu with the following options displays.

Cancel

Cancels the current action.

Cut

Removes the selected element(s) from the board. Element is placed on the Windows Clipboard and can be pasted again.

Copy

Copies the selected elements and stores them on the Windows Clipboard so they can be pasted again.

Paste

Pastes the item on the Windows Clipboard to its original layer (regardless of what layer is currently active). Creates new duplicate object(s) without the netlist information. New components will have RefDes that increment from the next available for that component type. For example, if the last resistor was R34, and a resistor is pasted, its RefDes will be R35.

Delete

Deletes the selection from the design.

Select All

Selects all items on the design.

Select Entire Trace

Use to select a whole trace when one or more segments of the same trace are selected.

Quick Layer Toggle

Use to toggle between the last copper layer or to the opposite copper layer if the last layer was not copper.

Lock

Locks the selected elements in place so they cannot be moved.

Unlock

Unlocks the selected elements.

Place

Displays a number of elements that you can place on the design.

Shape

Displays a number of shapes that you can place on the design.

Orientation

Use these commands to adjust the orientation of parts as they are placed on a design. For more details on the Orientation commands, see "6.1.3.6 Orienting Components" on page 6-11.

Align

Use these commands to align the edges of the elements.

Vertex

Use to add or remove vertices from all polygons, whether copper or non-copper.

In-Place Edit

Switches to In-Placed Edit mode for the selected item. For details, see "6.5.1 Editing a Placed Part (In-Place Edit)" on page 6-40.

Properties

Displays the properties dialog box for the type of element selected. For details, see "6.2 Viewing and Editing Properties" on page 6-17.

A.11.2 Right-drag Menu

When you select an area by dragging and releasing the right mouse button, a context menu appears with the following selections.

Select all in rectangle

Selects all objects in the rectangle that you "drew" by dragging and releasing the right mouse button.

Select objects on active layer

Selects only those objects in the rectangle that you "drew" by dragging and releasing the right mouse button that are on the active layer as defined in the **Layers** tab of the **Design Toolbox**.

Select objects on specified layers

Displays the **Select layer(s)** dialog box.

➢ To select the layers on which you wish to select the objects:

1. Highlight the desired layers in the **Select layer(s)** dialog box and click **OK**.

2. All items that are on the selected layers that appear within the rectangle that you "drew" by dragging the right mouse button are selected.

A.11.3 Place Trace Menu

If you click the right mouse button when you are placing a trace a context sensitive menu displays with the following options.

Cancel

Cancels the placement of the trace segment being drawn.

Narrow

Narrows the trace that you are drawing. Should be within **Minimum Width** when set.

Widen

Widens the trace that you are drawing. Should be within **Minimum Width** when set.

Glossary

Active Layer

The layer on which any actions you choose will be performed.

All Angle Routing

Routing that allows both 90° and 45° angles.

Aperture Code

Identifies the type of aperture to be used by the Gerber output.

Birds Eye View

The small window that appears, by default, at the top left of the screen. Gives you an overview of your complete PCB, with components shown as outlines. Allows you to quickly navigate through your board or display a specific area of the design, sized as you wish.

Blind Via

A via that connects the top or bottom layer with any inner layer.

Buried Via

A via that connects inner layers.

Chamfer Corners

Corners at an increment of 45° on the trace routes.

Component Grid

Determines the increments by which elements (parts and shapes) can be placed on the board.

Copper Area

A copper polygon.

Copper Island

A copper area that is not connected to any other copper.

Design Toolbox

By default, appears on left side of screen. Consists of multiple tabs used to manage a design.

DRC

Design Rule Check

Feedthrough Via

A normal via that connects all layers, top, bottom and inner.

Force Vector

A line that indicates the optimal location for a component, considering all the connections of the component to achieve the shortest possible connections.

Grid

Determines the increments by which traces can be placed on the board.

Internal Rip-up and Retry Autorouter

A rip-up and retry grid-based utility that rips up poorly placed connections and retries them until it finds the most optimal design.

Mouse Grid

Controls the increments by which the cursor moves and where items are placed.

Net

A network of traces, to which you can add pins and copper areas.

Netlist

Contains connectivity information about pins and components.

Normal Feed-Through Via

Any via that connects all layers (top, bottom, and internal).

Output Window

Gives you useful information on the status of your design. By default, is turned off.

Pad Stack

The connections between layers of the board.

Prepreg

Prepreg is fiber glass that is pre-impregnated with resin to make it semi-rigid. Prepreg sheets can then be placed between the copper foil and heated under pressure to activate and set the resin.

Ratsnest

Visual display showing linear connections between pins, using the shortest possible line. A guide for pin connections, not a realistic representation of the board.

Refdes

Reference designator, the unique name given to a net.

Reference Point

The point from which coordinates are calculated, in normal mode.
Set using **Options/Set Reference Point**.

SMD Pad

A pad without a drill hole (drill diameter property in the pad code set to 0).

Status Bar

Displays useful and important information at the bottom of your screen.

Thermal Relief

Area around a pin where no copper appears, but which is crossed by copper lines to make connections. A thermal relief is used to dissipate heat during the soldering process.

Through-Hole Via

Normal via.

Trace Code

The system provides 32 trace codes, each with a width and clearance. "Clearance" describes the space required between pads and pads, between traces and pads, and between traces and traces. This free space is continuously checked by the real-time design rule check. Trace code specifications can be exported as part of the design rules.

Ultiroute

An advanced autorouting and autoplacement tool from Electronics Workbench that interfaces with and is accessed from Ultiboard. Offers advanced autoplacement with state-of-the-art autorouting for optimal layout of printed circuit boards. Combines grid and gridless autorouting algorithms.

Vertex

A vertex is a point of a polygon. A side of a polygon connects two vertices.

Via

A plated through-hole in a printed circuit board used to route a trace vertically (from one layer to another).

Visual Grid

Provides a visual reference for you to visually align components and traces.

Workspace

The part of the screen where you build your design.

Index

Numerics

machine trace

trace, follow me - see follow me trace

trace, manual - see manual trace

traces

    connection machine, about 7-2

    continuous placement 7-2

    deleting 7-9

    follow me router, about 7-2

    manual, about 7-2

    net belongs to 7-14

    pre-routing 9-1

    removing segment 7-2

    start and end point 7-15

U

Undo command A-4

Unlock command A-6, A-27

unused vias, deleting 10-10

updates 1-14

user settings files 3-15

V

Vertex command A-8

via properties 7-17

    attributes 7-17

    thermal relief 7-19

vias

    about 7-16

    automatically deleting when traces deleted 3-18

    deleting unused 10-10

    placing 7-16

view

    full screen 4-12

    magnifying 4-13

    shrinking 4-13

visible grid

    setting 3-23

W

wire type

test point 6-35

Wizard toolbar 3-12

workspace

    options 3-14

Technical Support and Professional Services

Visit the following sections of the National Instruments Web site at ni.com for technical support and professional services:

- **Support**—Online technical support resources at ni.com/support include the following:
  - **Self-Help Resources**—For answers and solutions, visit the award-winning National Instruments Web site for software drivers and updates, a searchable KnowledgeBase, product manuals, step-by-step troubleshooting wizards, thousands of example programs, tutorials, application notes, instrument drivers, and so on.
  - **Free Technical Support**—All registered users receive free Basic Service, which includes access to hundreds of Application Engineers worldwide in the NI Developer Exchange at ni.com/exchange. National Instruments Application Engineers make sure every question receives an answer.

    For information about other technical support options in your area, visit ni.com/services or contact your local office at ni.com/contact.

- **Training and Certification**—Visit ni.com/training for self-paced training, eLearning virtual classrooms, interactive CDs, and Certification program information. You also can register for instructor-led, hands-on courses at locations around the world.

- **System Integration**—If you have time constraints, limited in-house technical resources, or other project challenges, National Instruments Alliance Partner members can help. To learn more, call your local NI office or visit ni.com/alliance.

If you searched ni.com and could not find the answers you need, contact your local office or NI corporate headquarters. Phone numbers for our worldwide offices are listed at the front of this manual. You also can visit the Worldwide Offices section of ni.com/niglobal to access the branch office Web sites, which provide up-to-date contact information, support phone numbers, email addresses, and current events.