ISBN 1-555169-094-2

Return to Presentation

Preface

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

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

About this Manual

This manual contains a general introduction to Ultiboard, an overview of the interface, and a series of chapters that explain common functions. Several appendices contain reference-type information, including a glossary and a summary of file extensions. This is not a tutorial manual.

For installation and configuration procedures as well as an introductory tutorial on the primary steps to follow to use Ultiboard, see the *Ultiboard Getting Started and Tutorial Manual*.

The contents of this manual are also available electronically as a PDF file (`Ultiboard_Guide.pdf`). On-line help is also available — use the **Help** menu or press F1 from a screen for information on that specific screen.

This manual applies to all versions of Ultiboard. Functions that are available only in some versions are clearly marked.

Manual Conventions

For the purposes of illustration, all images in this manual are shown with black lines on a white background. In the product itself, screens show colored lines on a black background.

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

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

When this manual says to "click", it means to single-click the left mouse button.

Return to Presentation

License Agreement

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

Return to Presentation

Table of Contents

Introduction

Beginning a Design

Design Setup

Return to Presentation

Placing and Editing Components

Return to Presentation

Placing Traces

Return to Presentation

Autorouting

Preparing for Manufacturing

Postprocessing

Return to Presentation

Commands and Menus

Return to Presentation

Return to Presentation

Return to Presentation

Appendix B File Types and Extensions

Appendix C Keystroke Commands

Appendix D Extended Ultiboard Libraries

Return to Presentation

Return to Presentation

Return to Presentation

Return to Presentation

Return to Presentation

Return to Presentation

Index

Return to Presentation

Chapter 1
Introduction

Return to Presentation

Return to Presentation

Chapter 1
Introduction

1.1 About this Chapter

This chapter introduces you to the Ultiboard product and its interface. For information on installing and customizing Ultiboard and an introductory tutorial, see the *Ultiboard Getting Started and Tutorial Manual*.

1.2 What is Ultiboard?

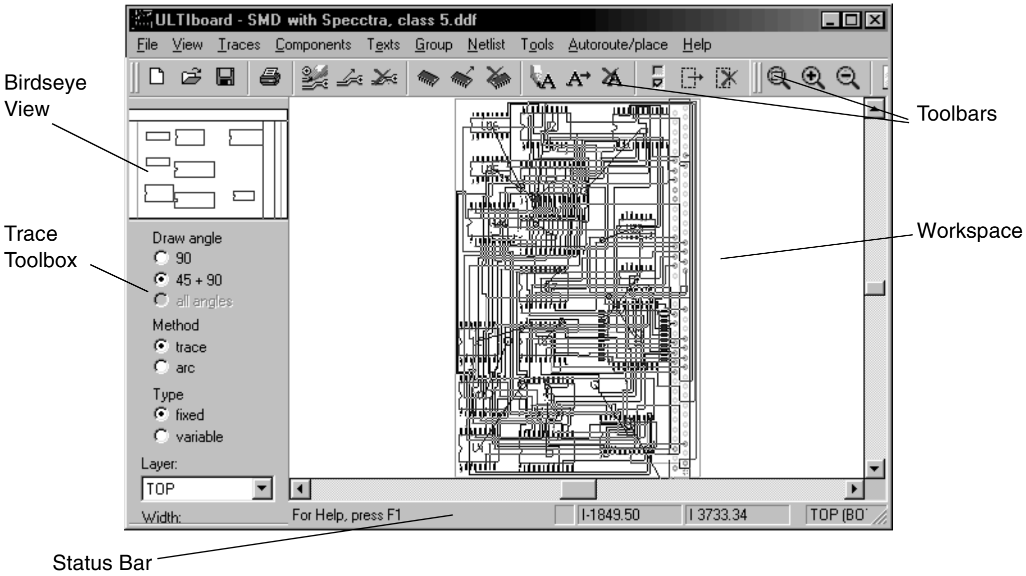
Ultiboard is a powerful printed circuit board design package from Electronics Workbench for producing high-quality, multi-layer printed circuit boards. With tight and seamless integration with the Electronics Workbench schematic capture program, you can incorporate board layout and design, and quickly bring well-designed boards to production.

Your board designs will be error-free due to the real-time online rule checking, which interactively guards against connectivity and design violations before they become costly manufacturing problems.

Return to Presentation

1.3 Introduction to the Ultiboard Interface

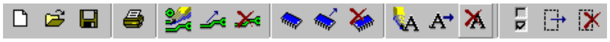
The Ultiboard interface is divided into several major work areas. You work on the PCB using the design workspace. Use the Birdseye for an overall look at the design. The Trace Toolbox provides quick access to the trace and via properties for new vias and traces to be placed.



The workspace uses different colors to indicate layers, connected/unconnected pins, vias, drills, selected items and more. Set your colors using the Layers & Colors tab of the **Tools/ Options** screen. Your color settings become the defaults for all future files. This tab is described in more detail on page 9-39.

1.4 Toolbars

1.4.1 Standard Toolbar



The Standard toolbar is displayed across the top of the application screen, below the menu bar. In addition to the menu system in Ultiboard (which is explained in “Commands and

Return to Presentation

Menus” on page 9-1), the Standard toolbar provides simplified access to the most common features. This toolbar can be moved to any location on the screen.

➢ To hide or display the Standard toolbar, choose **View/Standard.**

The Standard toolbar buttons are explained in the chart below.

	New design	Opens a new design. For details, see “Starting a Design” on page 2-1.
	Open Design	Opens an existing design. For details, see “Starting a Design” on page 2-1.
	Save Design	Saves the active design to its current name and directory. For details, see “Saving a Design” on page 2-4.
	Postprocessing	Sets print options and starts printing. For details, see “Introduction to Postprocessing” on page 8-1.
	Place Trace	Starts routing manually. For details, see “Editing Traces” on page 5-2.
	Move Trace	Moves a selected trace. For details, see “Moving, Dragging, and Deleting Traces” on page 5-9.
	Delete Trace	Deletes single segments of traces. For details, see “Moving, Dragging, and Deleting Traces” on page 5-9.
	Place Compo-nent	Lets you manually add a new component to your board. For details, see “Adding Components to Your Board” on page 4-1.
	Move Compo-nent	Moves a selected component. For details, see “Moving Components” on page 4-7.
	Delete Compo-nent	Deletes a selected component. For details, see “Deleting Components” on page 4-11.
	Place Text	Creates a text element. For details, see “Texts Menu” on page 9-17.
	Move Text	Moves selected text on the active layer. For details, see “Texts Menu” on page 9-17.
	Delete Text	Deletes selected text. For details, see “Texts Menu” on page 9-17.
	Options	Opens the Options screen, where you can set general board parameters. For details, see “Tools/Options” on page 9-26.

Return to Presentation

	Move Group	Moves a selected group around the design. For details, see “Moving Components” on page 4-7.
	Delete Group	Deletes a selected group. For details, see “Deleting Components” on page 4-11.

1.4.2 Display Toolbar

To display or hide the Display Toolbar, choose **View/Display**.

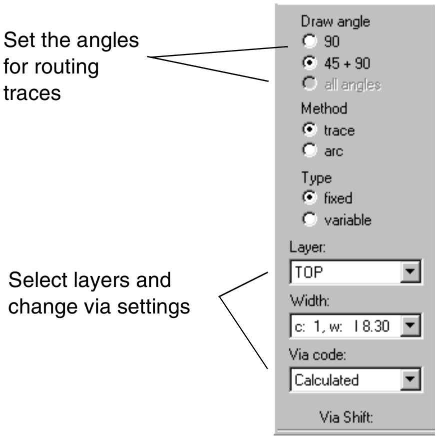
This toolbar can be moved to any location on the screen.

	Full zoom	Adjusts the size of the workspace so it displays the entire design.
	Zoom in	Zooms in on the design, providing a closer view.
	Zoom out	Zooms out on the design, providing a broader view.
	Redraw workspace	Redraws the currently active design workspace.

1.5 Trace Toolbox

Use the Trace Toolbox to set trace properties (method, type, code, drawing angles and layer) and via properties (via code and via shift).

➢ To hide or display the toolbox, choose **View/Trace**. The Toolbox can be moved to any location on the screen.



1.6 Birdseye View

The Birdseye View represents the total design area.

➢ To view a portion of the design workspace, select the area in the Birdseye View.

The selected area is shown in the Birdseye View as a white rectangle. This lets you tell at a glance exactly where your current workspace is situated.

➢ To hide or display the Birdseye View, choose **View/Birdseye**.

The view can be moved to any location on the screen.

1.6.1 Using the Birdseye View

➢ To use the Birdseye View for displaying a specific area of the design:

1. In the Birdseye View, click and hold the left button of your mouse at one corner of the area you want to view.
2. Drag the pointer to the opposite corner of the area you want to view, and release the mouse button. The area you selected is displayed in the design area of the View.

OR

1. In the Birdseye View, right-click on a point. The workspace displays the area of the design surrounding that point, with the selected point at the center.

Return to Presentation

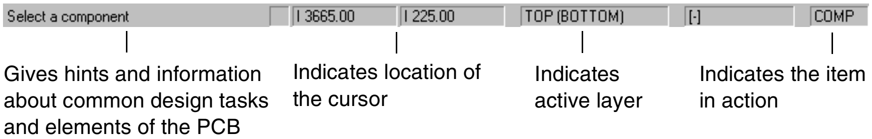
➤ To update the Birdseye View after components are moved around within the board, drag the Birdseye View to the work space area and then return it to its original location.

1.6.2 Using the Birdseye View to Zoom

➤ To change the zoom factor that affects both the Birdseye View and the workspace, press F7 (full screen), F8 (zoom in) or F9 (zoom out).

1.7 Status Bar

At the bottom of the Ultiboard screen is the status bar, which contains useful and important information.



1.8 Customizing the Interface

➤ To customize your Ultiboard interface:

- use the **View** menu to show or hide the different elements of the interface
- move any interface element (for example, the Birdseye View) to a different location on the screen
- control design preferences, such as colors associated to traces, pads, etc. and the items selected for display (using **Tools/Options**, as described in “Tools/Options” on page 9-26).

1.9 Working With Layers

Your board can be made up of between 2 and 32 layers. You control which layers are displayed by enabling the individual layers in the **View** menu. Alternatively, you can type “T” for top, “B” for bottom, or a number (the number of an inner layer, for example, “5”) to toggle that layer on or off.

Only two layers can be active at a time for routing traces. You set the active layers using **Traces/Set Active Layers**. You can easily toggle between the active layers by choosing from the Layer drop-down list in the Trace Toolbox or, while laying a trace, by pressing F2.

Return to Presentation

Chapter 2
Beginning a Design

Return to Presentation

Return to Presentation

Chapter 2
Beginning a Design

2.1 About this Chapter

This chapter explains the basic operations of working with Ultiboard.

This chapter assumes that you have installed and configured Ultiboard and now want to work on a design. For information on installing and configuring Ultiboard, see the *Ultiboard Getting Started and Tutorial Manual.*

2.2 Starting a Design

Ultiboard lets you start a design file in several ways.

2.2.1 Opening an Existing Design File

➢ To open an existing Ultiboard file:

1. Choose **File/Open**. A standard file selector screen appears. Navigate to the desired folder and, from the file list, select the Ultiboard design file (`.ddf`) you want.
2. Click **Open**. The design appears in the Ultiboard workspace.

The bottom of the **File** menu displays the names of the four most recently opened files. Click on one of these files to open it.

Only one file is open in Ultiboard at a time. You are prompted to save changes to an open design before it is closed or are prompted with the Save As screen before you exit an untitled design. If you close a design without saving it, you lose all changes made since last saving it or, if you have never saved it, you lose all design details.

If attempts to open your design file (`*.ddf` format) fail with the error message “Error during disk reading and writing”, the **Customer** field in the Board Settings tab is blank. The solution

Return to Presentation

is to edit the `.ddf` file in a text editor such as Notepad. Modify the first line by typing in a few characters after the "*P", for example: *Pabcd. The same error message appears if you try to load `.net` and/or `.plc` files using the **Open** command instead of the proper **Import** command.

2.2.2 Retrieving a Design from Electronics Workbench Schematic Capture

Electronics Workbench Schematic Capture is the schematic capture and simulation program from Electronics Workbench.

➢ To open a file from Electronics Workbench Schematic Capture:

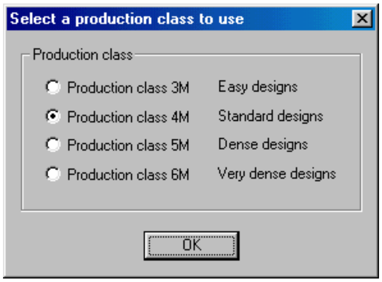
1. Choose **File/Open**. A standard file selector screen appears.
2. In the **Files of Type** list, choose **All files**. Navigate to the desired folder and, from the file list, select the Electronics Workbench Schematic Capture file (`.ewb`) you want.
3. Click **Open**. The circuit appears in the Ultiboard circuit workspace.

Schematics are imported to Ultiboard using a combination of .plc and .net files. Differences between the old and new netlists are saved in a text file in the same folder as the netlists, with the file name *`name`*`.dif`, where *`name`* is the netlist file name. You can display this file using **Netlist/Compare Netlist**.

2.2.3 Starting a New Design File

➢ To start a new design file:

1. Choose **File/New**. The Production class screen appears.



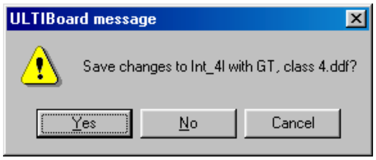
Return to Presentation

2. Select the production class you want. The production class defines the level of complexity and rule checking, and ultimately influences the parameters which determine the cost of production. For more on production classes, see “About Production Classes (Technology Files)” on page 3-1.
3. Click **OK**. An empty Ultiboard circuit board workspace appears, using the production class you chose.

If you already have an Ultiboard design open when you select **File/New**, it is closed. You are prompted to save changes to an open design before it is closed or are prompted with the Save As screen before you exit an untitled design. If you close a design without saving it, you lose all changes made since last saving it or, if you have never saved it, you lose all design details.

2.3 Closing a Design

➢ To close your design, choose **File/Close**. If you have unsaved changes in the design, you are prompted to save the file.



If you close a design without saving it, you lose all changes unless you have enabled **Automatic Save** in the System Settings tab of **Tools/Options**. If you have never saved your design, you lose all design details.

Ultiboard automatically creates a new design after closing a design. You can also use the **File/New** command to create a new design.

Once all changes have been saved, the design file closes.

2.4 Backannotation to Electronics Workbench Schematic Capture

Backannotation is a highly automated process which ensures that modifications made to an Ultiboard design are transferred to the board’s schematic in the Electronics Workbench Schematic Capture program. This process helps keep your schematics and board layouts consistent with one another.

Backannotate after renumbering your board.

➢ To backannotate your revisions:

Return to Presentation

1. Save and close your design in Ultiboard.
2. Open Electronics Workbench Schematic Capture.
3. Choose **File/Open** and give the name of your design file. Your design opens in the Electronics Workbench Schematic Capture screen.
4. Choose **Tools/Backannotate**. The backannotation starts automatically, and a report on changes appears.
5. Generate the output again to see changes caused by powerplanes.
6. Choose **File/Close** to exit Electronics Workbench Schematic Capture.

Backannotation is an important feature of CAD design software. Pin and gate swaps, as well as component renaming or renumbering, cause inconsistencies between the schematic and the PCB design. Backannotation can overcome these inconsistencies. To backannotate, Electronics Workbench Schematic Capture 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 Electronics Workbench Schematic Capture. The following changes can be backannotated:

- pin swaps (only when the pin swap command is used)
- gate swaps (only when the gate swap command is used)
- component renumbering
- component renaming

2.5 Saving a Design

To save your design, choose **File/Save**. If you have not saved the file before, the Save As screen appears, to let you choose a folder and enter a file name.

Chapter 3
Design Setup

Design Setup

Return to Presentation

Design Setup

Return to Presentation

Chapter 3
Design Setup

3.1 About this Chapter

Effective PCB design is not just a matter of drawing traces and ensuring that you make all the connections correctly. You also require the correct design rules such as pad sizes, via sizes, and trace sizes. These design rules determine the production class of the board. In general, the higher the production class, the higher the manufacturing cost. This chapter describes how to set up your design to get the most out of Ultiboard's powerful features.

3.2 About Production Classes (Technology Files)

Ultiboard makes it easy for you to design your PCB for the production class of your choice because it uses predefined design rules for each of the supported production classes. Ultiboard also warns you if you use trace sizes or pad sizes that require a higher (and more expensive) production class.

Ultiboard is preconfigured with design rule settings for the four production classes used most commonly in PCB manufacturing. The production classes that Ultiboard supports are:

- Class 3M — used for easy designs
- Class 4M — used for standard designs
- Class 5M — used for dense designs
- Class 6M — used for very dense designs

Ultiboard uses Production Class 4M design rules by default because it is the most common class, and most PCB manufacturers are able to manufacture boards to this class. This class is also well suited for prototype boards.

To get the best routing results, Ultiboard uses class-specific settings for the trace width, pad size, clearance, component placement grid, and routing grid. These setting are summarized in the table below:

	Class 3M (easy)	Class 4M (standard)	Class 5M (dense)	Class 6M (very dense)
Trace width (code 1)	12.5	10	8.3	6.25
Trace clearance	11.5	9	7.3	5.2
Pad width (code 1)	60	50	58.3	60
Pad clearance	11.5	9	7.3	5.2
Component grid	50	20	50	50
Routing grid	25	20 (10*)	16.67	12.5 (6.25*)
Via width	40	35	28	22
Hole size	23.33 (0.6mm)	19.69 (0.5mm)	15.74 (0.4mm)	11.67 (0.3mm)

*Sub-grid

Note The settings built into the class definitions replace all settings saved in the original circuit design. This is done to get the best results with both manual trace editing and the rip-up and retry autorouter.

3.3 Setting Units of Measurement

Ultiboard supports both metric and imperial units of measurement. All displayed values and measurements are prefaced with either an “M” (for metric) or “I” (for imperial). Metric measurements are given in millimeters, while imperial measurements are given in mils (1 mil = 0.001 inches).

Part of Ultiboard’s flexibility comes from its ability to switch back and forth between the two measurement systems while you design. You can, for example, place parts using metric measurement, yet define your board outline using imperial values.

➢ To switch between metric and imperial measure:

1. Choose **Tools/Options**. The Options screen appears.
2. In the Options screen, click the System settings tab.
3. Choose the appropriate unit option, and then click **OK.**

When you change the unit in which you are working, Ultiboard changes each measurement in your current project to the equivalent measurement in the new unit.

Tip You can use shortcut keys to switch back and forth between metric and imperial measure. Press ALT+I to change to imperial measure, and ALT+M to change to metric measure.

3.4 Setting Grids

A grid defines the fixed locations on which you can place components and traces. The smaller the internal grid of your board, the closer together you can place components and traces. Selecting a small grid for your board may have some or all of the following effects:

- Increase in manufacturing costs — advanced manufacturing technology may be needed.
- Difficulty manipulating traces — the pointer may be difficult to move with precision on a fine grid.
- Slow processing time — the increased number of necessary computations may cause the autorouter and other advanced algorithms to run more slowly and consume more memory than normal.

Ultiboard offers two grids: the mouse grid, which affects your placement of components and traces, and the visual grid, which you can use to visually orient your components and traces.

3.4.1 Setting the Mouse Grid

The mouse grid settings define the points along which the mouse cursor moves on your screen. The precision with which you can place components and traces depends on the size you assign to the mouse grid.

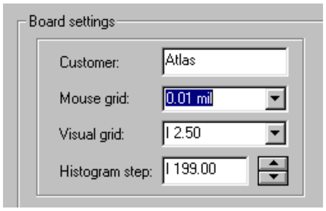
When you first start designing, you can use a large mouse grid for the hand control needed for component placement purposes, and then reset the mouse grid to allow fine placement of traces.

➢ To change the mouse grid size:

1. Choose **Tools/Options**. The Options screen appears.
2. In the Options screen, click the Board settings tab.

Return to Presentation

3. Select a grid size from the **Mouse grid** drop-down list or enter the grid size of your choice.



4. Click **OK**.

3.4.2 Setting the Visual Grid

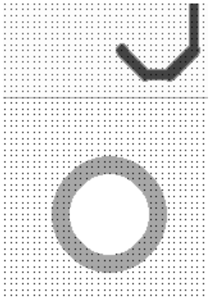
The visual grid is a set of regularly spaced points in your main workspace against which you can orient components and traces. You can switch the visual grid on or off, and adjust the spacing of the grid points according to your preferences.

➢ To switch the visual grid on and off:

1. Choose **Tools/Options**. The Options screen appears.
2. In the Options screen, click the View Items tab.
3. Enable the **Visual grid** option.
4. Click **OK**. The visual grid is now active in your workspace.

➢ To change the visual grid size:

1. Choose **Tools/Options**. The Options screen appears.
2. In the Options screen, click the Board Settings tab.
3. Select the desired grid size from the **Visual grid** drop-down list, or enter a grid size.
4. Click **OK**.



The edge of a component and pins, set against a 2.5 mil visual grid, seen under high magnification.

Note If you are using a fine visual grid setting, or viewing the workspace at low magnification, you may not be able to see the visual grid. Increase the zoom setting to see fine visual grids.

3.5 Defining the Board

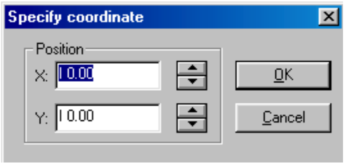
On start-up, Ultiboard displays an empty blue border in the workspace. The border represents the board size at its default size of 160mm X 100mm. To define the clearance around the board outline, see "Setting Clearance Values" on page 3-16.

Ultiboard allows you to create board outlines in several different ways depending on the shape and complexity of the board you need. For basic boards, a rectangular shape is the quickest to define. For more complex boards, use the board editor to create custom shapes.

3.5.1 Defining a Board Outline as a Rectangular Shape

➢ To define a rectangular board outline (the easiest way to define the shape of a new board):

1. Choose **Tools/Board outline/Define by rectangle**.
2. Define the board size by using the mouse or entering coordinates directly.
   - Using the mouse: In the workspace, click where you want to place the first corner of your board. Then, move the cursor to where you want to place the opposite corner of your board. When the board shape and size is correct, click again to set the board in the workspace.
   - Entering coordinates: Press the asterisk key (*) on the numeric keypad to open the Specify coordinate screen.



   Type the X and Y coordinates of the first corner of the board, and click **OK**. Press the asterisk key on the numeric keypad again to reopen the screen. Now type the X and Y coordinates of the opposite corner, and click **OK**.

Return to Presentation

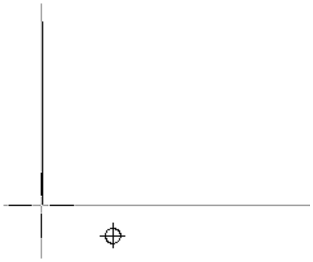
3. Once you set the dimensions, Ultiboard prompts you to delete the old board shape:
    - If you click **Yes**, Ultiboard deletes the old board and replaces it with the one you defined.
    - If you click **No**, your new definition is discarded and the old definition remains.

3.5.2 Defining a Board Outline as a Polygon

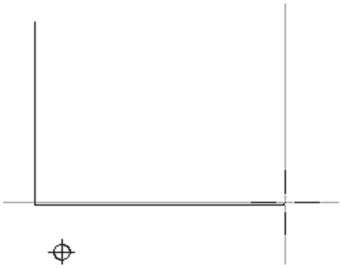
Using a polygon is a more complex way of defining a board shape than using a rectangle. You can create whatever polygonal shape is necessary for your application.

➢ To define a polygonal shape:

1. Choose **Tools/Board outline/Define by polygon**.
2. Move the pointer to the place in the workspace where you want to begin defining the board shape. Click to mark the first vertex of the board.
3. Position the cursor where the next vertex should be placed, and click to mark the vertex.
4. Mark all remaining vertices. Ultiboard draws the outline as you work.
5. Complete the shape by joining the final segment of the outline to the first vertex and clicking the mouse.

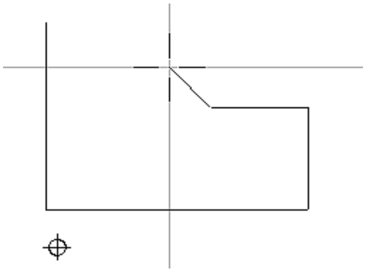


First vertex

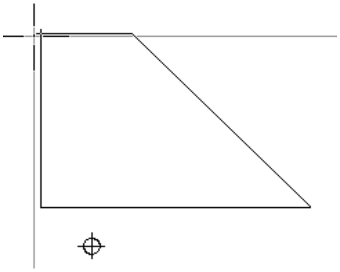


Second vertex

Return to Presentation



Add as many as you need



Join the last segment to the first

6. When the shape is complete, Ultiboard prompts you to delete the old board shape:
    - If you click **Yes**, Ultiboard deletes the old board and replaces it with the one you defined.
    - If you click **No**, your new definition is discarded and the old definition remains.

Note When you define a board outline as a polygon, you must create a closed polygon shape. Therefore, the end point of the shape must connect to the beginning point.

Tip If you are having difficulty matching the start and end points, try using a larger grid size.

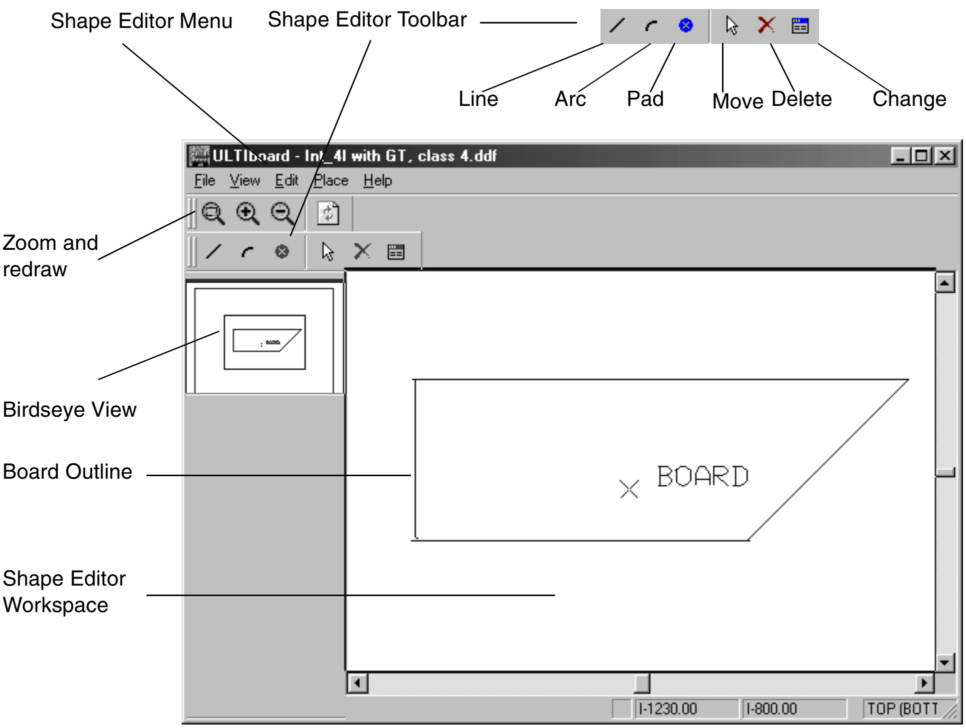
Return to Presentation

Design Setup

3.5.3 Defining a Board Outline of Any Shape

In addition to defining a new board outline as a rectangle or a polygon, you can also define it using any combination of line segments and curves that you choose. To do this, use the Shape Editor.

Choose **Tools/Board outline/Edit**. Ultiboard opens the Shape Editor screen.



➢ To place a line segment in your new board outline:

1. Click the **Line** button on the toolbar, or choose **Place/Line,** and move the mouse pointer to the place in the workspace where you want to begin drawing the line.
2. Click to anchor the first vertex of the line.
3. Move the mouse pointer and click to anchor each successive vertex. Every time you anchor a point and move the mouse cursor, a “rubber band” line appears, to show you the path of the next segment. The segment is drawn when the next segment is anchored. Right-click, or press ESC, to cancel the line command.

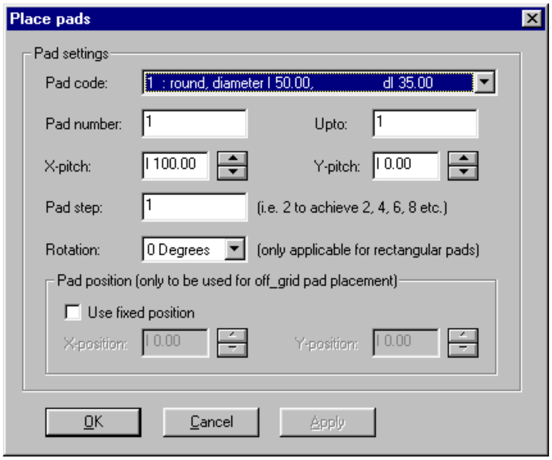
Return to Presentation

➢ To place an arc segment in your new board outline:

1. Click the **Arc** button on the toolbar, or choose **Place/Arc**, and move the mouse pointer to where you want to begin drawing the arc.
2. Click to anchor the first point of the arc.
3. Move the cursor to the origin point of the arc radius (the arc's centre point). Click to anchor.
4. Define the arc angle by moving the pointer. Click to complete the arc.

➢ To place a pad:

1. Click the **Pad** button on the toolbar, or choose **Place/Pads** to open the Place pads screen.



2. Select the pad code and the pad rotation for the pads you want to place.

Note The default pad code is defined through **File/Preferences**.

3. If you want to place a single pad:
   - Enter a number or name for the pad in the **Pad number** field. Ensure that the **Upto** field contains the same number.
   - To place the pad using the mouse, just close the screen, or enable the **Use fixed position** option and specify the X-Y coordinates to be used.
4. If you want to place a row of pads at equal distances from each other:
   - Specify a number or name for the first pad in the **Pad Number** field.
   - Specify a number or name for the final pad in the **Upto** field.
   - Specify the distance between the pads using the **X-pitch** and **Y-pitch** fields.
   - Specify the pad step in the **Pad step** field. For example, if you want to number the pads in increments of two, enter "2".

Return to Presentation

- To anchor the position of the first pad in the row using the mouse, just close the screen, or enable the **Use fixed position** option and specify the X-Y coordinates to be used as the anchor.

5. To temporarily apply your changes, click **Apply**. To apply your changes and close the screen, click **OK**. To cancel your changes, click **Cancel**.

Note When placing a row of pads:

- The pad number in the **Pad number** field must contain an integer value.
- The pad number in the **Upto** field must be an integer value greater than the integer value in the **Pad number** field.
- To place a row of pads from the right to the left or from the top to the bottom use negative values in the **X-pitch** and **Y-pitch** fields.
- The **Pad step** field cannot contain a negative value.

For example:

Pad number	Upto	Pad step	Placed pads
1	5	2	1, 3, 5
A1	7	3	A1, A4, A7
4B	7	1	4B, 5B, 6B, 7B

➢ To move a shape object you have already placed:

1. Click the Move object button on the toolbar, or choose **Edit/Move,** and click on the object you want to move. The object is highlighted and attaches itself to the pointer.
   - To rotate the object, press F2.
   - To move the object, move the mouse or press the asterisk key (*) on the numeric keypad and enter the coordinates.
2. Click to anchor the object temporarily. A temporarily anchored object is highlighted in green.
3. Right-click to anchor the object permanently.

➢ To delete a shape object you have already placed:

1. Click the Delete object button on the toolbar, or choose Edit/Delete, and then click on the object you want to delete. The selected object is highlighted.
2. Ultiboard prompts you to confirm the deletion.
   - If you click **Yes**, Ultiboard deletes the selected object.
   - If you click **No**, the object is not affected but the delete function remains active.

Return to Presentation

3.6 Editing an Existing Board Outline

In addition to defining new board outlines, the Shape Editor can also edit existing outlines. To edit an existing outline, load the outline into the main Ultiboard workspace and then choose **Tools/Board outline/Edit**. The shape is transferred into the Shape Editor automatically and may be edited using Shape Editor commands. For more information on the Shape Editor and the Shape Editor commands, see "The Shape Editor" on page 9-46.

3.7 Loading a Board Outline from a File

If you have previously saved a complete design in Ultiboard and wish to reuse the board outline, you can import the board shape without loading any of the components or traces. This feature allows you to reuse custom-designed outlines without having to define the board from scratch each time.

➢ To load a board outline from an existing design file:

1. Choose **Tools/Board outline/Import from file**.
2. When Ultiboard prompts you to select a design file, select the appropriate file and click **OK**. Ultiboard extracts the board outline from the design file and centers it in the workspace.

3.8 Setting the Board Layers

Ultiboard lets you define boards between 2 and 32 layers thick. Before you can create multilayered boards, you need to know how they are manufactured. This knowledge will help you decide what kind of layering to use in your design.

First, some terminology:

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 feed-through via — any via that connects all layers, top, bottom, and internal.

Some examples of multi layered boards include:

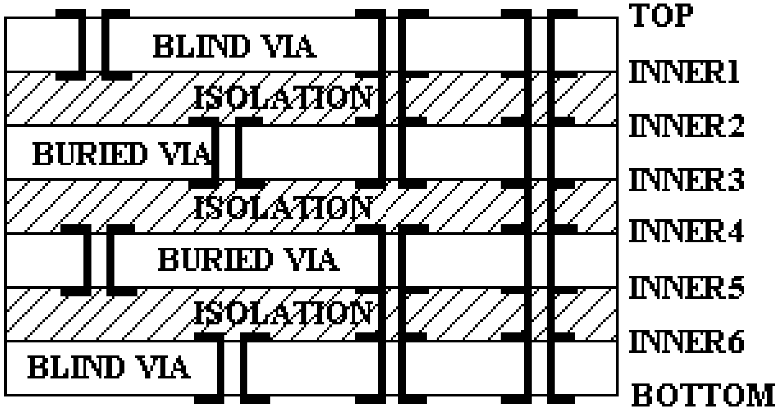
- 4-layer with normal feed-through vias
- 6-layer with full blind and buried via support

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.

Return to Presentation

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.

All multi layer boards are built in sets of two layers called *bi-layers*. As an example, consider an 8-layer board. Ultiboard counts the layers and bi-layers as follows:



The shaded areas represent the isolation between the bi-layers. From this illustration, you can see which vias are physically possible. To create the vias seen here, the manufacturer drills the first bi-layer and then the second bi-layer, bonds these two bi-layers together, drills the glued assembly, drills the third layer, and so on until the layering is complete.

Because of the way multi-layered boards are manufactured, Ultiboard needs a reference.

➢ To choose the number of layers and set the layer lamination sequence:

1. Choose **Tools/Options**.
2. In the Options screen, display the Board settings tab.
3. Choose the appropriate number of layers from the drop-down list.
4. In the **Layer lamination sequence** field, enter the appropriate drilling and bonding sequence for your board.

   As an example, consider an 8-layer board. By default, Ultiboard assigns the following lamination sequence: (|+|+|+|). This sequence specifies that all layers should be bonded together, and then drilled and metallized as a single unit. In other words, if you want to make a connection between Inner 2 and Inner 3, the via will be drilled through the entire board. This approach is cost-effective, but has the disadvantage of placing vias on layers where they are not needed.

   You can choose a more costly, yet efficient, design by changing the default lamination sequence to another configuration such as (((|)+(|))+((|)+(|))). This specification states: drill and metallize bi-layers one through four individually, then glue layer one to layer two

Return to Presentation

and layer three to layer four to create two bonded packs. Drill and metallize these two packs. Finally, bond the two half products together and drill and metallize for the final time.

The result is an 8-layer board with full blind and buried via support.

5. When the number of layers and lamination sequence is correct, click **OK**.

3.9 Setting Design Rules

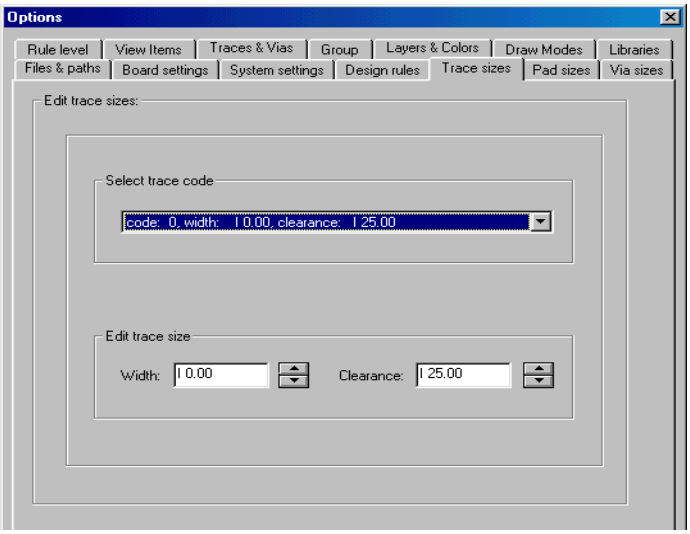
Ultiboard lets you set the sizes of traces, vias and pads, as well as define clearance values. This section explains how to set such parameters as well as how to use Ultiboard’s design rule check.

3.9.1 Setting Trace Widths

Ultiboard lets you define up to 32 trace widths for each design.

➢ To define the trace widths:

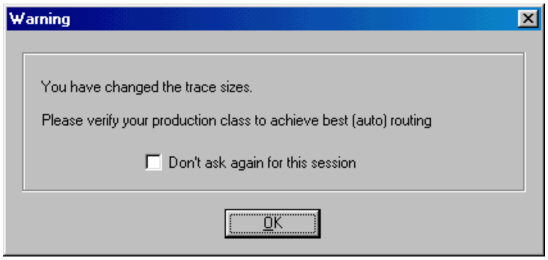
1. Choose **Tools/Options**. The Options screen appears.
2. In the Options screen, display the Trace sizes tab.



3. Choose the trace code you want to edit from the **Select trace code** drop-down list.

Return to Presentation

4. Click the arrows beside the **Width** field to increase or decrease the trace width. Optionally, you may type the new width directly into the **Width** field.
5. Click the arrows beside the **Clearance** field to increase or decrease the spacing between the trace and any other object placed on the board. Optionally, you may type the new clearance directly into the **Clearance** field.
6. To temporarily apply your changes, click **Apply**. To apply your changes and close the screen, click **OK**. To cancel your changes, click **Cancel**. If you click **Apply** or **OK**, Ultiboard prompts you to verify your production class.



7. To close this screen, click **OK**.
8. To close the Options screen, click **OK**.

Note You can define keep-out areas on your board by defining a trace with a zero width, but with a large clearance. A trace with zero width will not be plotted and will not result in any copper on your board, but will not allow an autorouter to pass over it.

3.9.2 Setting Pad Sizes

You can work with up to 240 different pad sizes in each design. Ultiboard supports the following pad shapes (with or without drill holes):

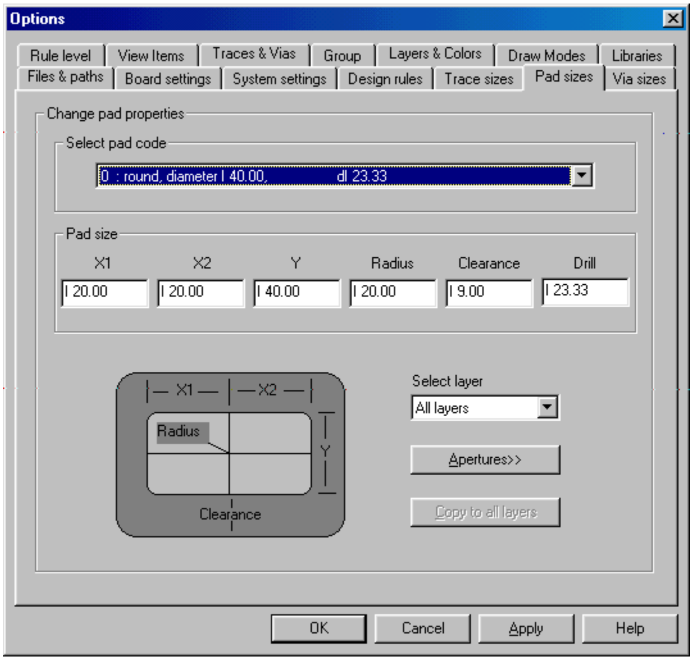
- round
- square
- rectangular
- rectangular or square with round corners.

➢ To define a pad:

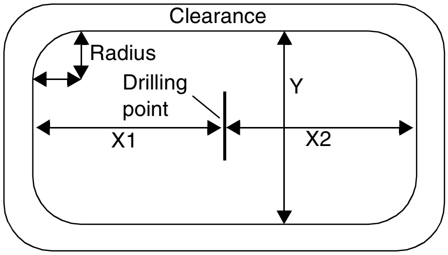
1. Choose **Tools/Options**. The Options screen opens.

Return to Presentation

2. In the Options screen, display the Pad sizes tab.



3. Choose the pad code you want to edit from the **Select pad code** drop-down list. Note that pad codes 0-100 and 169-239 are used by shapes in the Ultiboard components library. Avoid making changes to these pads as they might affect library shapes on a global basis.
4. The pad size contains the parameters shown in the diagram below:



As necessary, edit each of the following parameters in the appropriate fields:

- **X1** — the width of the left side of the pad, as measured from the drill hole
- **X2** — the width of the right side of the pad, as measured from the drill hole
- **Y** — the height of the pad
- **Radius** — the radius of the pad corners

- **Clearance** — the width of the area around the pad in which no other objects may be placed
- **Drill** — the drill diameter size

5. From the **Select layer** drop-down list, choose the layers to which you want your pad modifications to apply.
6. If you are using non-standard apertures, click **Apertures**. Make appropriate changes to the Aperture codes screen and click **OK**.
7. To temporarily apply your changes, click **Apply**. To apply your changes and close the screen, click **OK**. To cancel your changes, click **Cancel**.

Note The **Aperture** button is for advanced, non-standard applications only.

3.9.3 Setting Via Sizes

You can work with up to 16 via sizes in each design. Via setup is similar to pad setup as described in "Setting Pad Sizes" on page 3-14.

➢ To define a via:

1. Choose **Tools/Options**. The Options screen appears.
2. In the Options screen, display the Via sizes tab.
3. Define the via sizes in the same way as you defined pad sizes in the previous section, or see "Via Sizes Tab" on page 9-33.

3.9.4 Setting Clearance Values

In addition to clearance values for pads, vias, and traces, you can also set clearance values for the entire board.

➢ To set clearance values for the entire board:

1. Choose **Tools/Options**. The Options screen opens.
2. In the Options screen, display the Board settings tab.
3. Click the arrows beside the **Clearance** field. Optionally, enter a new clearance value directly into the field.
4. To apply the values, click **OK**.

Return to Presentation

3.9.5 Running a Design Rule Check

Ultiboard checks the spacing between objects to ensure there are no design rule violations. It also checks to ensure that only pins that belong to a common net are connected.

You can perform either real-time or batch design rule checks on your boards.

3.9.5.1 Real-Time Checks

The real-time design rule check can run in three different modes:

- Real-time full check

  In this mode it is impossible for you to commit a design rule violation. For example, Ultiboard will not allow you to place a trace across another trace if those traces are from separate nets. When you commit a design rule violation, your computer gives you an audible warning and marks the place where the violation occurred with a small circle.

  It is recommended that you always use this real-time full check because it detects design rule violations as they are committed. The sooner you able to detect an error, the easier it is for you fix it. If you use this mode, you ensure that your boards are error free before they go into production.

- Real-time overrule

  Real-time overrule is similar to the real-time full check, except that it allows you to continue even if an error is found.

- Real-time disabled

  This option disables Ultiboard*'s* design rule checking. Without real time checking, it is especially important to run frequent batch checks on your board to locate errors.

➢ To choose between real-time full check, real-time overrule, and real-time disabled:

1. Choose **Tools/Options**. The Options screen opens.
2. In the Options screen, display the Rule level tab.
3. Enable the appropriate design rule check option and click **OK**.

3.9.5.2 Batch Checks

In contrast to real-time checks, batch checks are intended as a final design check before your design goes into production. When you run this option, Ultiboard places small circles at all locations where violations occur. Ultiboard also generates a summary file detailing each error. This file uses the same name as your design, with an `.err` extension. Batch check allows you to perform a design rule check on your entire board or on a single net.

Design Setup

Return to Presentation

➢ To do a design rules batch check:

1. Choose **Tools/Design Rule Check**. The Design Rule screen appears.
2. Select **Entire Board** or **Selective net**.

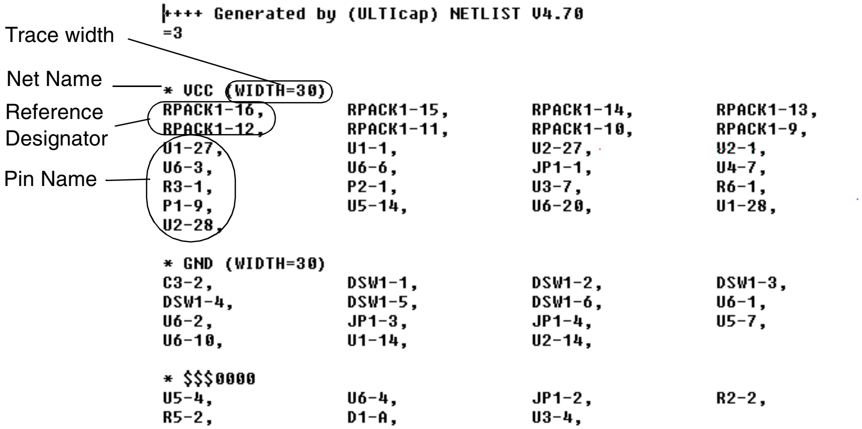
For more detailed information on Design Rule Check, see “Checking for Errors” on page 7-1.

3.10 Understanding .net and .plc files

Electronics Workbench Schematic Capture and other schematic capture programs generate both a `.net` file and a `.plc` file. The `.net` file contains information about a given board’s nets, its pins and its connections. The .plc file contains information about components. Some designers edit these files to add new components and change reference values before loading/ importing them into Ultiboard.

.net File Example

The following shows the contents of a `.net` file:



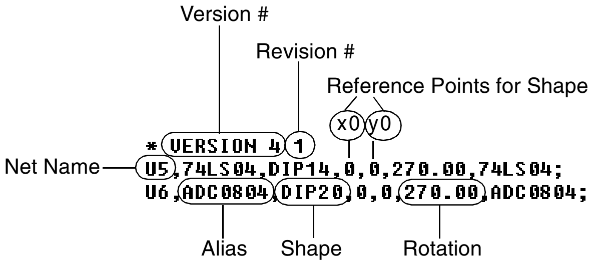

```
++++ Generated by (ULTIcap) NETLIST V4.70
=3

* VCC (WIDTH=30)
RPACK1-16,      RPACK1-15,      RPACK1-14,      RPACK1-13,
RPACK1-12,      RPACK1-11,      RPACK1-10,      RPACK1-9,
U1-27,          U1-1,           U2-27,          U2-1,
U6-3,           U6-6,           JP1-1,          U4-7,
R3-1,           P2-1,           U3-7,           R6-1,
P1-9,           U5-14,          U6-20,          U1-28,
U2-28,

* GND (WIDTH=30)
C3-2,           DSW1-1,         DSW1-2,         DSW1-3,
DSW1-4,         DSW1-5,         DSW1-6,         U6-1,
U6-2,           JP1-3,          JP1-4,          U5-7,
U6-10,          U1-14,          U2-14,

* $$$0000
U5-4,           U6-4,           JP1-2,          R2-2,
R5-2,           D1-A,           U3-4,
```

.plc File Example

In the **.plc** file, the component’s reference designator, alias, shape, coordinates, angle of rotation and number of pins are listed.

Return to Presentation

The following shows the contents of a `.plc` file generated by Electronics Workbench Schematic Capture:



3.11 Identifying Netlist Changes Since Previous Import

The **Netlist/Compare Netlist** command compares a netlist file on disk with the current netlist in memory. This is a necessary tool to keep your designs in synchronization. The default extension for the netlist file on disk used for this comparison is `.net`.

All differences between the compared netlists are saved in a text file on disk with the extension .dif. The filename equals the filename of the netlist from disk. This text file is saved in the same directory as the netlist from disk.

Compare netlist is useful for two reasons:

- It shows how the nets in an active design have changed since the last netlist file (`.net`) was created.
- It shows the changes made to the schematic since the last netlist file was created.

Return to Presentation

Return to Presentation

Chapter 4

Placing and Editing Components

Return to Presentation

Return to Presentation

Chapter 4
Placing and Editing Components

4.1 About this Chapter

This chapter explains the various ways you can place components on your board using Ultiboard, and your options for editing components once they are placed. In addition, you may want to consider the Ultiroute product from Electronics Workbench, which offers fully automated autoplacement capabilities.

4.2 Adding Components to Your Board

Netlist and component information is imported automatically when a design is transferred from Electronics Workbench Schematic Capture. If you need to add components, you can do so by either retrieving the component from the component library or by importing a component list file.

4.2.1 From Component Libraries

Use component libraries for components that you need on your board, but which have no equivalents in your schematic capture (e.g. mounting holes).

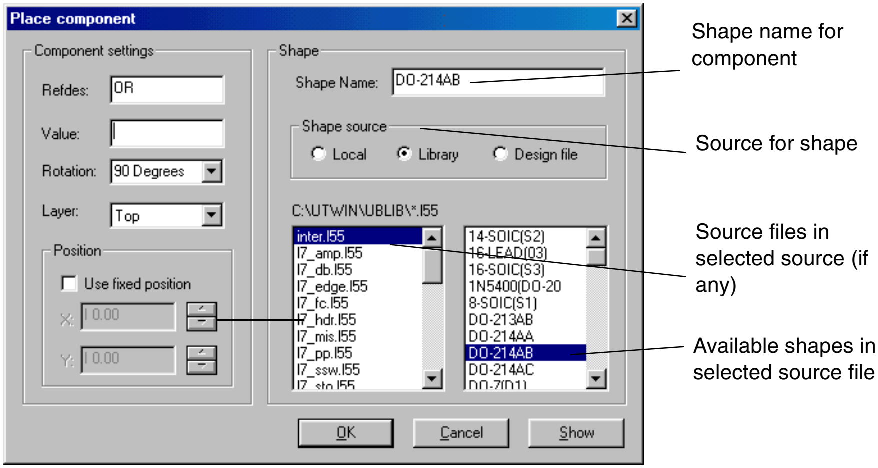
You can choose components from three sources:

- local — components in your current design
- library — Ultiboard shapes libraries
- design file — other designs that you have created but that are not the active design.

Return to Presentation

➢ To import components from the libraries:

1. Click the Place Component button on the toolbar or choose Components/Place. The Place component screen appears. :



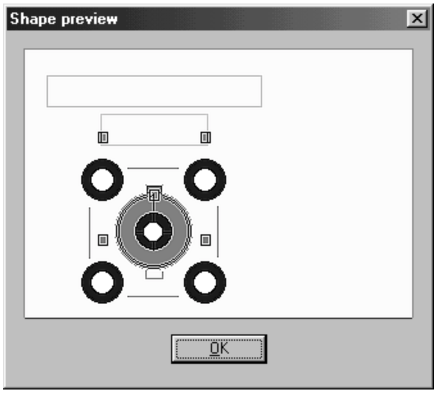
Refdes	(Component reference designator) The reference designator of the component. Required.
Value	The value of the component.
Rotation	The rotation of the component. The rotation can be selected from the drop-down list or entered manually.
Layer	The layer (top or bottom) where the component should be positioned.
Position	The default starting position for the component. The reference point of the component (i.e. the reference point of the selected shape) is placed at the selected position. The X- and Y- coordinates are given in the default unit of measurement.
Shape	The component shape from the selected library file (shape source option set to library) or from the selected design file (shape source option set to design file).
Shape source	The source (local, library or design file) from which the component shape should be selected.
Show Button	Shows a preview of the currently selected component shape.

➢ To select the desired component/shape:

1. From the options listed under **Shape source**, enable one of the sources.

Return to Presentation

2. If you chose **Library** or **Design file**, the left file list shows the available source files in that source. Select the desired source file, and the shapes in that file appear in the right file list.
3. Select the desired shape from the right file list.
4. To preview the shape, click **Show**. The Shape preview screen appears, showing details of the component.



To close the Shape preview screen, click **OK**.

5. When you locate the component you want, enter a reference designator and a value for the component in the **Refdes** and **Value** fields of the Place component screen.
6. Click **OK**. The Place component screen closes and you return to the board design screen.
7. Make sure that the layer you want the component on is the active layer in the design.
8. Position the component, visible in ghost-line, where you want it on the board. Use the x/y coordinates on the status bar to help position the component.
9. Click to drop the component into place. The Place component screen reappears immediately.
10. Click **OK** to place the same component again or choose another component. Click **Cancel** to close the screen and return to the design.

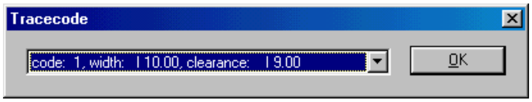
4.2.2 From Component List Files

A component list file (`.plc`) or a netlist file (`.net`) can either be created by you, as explained in "Understanding .net and .plc files" on page 3-18, or can be created by Electronics Workbench Schematic Capture.

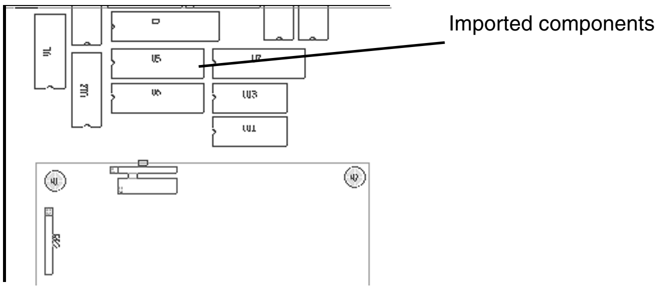
Return to Presentation

➤ To import a component list file:

1. Choose **File/Import/Components**. A standard file selector screen appears.
2. Select the file you want and click **OK**. You are prompted to confirm the overwrite of shapes in the active design. Any existing shapes in your design that have the same reference designator (e.g. U2) as the shapes you are importing will be overwritten.
3. Click **Yes**. You are prompted to confirm that you want to load the netlist file corresponding to the `.plc` file.
4. Click **Yes** to import the file. A Tracecode screen appears.



5. Choose the tracecode that you want from the drop-down list and click **OK**.
6. All components in the list file are placed around the outside of the board outline. In the illustration below, they are shown above the actual board.



4.3 Adding Component Connections to Your Board

The `.net` file should have all the connectivity information for the components that you imported through the `.plc` file. If your design came directly from Electronics Workbench Schematic Capture, then the `.net` file was automatically loaded as well. Pin connectivity is covered more thoroughly in the following section.

Return to Presentation

4.3.1 About Component Connections

A net is a collection of connected pins (“net” is short for “network”). A net is usually assigned a name and carries information on all the pins to which it connects.

A net***list*** is the conceptual expression of logical connectivity between pins, but it has no physical counterpart on the circuit board. Nonetheless, it is essential to define netlists in Ultiboard so that the program knows which pins should be connected. Pins are physically connected when a trace is placed between them.

Ultiboard uses the netlist information to:

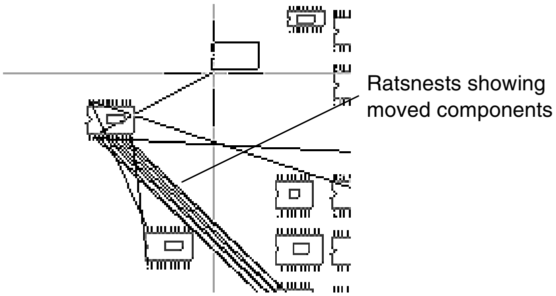
- warn you when you connect a trace between pins that are not part of the same net
- inform the autorouter of where it needs to place traces when autorouting.

Remember, a netlist expresses electrical/logical, not physical, connections between pins.

4.3.2 Displaying Ratsnests

A ratsnest is a straight line connection between pads or pins, indicating their connectivity. The ratsnest identifies the pins/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, appearing when you move a component.

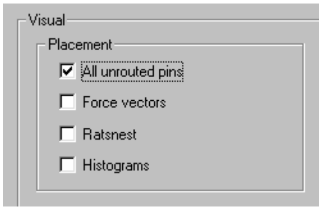


You can display the ratsnest for the whole board, or only for a selected component. If you display the information for a board that contains many components, the mass of ratsnest lines just becomes a blur.

Components

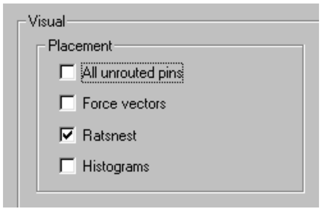
Return to Presentation

➢ To display the ratsnest information for the whole board, choose **Tools/Options**, display the View Items tab, and enable **All unrouted pins**.



The ratsnest information for the entire board appears.

➢ To display the ratsnest for just a single component, choose **Tools/Options**, display the View Items tab, and enable **Ratsnest**.



Ratsnest information will be displayed for each part when it is moved. For details, see "Moving Components" on page 4-7.

Note To make physical connections between pins/pads, you must place traces. For more information on placing traces, see "Drawing Traces" on page 5-4.

4.3.3 Adding Pins to a Net

In some cases you will need to add components or pins that were not part of an existing net.

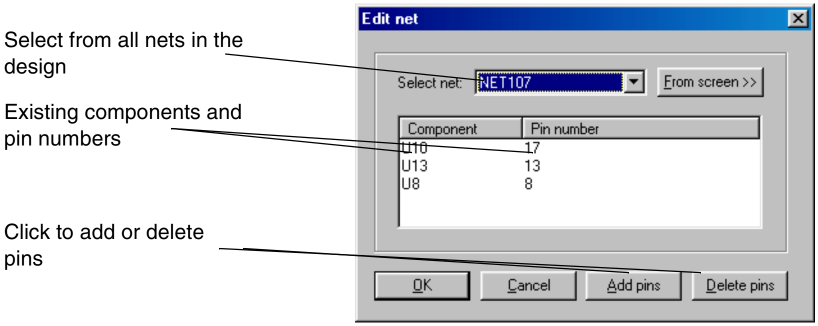
➢ To add pins:

1. Click on the component pins you want to add to a selected net. The component pins that belong to the selected net are marked with the character specified in the Draw Modes tab of the **Tools/Options** screen (by default, this is a cross).

   If you try to add a component pin that belongs to another net, you are prompted to confirm that you want to delete this component pin from the other net. Click **Yes** to confirm the override, or **No** to cancel and to return to the design screen.

Return to Presentation

2. Choose **Netlist/Edit**. The Edit net screen appears.



3. Choose the net you want to add to from the **Select Net** list. A list of the components in that net appears.
4. Click **Add Pins**. The design reappears and the free pin in the selected net changes color.
5. Click the pin on the connector that you want to add to this net.
6. To confirm that the pin has been added to this net, choose **Netlist/Edit** again and select it from the **Select Net** list. You will see that your previously unconnected pin has been added to the net.

4.4 Moving Components

4.4.1 Moving Components with the Mouse

➢ To move a component:

1. Choose **Component/Move**, or click the Component Move button on the toolbar. The component list appears.
2. From the list of components, select the component you want to move, or type the component's refdes. You can zoom in on the component you want in order to see the number. Once selected, the component appears in "ghost-line", attached to the mouse.
3. Move the component to the location where you want it placed. Use the x / y coordinates on the status bar to get a precise reading on the location of the cursor.
4. Click to temporarily drop the component in place. If you want to move it from this location, just click on a new location.

Return to Presentation

5. When you are on the exact location of the component, right-click or press ESC to leave it in position. The component list appears again so you can choose another component to move, or you can simply close the list by clicking the “X” in the top right corner of the screen.

Tip To set the view of the board to show only components and make it easier to see what you are moving, enable only the **Components** option on the Group tab of the **Tools/Options** screen.

Tip While the component is attached to the mouse you can rotate it by pressing F2.

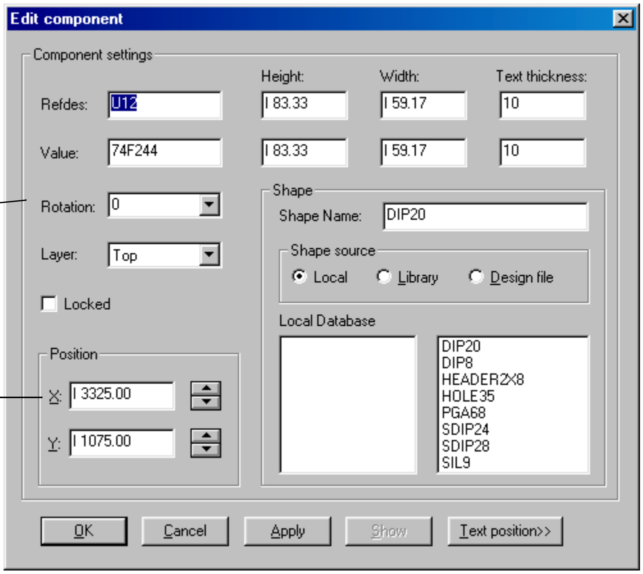
4.4.2 Moving a Component Using Coordinates

➢ To move or place a component by using exact coordinates:

1. Choose the component that you want to move by either double-clicking the component, or by choosing **Component/Edit** and selecting the one you want from the component list. The Edit Component screen appears.

Set the angle of rotation

Set the x/y coordinates



2. In the **Position** field, in the bottom left of the screen, assign the X and Y coordinates for the component’s position.

3. Change the angle of rotation by selecting from the options in the **Rotation** list.

4. To temporarily apply your changes, click **Apply**. To apply your changes and close the screen, click **OK**. To cancel your changes, click **Cancel**.

4.4.3 Swapping a Component to Another Layer

You sometimes need to place components on another layer, particularly for SMD technology and connectors, where you may need to place components on the bottom layer.

➢ To place a component on another layer:

1. Be sure the layer you want to use is one of the active layers. You set the active layers using **Traces/Set Active Layers**.
2. While moving the component, press F5 to toggle between the two active layers.

   OR

1. Double-click on the component or choose **Component/Edit** to open the screen.
2. Choose **Bottom** from the Layer drop-down list.

3. Click **Apply**, and click **OK** to return to the board, or click **Cancel** to close the screen without applying the changes.

4.4.4 Dragging Components

The **Components/Drag** command allows you to drag a selected component while maintaining its connections (if those connections are possible).

As components are dragged, Ultiboard makes use of its autorouting routines to redraw traces. Design rule check is always enabled while you drag, and it monitors for potential short circuits and clearance errors. If the move would cause short circuits or clearance errors, the connections are not made. For more on design rule checks, see “Setting Design Rules” on page 3-13.

Always drag the selected component in small steps. To get the best possible performance from the autorouting routine when dragging components, temporarily place the component by clicking the left mouse button at each grid step you make.

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).

Return to Presentation

➢ To drag a component:

1. Choose **Component/Drag**. The component list appears.
2. From the list of components, select the component you want to drag, or type the component's refdes. You can zoom in on the component you want in order to see the number. Once selected, the component appears in "ghost-line", attached to the mouse.
3. Drag the component to the location where you want it placed. Use the x / y coordinates on the status bar to get a precise reading on the location of the cursor.
4. Click to temporarily drop the component in place. If you want to drag it from this location, just click on a new location.
5. When you are on the exact location of the component, right-click or press ESC to leave it in position. The component list appears again so you can choose another component to drag, or you can simply close the list by clicking the "X" in the top right corner of the screen.

Tip To set the view of the board to show only components and make it easier to see what you are dragging, enable only the **Components** option on the Group tab of the **Tools/ Options** screen.

Tip While the component is attached to the mouse you can rotate it by pressing F2.

4.4.5 Moving Components by Group

➢ To reposition a group of components:

1. Choose **Group/Move**, or click the **Component Move** button on the toolbar.
2. Select the group by clicking on the opposite corners of a rectangular area of the workspace that encloses the components you want to move.

   You can choose **View/Zoom in** or click the Zoom in button on the toolbar to zoom in on the components to see their numbers.

   Once selected, the components appear in ghost-line, attached to the mouse. Ratsnests are also displayed during the move process if you have enabled them in the Group tab of the **Tools/Options** screen.
3. Move the components to the location you want. Use the x / y coordinates on the status bar to get a precision reading on the location of the cursor.
4. Click to drop the component in place. If you want to move it again, click in a new location.
5. When you are on the exact location for the group, right click or press ESC to leave it in position. Press F2 while moving the group to change the orientation.

Return to Presentation

Tip To set the view of the board to show only components and make it easier to see what you are dragging, enable only the **Components** option on the Group tab of the **Tools/ Options** screen.

Note The design rule check does not automatically monitor connections during a group move.

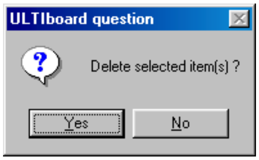
4.5 Deleting Components

➢ To delete a component:

1. Choose **Component/Delete**, or click the **Delete component** button on the toolbar. A component list appears.

2. Select the component you want to delete from the component list.

   A message appears, asking you to confirm the deletion of the selected components.



3. To delete, click **Yes**. To cancel the deletion, click **No**.

➢ To delete groups:

1. Choose **Group/Delete** or the **Delete Group** button on the toolbar.
2. Select the group of components you want to delete. A message appears, asking you to confirm the deletion.
3. To delete the group, click **Yes**. To cancel the deletion, click **No**.

4.6 Locking and Unlocking Components

You can lock components in place to ensure that you don't move one without getting a warning message from Ultiboard. This is a valuable feature for components that require precise placement. No move can take place without the warning to unlock appearing.

By locking components in place, you are protecting them from being moved by mistake.

Return to Presentation

4.6.1 Locking Components

➢ To lock a group of components:

1. Choose **Components/Group lock**.
2. Click in the workspace to start delineating a rectangular area enclosing the components.

   To lock a single component, select an area encompassing only one component. Zooming in on the area might help.
3. When you click the second time to delineate the group, components in the selected area are locked in place. Locked components are grayed out so you can identify them easily.

➢ To lock an individual component:

1. Choose **Components/Edit** or double-click the component to open the Component Edit screen.
2. Ensure the component's refdes, **Reference Designator**, is correct.
3. Enable **Locked**.

4. To complete the locking of the component, click **Apply**, and click **OK** to return to the board.

While components are locked, any attempt to move them results in a message warning you to unlock.

4.6.2 Unlocking Components

➢ To unlock a group of components:

1. Choose **Components/Group/Unlock**.
2. Select the area containing the components forming the group. The group is unlocked.

➢ To unlock a single component:

1. Choose **Components/Group unlock**.

Return to Presentation

2. Select the area containing the single component. Zooming before selecting might be helpful.

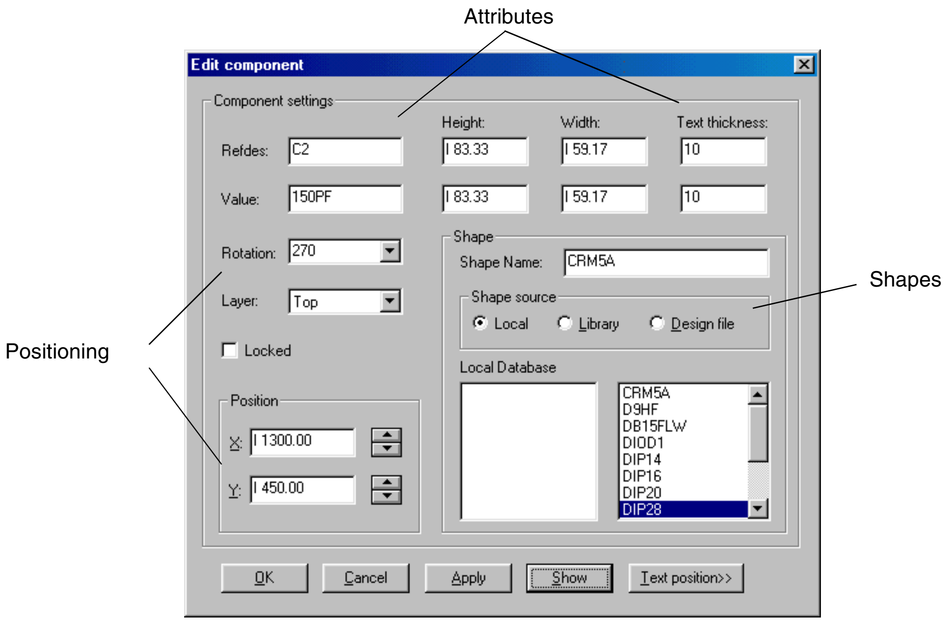
   OR

1. Double-click on the component to select it or choose **Component/Edit**. The Component edit screen appears.
2. Disable the **Lock** option.
3. Click **Apply** and **OK**.

4.7 Editing Components

➢ To edit the attributes of a component:

Choose **Component/Edit**, or double-click on the component. The Edit component screen appears:



Return to Presentation

4.8 Editing Shapes

➢ To edit shapes:

1. Choose **Components/Shape/Edit Shape** to edit shapes in the Ultiboard Shape Editor screen. The Shape name drop-down list appears.
2. Edit an existing shape or create a new shape by:
   - clicking on the component shape using the mouse
   - entering the name of the component in the drop-down list field
   - selecting the shape from the drop-down list.

   When you enter a shape name in the drop-down list field, Ultiboard first searches in the local library for the shape. If the shape name is not found, Ultiboard then searches through the libraries set in the Libraries tab of the **Tools/Options** screen. If the shape name is still not found, Ultiboard creates a new shape with the specified shape name in the local library.

The Shape Editor screen appears, with your selected shape displayed for editing, or with a clean palette for creating your new shape. (For more information on the Shape Editor menus and commands, see "The Shape Editor" on page 9-46.)

After editing the shape you can choose to overwrite all components of that shape with the modifications. If you re-shape only one component, the shape is saved with the suffix "!n" (n=0,1,2,...).

For example:Dip16 Dip16!0

Dip8 Dip8!1

➢ To return to the Ultiboard workspace:

1. Choose **File/Exit**. A message appears asking you if you want to overwrite the shape or not.
2. Click **Yes** to overwrite the shape in the local library, or No to return to the workspace without saving your changes.

4.8.1 Copying Shapes

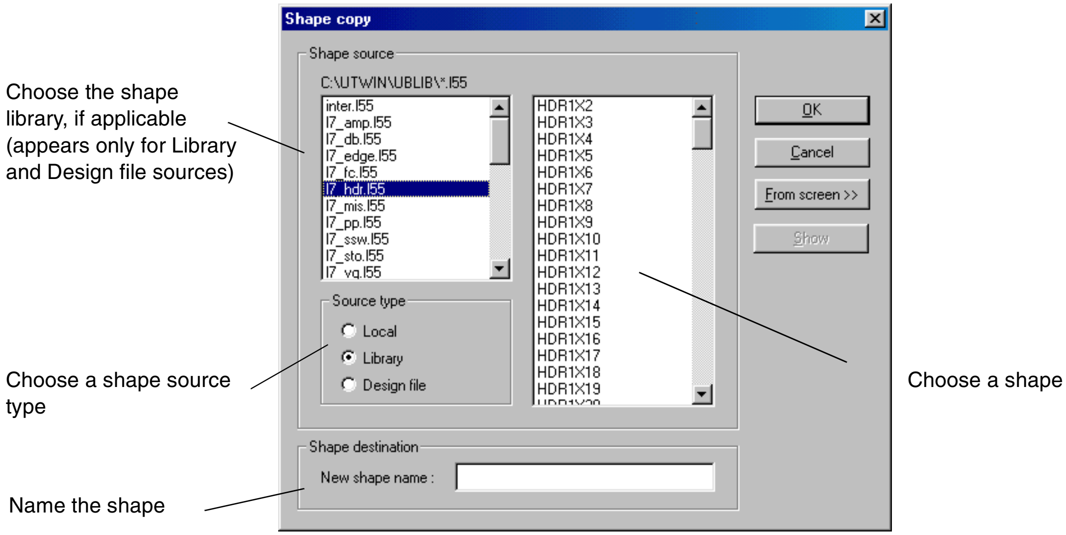
You can copy an existing shape and edit the copied shape in the Shape Editor. Copying shapes is useful when you want to create shapes which differ slightly from one another (for example, a difference in the number of pads, pad code, outline etc).

The shape to be copied, the "source shape", can be from a local library, an external library or an external design.

Return to Presentation

➢ To copy a shape:

1. Choose **Components/Shape/Copy shape**. The Shape Copy screen appears.



2. Choose the **Source type** from among **Local**, **Library** or **Design file**. If you chose **Library** or **Design file**, a list of libraries or design files appears on the left and a list of shapes in the selected library or design file appears on the right. If your source is local, only the list of shapes appears.
3. If you chose **Library** or **Design file**, select a library or design file from the left list. A list of shapes in that library or file appears in the right list.
4. Select an existing shape. To have a closer look at the shape in the Shape preview screen, choose **Show**. To close this screen, click **OK**.
5. Enter the name of the new shape in the **New shape name** box.
6. Click **OK**. The Shape Editor appears, already loaded with the shape you just named.
7. Make modifications to the newly named copy.
8. When finished, choose **File/Exit**. You return to the Ultiboard workspace.

The new shape you create is an exact copy of the source shape but with a different shape name. The copied shape exists only in the local library.

You are prompted to overwrite the existing shape if the selected shape name already exists in the local library.

Return to Presentation

Note You can not copy a shape from an external design file or library file if the name of that source shape already exists in the local library. Ultiboard will prompt you to use the shape from the local library as the source shape instead.

4.8.2 Deleting Unused Shapes

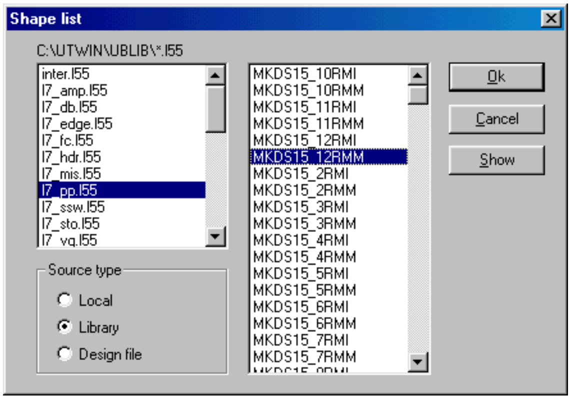
➢ To delete shapes from the local library.

1. Choose **Components/Shape/Delete unused shape**. The Delete shape list appears.
2. Select the shape to be deleted from the list. Only shapes that are not in use in the design can be deleted from the local library. The shape is deleted when you select it.
3. To close the Delete shape list and return to the Ultiboard workspace, click on the “X” in the top right corner of the screen.

4.8.3 Listing Shapes

Use to list and preview all shapes in the local library, an external design file or an external library file.

1. Choose **Components/Shape/List shapes**. The Shape list appears.



2. Enable the **Source type**. If you chose **Library** or **Design file**, a list of possible libraries or design files appears on the left.
3. Double-click on a shape name in the right list, or click **Show** to open the Shape preview screen. To close the Shape preview screen, click **OK**.

Return to Presentation

4. To close the Shape list screen and return to the Ultiboard workspace, click **OK**.

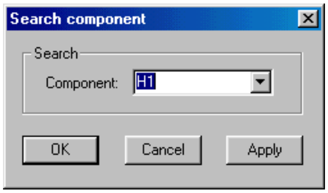
4.9 Finding Components

You can easily search for a component in a design by its refdes.

If the component is found it will be highlighted. If the refdes does not exist, an error message is displayed.

If the component you try to find is not positioned in the current drawing area, the design will be redrawn with the same zoom level, but with the found component in the middle of the screen.

1. Choose **Components/Find**. The Search component screen appears.



2. Choose the component from the Search component list or enter the component's refdes.
3. To highlight the component and keep the Search component screen open, click **Apply**. To highlight the component and close the screen, click **OK**.

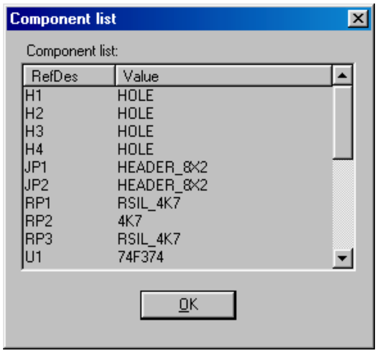
   To close the Search component screen without making a choice, click **Cancel**.

4.10 Listing Components

You can list all the components that are present in the design. Components are listed by refdes.

Return to Presentation

1. Choose **Components/List**. The Component list screen appears.



2. To close the Component list, click **OK**.

4.11 Achieving Optimal Placement

Achieving the best possible placement of your components is critical to the overall performance of your board and to the cost of production. Designers commonly recognize the importance of strategic trace placement, but overlook the importance of component placement. Optimal placement of components makes routing your traces easier and more efficient. Ultiboard's force vectors and histogram features help you achieve optimal placement of your components.

Another way to achieve optimal part placement is to use Electronics Workbench's Ultiroute. Ultiroute performs completely automated placement of all your parts to achieve optimal routing results.

4.12 Force Vectors

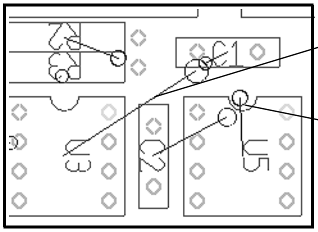
Force vectors are powerful aids that help you place components with optimum intelligence. When components are placed manually on the board, you should pay careful attention to the force vectors coming from that component. They allow you to place the component as close as possible to other components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

Try to minimize the ratsnest distances from that component to other pins on the board. Force vectors work by treating the force ratsnest lines coming from each component as if they were vectors, adding them together as a vector sum, and producing a resultant force vector.

Return to Presentation

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.



➢ To enable force vectors:

1. Choose **Tools/Options**. Click the View Items tab.
2. Enable **Force Vectors** in the Visual section of the screen.
3. To temporarily apply your changes, click **Apply**. To apply the changes and close the screen, click **OK**.

➢ To set the color for force vectors:

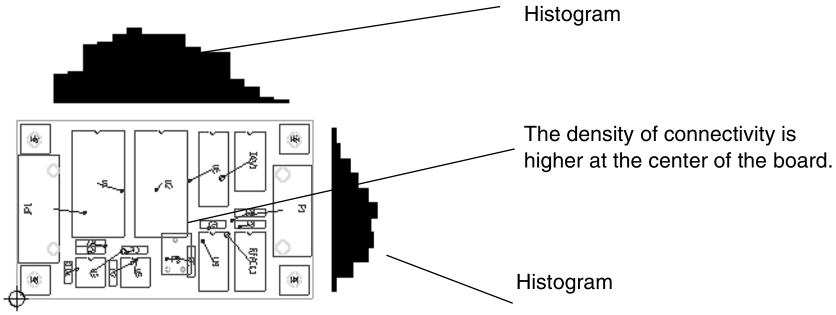
1. Choose **Tools/Options**. Click the Layers & Colors tab.
2. Double-click on the square of color beside **Force Vectors**.
3. Select your color.
4. Click **OK** to return to the Layers & Colors screen.
5. To temporarily apply your changes, click **Apply**. To apply the changes and close the screen, click **OK**.

4.13 Density Histograms

Density histograms indicate the connectivity density at cross-sections of your board. The histogram is higher at cross-sections with high connection density. The higher the connection density at any given cross-section, the more difficulty you will have routing traces through

Return to Presentation

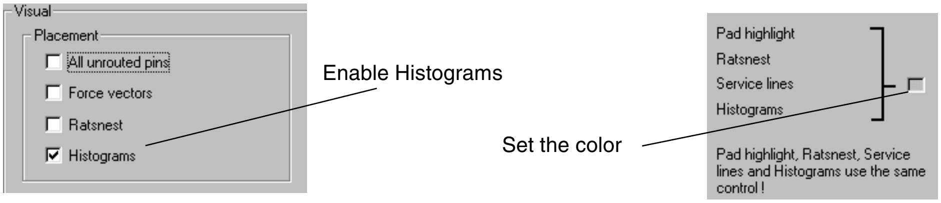
that section of the board. When placing parts you should strive to achieve relatively flat density distributions to avoid difficult-to-route areas.



It will probably not be possible to achieve truly flat density distributions. The center of the board always has a relatively higher density than the perimeter areas. Your strategy should be to try and avoid areas that appear to exhibit higher density distributions than normal.

➢ To see the histogram for your board design:

1. Enable **Histograms** in the View Items tab of the **Tools/Options** screen,
2. Set the color for histograms in the Layer & Colors tab.

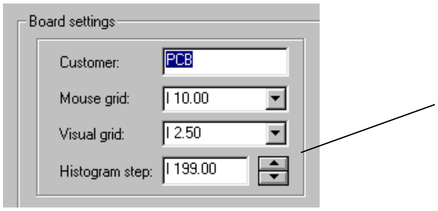


While you are placing a component, check the histogram readings. As you move the point of insertion, the histogram monitors the relative density of the position.

➢ To set the histogram step size:

1. Choose **Tools/Options**. Click the **Board Settings** tab.
2. Use the incremental counter to set the histogram step size.
3. Click **Apply**.

4. Click **OK** to close the screen and return to the workspace.



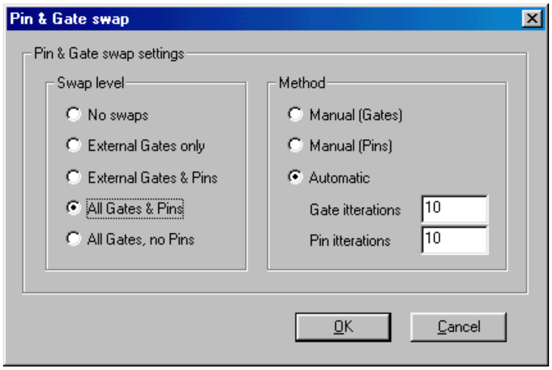
Use the incremental counter to set the histogram step size.

4.14 Swapping Gates and Pins

The pin and gate swap feature depends on the contents of the `pkg.dat` and `$pkg.dat` files. Ultiboard scans these files and collects swap information for all components in the design. This information is stored in the device (`.dvc`) file. The pin and gate swap feature is based on the components' value property. For swapping to work correctly, all components must have the correct value property assigned (they can not be empty).

➢ To carry out a pin and gate swap:

1. Choose **Tools/Pin & Gate Swap**. The Pin & Gate Swap screen appears, as shown below:



Return to Presentation

2. Set your desired pin and gate swap options:

Swap Level	Set a privilege level which allows swaps to be backannotated to your particular schematic. This avoids any non-automatic backannotation, but may produce less desirable designs if your schematic program cannot handle all gates and pins. The possible choices include: • **No Swaps** (no swaps allowed) • **External Gates only** (only external gate swaps allowed) • **External Gates & Pins** (external gate and pin swaps allowed) • **All Gates & Pins** (all gate and pin swaps allowed) • **All Gates, no Pins** (only gate swaps allowed).
Manual (Gates)	Use to swap gates manually. Select the individual gates to swap by clicking on them or directly enter coordinates.
Manual (Pins)	Use to swap pins manually. Select the individual pins to swap by clicking on them or directly enter coordinates.
Automatic	Use to automatically begin to inspect the PCB. When the system finds equivalent gates or equivalent pins which, when swapped, shorten the point-to-point connection distances, it swaps them. You must set the maximum number of iterations. This command operates only within the criteria established by the **Swap Level** option.

3. To begin the swap, click **OK**.

The device file is only created once. When you change the swap information in the `pkg.dat` or `$pkg.dat` file, you should delete the device file manually, thereby forcing Ultiboard to create a new device file containing the changed swap information.

Always remember that you are changing your schematic with every swap you make. Backannotation is used to update your schematic to ensure the integrity of the data in both the circuit board and the schematic. For more information on the backannotation procedure, see “Backannotation to Electronics Workbench Schematic Capture” on page 2-3.

Chapter 5
Placing Traces

Traces

Return to Presentation

Traces

Return to Presentation

Chapter 5
Placing Traces

5.1 About this Chapter

Placing traces is possibly the most critical part of any PCB design. This chapter explains how to place traces manually. Automatic placement of traces is explained in the next chapter. Even if you prefer to route your traces automatically, it is important to be able to route some traces manually. It is customary to pre-route critical traces before autorouting. Such critical traces may include power and grounds.

5.2 Changing Trace Widths

Ultiboard comes with 32 different trace codes. A trace code identifies a trace type with its own unique width and clearance. Assigned trace widths are passed on to autorouters.

5.2.1 Changing Trace Widths in the Netlist

You can manually edit the trace width information at the netlist level when importing a `.net` file. For an explanation of this type of file, and its uses, see “Understanding .net and .plc files” on page 3-18.

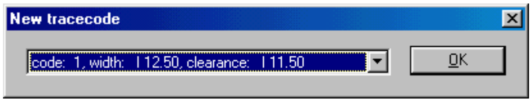
5.2.2 Changing Widths While Laying Traces

You can change trace widths as you are laying a trace. This valuable feature allows you to place wider traces where possible but, when necessary, switch to a narrower trace width. For example, you can change the width when routing through pins.

Return to Presentation

➢ To switch trace widths while routing a trace:

1. Press F5.
2. Choose a new trace code from the New Trace Code list that appears.



3. Click **OK** to finish and return to the workspace.

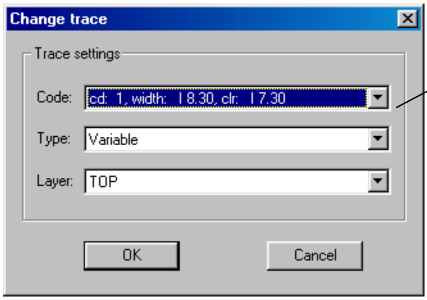
OR

1. While placing a trace, choose a new width from the **Width** box in the Trace Toolbox.

5.2.3 Changing Trace Widths after Traces Have Been Placed

➢ To change trace widths after a trace has been placed:

1. Double-click on the trace. The Change trace screen appears.
2. Select a new trace code from the list in the drop-down box.
3. Click **OK** to close the screen.



Select a new trace code from the drop-down list.

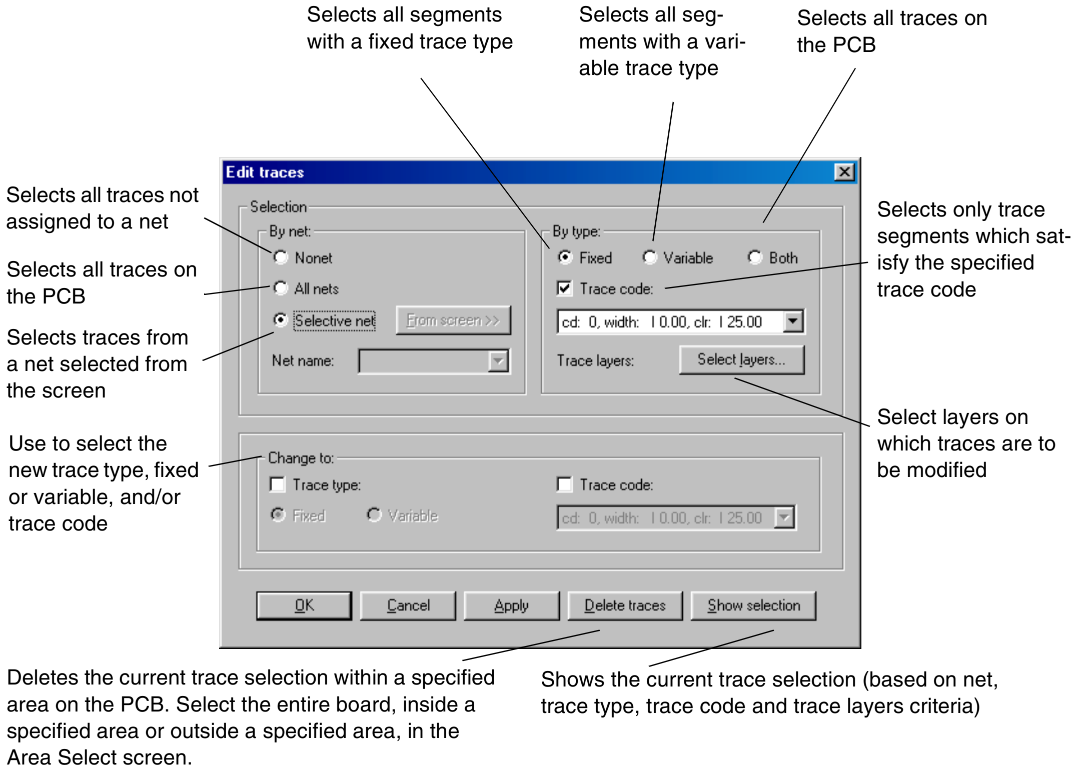
5.3 Editing Traces

Use to delete or modify a selection of traces.

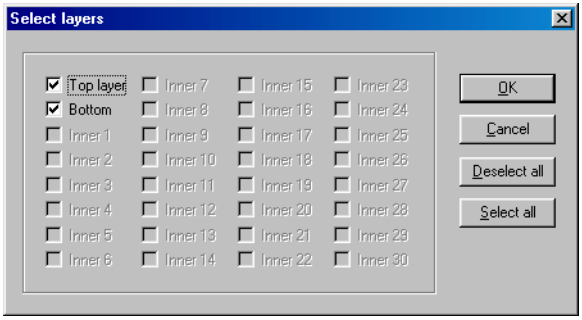
➢ To edit a trace:

Return to Presentation

1. Choose **Traces/Edit.** The Edit Traces screen appears.



2. Enter the new trace code or trace type for the selected traces. The bottom portion indicates the new net values that you want to assign. You can change the trace widths for each layer.
3. Optionally, click **Select Layers**. The Select Layers screen appears.



Traces

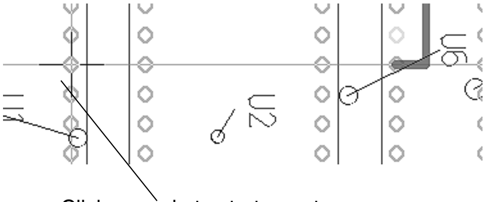
Return to Presentation

4. Enable those layers whose traces you want modified. To select all layers, click **Select all**. To deselect all layers, click **Deselect all**. To cancel the changes, click **Cancel**. To save the changes and return to the previous screen, click **OK**.
5. To temporarily apply your changes, click **Apply**. To apply your changes and close the screen, click **OK**. To cancel your changes, click **Cancel**.

5.4 Drawing Traces

➢ To draw a trace:

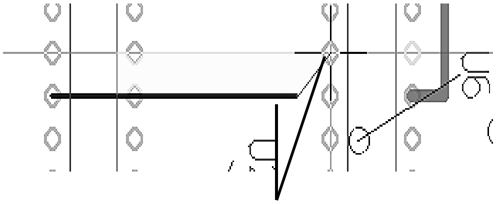
1. Choose **Traces/Place,** or click the **Place trace** button on the toolbar to start drawing a new trace.
2. Click on the workspace where you want to start the new trace.



Click on a pin to start your trace

Always start a new trace on a component pin that is part of the net you want to route manually. Component pins that belong to that net are highlighted.

3. Move the mouse to the next point of the trace. The trace is placed on the current active layer.

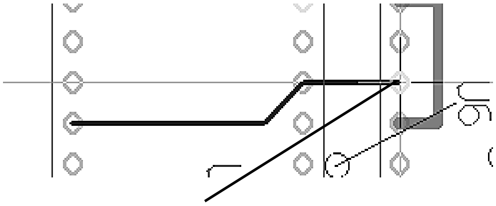


Click to set each segment of your trace

4. Click to anchor a segment of the trace, then continue to move the cursor.

Return to Presentation

5. Right-click to finish the trace.



Right-click to finish your trace

The new trace is shown in the default color for its layer, as set in the Layers & Colors tab of the **Tools/Options** screen.

Some points to remember about placing traces:

- The status bar shows which net you are routing as well as the current trace code and trace type. Ultiboard automatically assumes a trace code of 1 for unspecified nets.
- To change the trace code during routing, press F5.
- To continue routing on the other active layer, press F2 or click.
- To change the current via code, press SHIFT + F5.
- To toggle between orthogonal routing, 45º routing, and all angle routing, press the space bar.
- To go directly to **Place Arc**, press SHIFT+F3.
- Ultiboard automatically inserts a via at the end of the last trace segment you place.

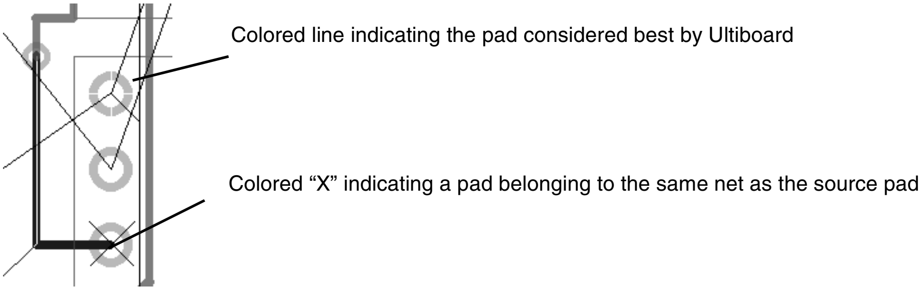
5.5 Making Connections

➢ To make a connection between pins or pads:

1. Choose **Traces/Place** or click the Place Trace button.

Return to Presentation

2. Click on the pad that you want to start routing from. When you click, Ultiboard highlights, with colored “Xs”, all the pins that are part of the same net as the pad you started routing from.



You can easily locate all the pads to which the source pad could be connected. Ultiboard also recommends which pad you should connect to next by placing a colored line directly between the source pad and the pad considered “best” by Ultiboard.

3. Click to anchor a segment of the trace.
4. Right-click to finish placing the trace.

5.5.1 Changing Drawing Angle

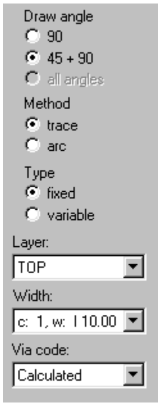
You can route using three different routing settings:

- 90 degrees (routes horizontally and vertically)
- 45 and 90 degrees (routes horizontally, vertically, and at right angles)
- all angles (routes at any angle)

You can change the angle mode while you are placing a trace.

Return to Presentation

➢ To change the angles, choose from among the Angles listed in the Traces Toolbox.



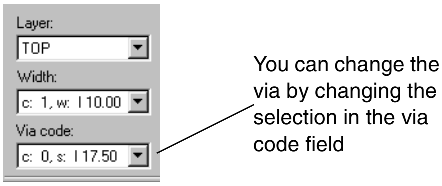
If the **All angles** option is grayed out in the Trace Toolbox, go to the Traces and vias tab of the **Tools/Options** screen, and enable **Allow all angles**.

5.5.2 Changing Layers and Inserting Vias

➢ To change between layers while you are placing a trace:

1. Choose a layer from the **Layer** drop-down list in the Traces Toolbox.

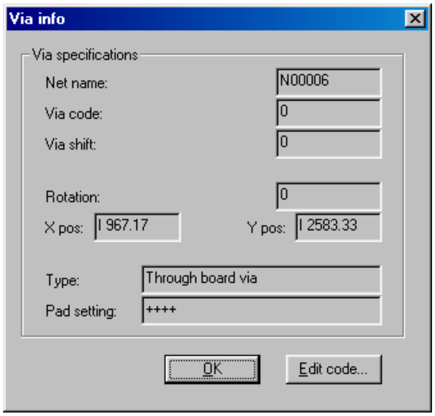
   Ultiboard automatically inserts a via at the location of the last vertex that was placed in the design. The code displayed in the via code field determines what type of via is inserted.



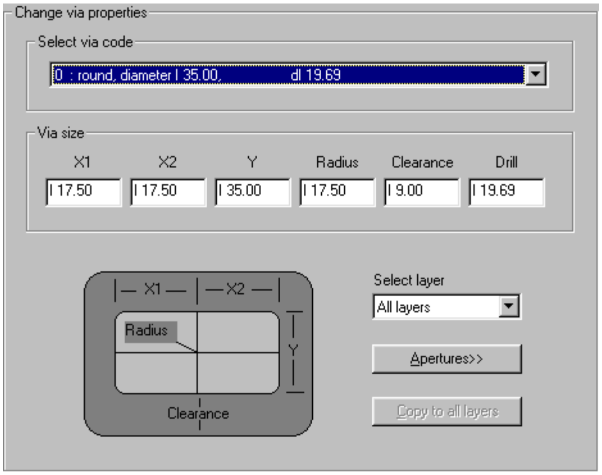
➢ To change the via's properties:

Return to Presentation

1. Double-click on the via to open the Via info screen.



2. Click **Edit code** to open the Change via properties screen.



3. Change the via code, or other properties, and then click **OK** on the Options screen to close it and return to the workspace.

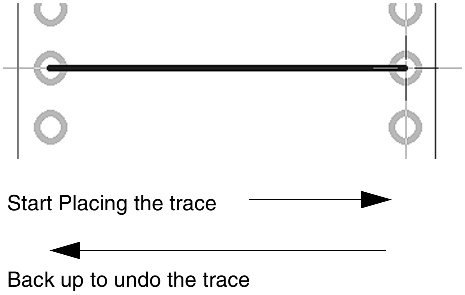
Traces

5.5.3 ErasingTraces

Often when routing, you decide that you want to take a completely different path from the one you started. Rather than having to exit **Traces/Place**, you can undo the trace you have drawn by backing over it.

Return to Presentation

➢ To undo a trace, back up over the same path that you took to draw the trace.



5.6 Moving, Dragging, and Deleting Traces

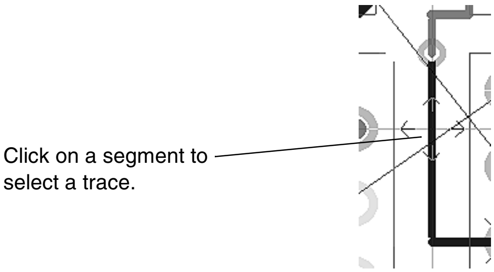
5.6.1 Moving Traces

As you adjust the routing in a design, the “reroute while move” feature automatically shoves traces and maintains spaces (clearance rules). This feature depends on having a legal space into which to move the traces and vias. You receive an error message if this space is not available.

You can change the active layers in a multi-layer design by pressing the + and - keys on the numeric keypad.

➢ To move a trace:

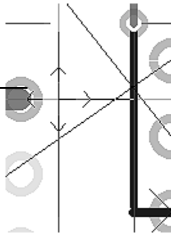
1. Choose **Traces/Move** or the **Move Trace** button, on the toolbar.
2. Click on a trace segment or a component pin to select it. When you move the cursor, a rubber band line appears between the cursor and the point of selection.



Return to Presentation

3. Move the mouse to the point where you want to reposition the trace.

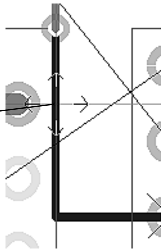
The cross-hairs indicate the position where the trace will be moved to.



The incremental distance for a move is set in the Mouse Grid box in the Board Settings tab of the Tools/Options screen

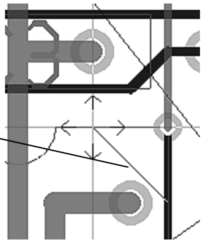
4. Click the mouse again, and the trace is repositioned.

Each time you click the mouse, the Move command repositions the associated segments and vias.



5. To stop moving the selected trace, right-click or press the ESC key. You can then select another trace to move, or stop moving traces by clicking the right mouse button again or by pressing ESC.

The rubber band that attaches the segment to the mouse pointer remains active as long as you are in Move Trace mode.



Note While moving a trace, the real-time design rule check is always enabled, independent of the settings for the design rule level.

Vias are moved in the same manner as traces. Click on the via and move it to a new location. Associated traces are moved with the via.

Return to Presentation

5.6.2 Dragging Traces

Traces are dragged at the same time as components, moving with components as they move. The design check feature is on while a move is made, so any connections that may be broken by a move are flagged by the program. For more on components and their attached traces, see “Moving Components” on page 4-7.

5.6.3 Deleting Traces

➢ To delete a trace:

1. Choose **Traces/Delete/Delete Segment** to delete a single segment.
2. Click on the trace segment to select it. When you move the cursor, a rubber band line appears between the cursor and the point of selection.
3. Click one more time to delete the trace segment.

   Press the ESC key to cancel the deletion. Press the ESC key again to stop deleting traces. Delete trace only works on traces in the current active layer and only on one trace segment at a time.

Additional commands are available in the **Delete Traces** sub-menu:

- **Delete Open trace ends** deletes all open trace ends in the design to clean up the design when completed.
- **Delete Single Net** deletes all traces of a single net. Select the net from the list box or click a trace-segment or component pin that belongs to the specific net.
- **Delete All traces** deletes all traces in a specified area on the PCB. You are prompted to choose to delete all traces on the entire board, all traces inside a specified area or all traces outside a specified area. If you choose to delete inside or outside an area, select the area by clicking at the two opposite corners of the rectangle that includes the traces.

Note **Delete Trace** will not delete associated vias.You must use **Delete Single Via** or **Delete Unused Vias** to delete associated vias. Use **Delete unused vias** to delete all vias that do not have any trace-segments or copper areas connected to them.Vias that have one trace segment or one copper area connected to them will not be deleted by this action. Use the **Delete single via** command to delete these vias.

Return to Presentation

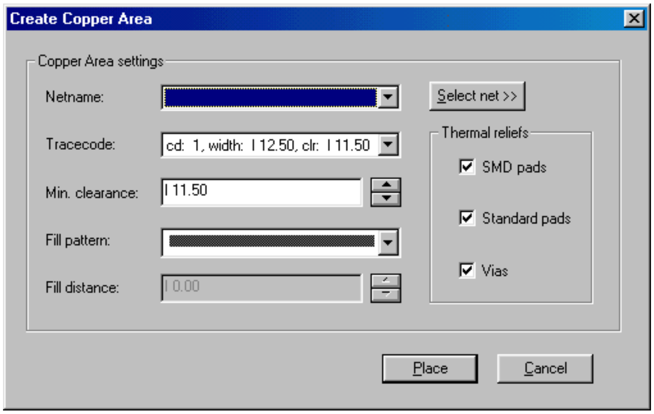
5.7 Using Copper Areas

5.7.1 Creating Copper Areas

Use the **Copper area/Place** command to create copper areas in a specified net.

➢ To create a new copper area:

1. Choose **Traces/Copper Area/Place**.
2. Set the angle value in the Trace Toolbox to **All angles**. This gives you almost limitless options for the shape of your copper area.
3. Choose your copper area settings in the Create Copper Area screen.



Traces

Fill in the following fields:

Netname	Attaches the copper area to a net. Select your net from the list of netnames using the drop down list or use the Select button to select the net directly from your design workspace.
Tracecode	Affects how fill patterns are drawn.
Min. Clearance	The minimum allowable clearance for this object. Used by the design rule check.
Fill pattern	The distinctive pattern that you can assign to a particular copper area, related to how much copper is used in the production of the board.
Fill distance	The distance between lines in a fill pattern.
Thermal reliefs	Establishes whether or not you want thermal reliefs for the SMD pads, standard pads, and vias.

4. Click **Place** to close the screen and to draw the outline of the copper area in the workspace.
5. Click the starting position of the copper area outline.
6. Click on the desired outline vertices.
7. When you are finished placing the vertices, right-click or press the ESC key to close and place the copper area. A message appears asking if you want to place the copper area.
8. Click **Yes** to place the copper area, or **No** to cancel. When you click **Yes**, the program places the area. The Create Copper area screen opens again.
9. Create another copper area, or click **Cancel** to close the screen and return to the workspace.

5.7.2 Deleting a Copper Area

➢ To delete a copper area:

1. Choose **Traces/Copper Area/Delete.**
2. Click to select the area you want to delete. A message asking if you want to delete the copper area appears.
3. Click **Yes/No** to confirm the delete, or to cancel it.

   Cancel the **Copper Area/Delete** command by right-clicking or pressing the ESC key.

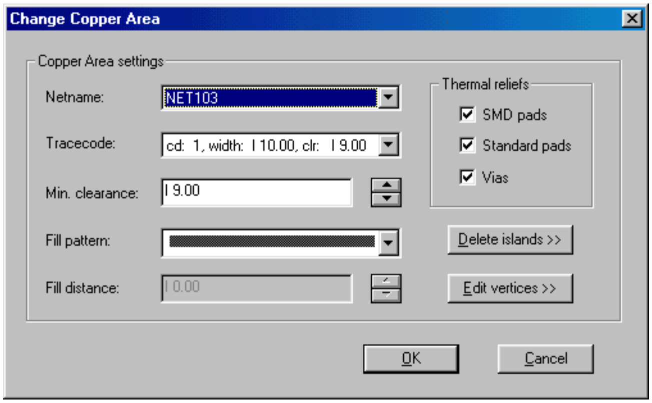
Note The **Copper area/Delete** command works only on copper areas that are in the current active layer. Press F2 to select the other active layer.

Return to Presentation

5.7.3 Editing a Copper Area

➢ To edit a copper area:

1. Double-click on the copper area to select it or choose **Traces/Copper area/Edit**. The Change Copper Area screen appears.



2. Change any of the options, as required.
3. If necessary, click **Edit vertices** to change the copper area outline. The Vertices edit screen appears.

4. Choose **Move**, **Add** or **Delete**.
5. Click a vertex to move, add or delete. Use the cursor to position or add a new vertex.

   If you delete, a message appears asking you to confirm the delete. Click **Yes** or **No**.
6. Press ESC, or right-click to finish and return to the Edit Vertices screen.
7. To delete islands, click **Delete islands**. You return to the workspace, where you can select the islands to be deleted.

Return to Presentation

8. Once done, click **Close** to return to the workspace.

Note Only copper areas in the current active layer can be selected. Press F2 to select the other active layer. You can toggle through the active layers in a multi-layer design by using the + and - keys on the numeric keypad.

5.7.4 Updating a Copper Area

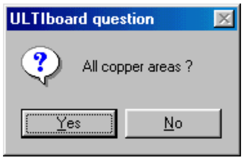
5.7.4.1 Update All

Update all copper areas after adding traces, vias or pads within copper areas. Ultiboard recalculates the necessary voids to pads, vias and traces in different nets and thermal reliefs to pads and vias in the same net for each copper area in the design.

Copper area settings and the copper area outline will remain the same.

➢ To update all copper areas:

1. Choose **Traces/Copper Area/Update All**. A message appears, asking if you want to update all copper areas.



2. Click **Yes** to update all copper areas. The program updates the areas automatically.

   Click **No** if you want to update only one area. Click on the area you want to update. The program updates only the selected area.

3. When the message “Click copper area to change” appears in the status bar, either click another area to update it, or press ESC to finish.

5.7.4.2 Update Single

Update a single copper area after adding traces, vias or pads within a copper area.

➢ To update a single copper area:

1. Choose **Traces/Copper Area/Update Single**. A message appears in the status bar asking you to click on the area to update.
2. Click on the copper area you want to update. The program updates the copper area.

Return to Presentation

Ultiboard recalculates the necessary voids to pads, vias and traces in different nets and thermal reliefs to pads and vias in the same net for the selected copper area. Copper area settings and the copper area outline will remain the same.

Note Only copper areas in the current active layer can be selected. Press F2 to select the other active layer. You can toggle through the active layers in a multi-layer design by using the + and - keys on the numeric keypad.

5.8 Creating and Editing Nets

5.8.1 Creating Nets

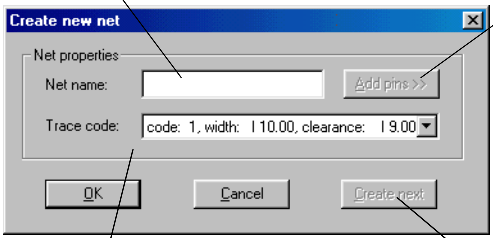
You can create a new net manually.

➢ To create a net:

1. Choose **Netlist/Create**. The Create new net screen appears.

Enter a string representing the new net name. Only use alphanumeric characters (0..9 and A..Z). Must be unique.

Click on the component pins you want to add to the selected net. The component pins that belong to the selected net are marked with the character set specified on the Draw Modes tab of the Tools/Options screen. If you try to add a component pin that belongs to another net, you have to confirm that you want to delete this component pin from the other netlist.



Select or change the trace code for the currently selected netlist. The netlist's default trace code is used by the autorouters to route the net. When routing a net manually the traces use the netlist's trace code by default.

Creates a new net with a net name based on the previously created net's net name. The previously created net is stored. Empty nets are also stored.

2. In the Create new net screen, specify a net name for the net to be created. If the net name you specified already exists, an error message appears. After you specify the new net name, you can add component pins to this new net.

Return to Presentation

3. Click **Add pins** if you want to add new pins. The screen closes and you return to the work-space to select the pins to add. Right-click to reopen the Create new net screen.
4. Click to add pins to the net.
5. Click **OK** to finish and return to the workspace, or click **Cancel** to close the screen.

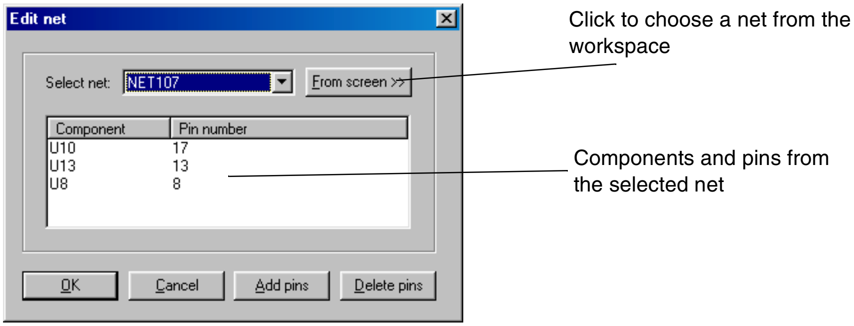
Note If any changes are made in a particular net, you will not be able to backannotate these changes automatically to the schematic.

5.8.2 Editing Nets

You can add or delete component pins from an existing net.

➢ To edit a net:

1. Choose **Netlist/Edit**. The Edit net screen appears.



2. Choose the net you want to edit from the drop-down list or click **From screen** to return to the workspace so you can choose a net by clicking on it. A list displays the component pins that belong to the selected net.
3. Add or delete component pins from the net by clicking **Add pins** or **Delete pins**. The screen closes so you can select pins for deletion or position new pins in the workspace.
4. Right-click to reopen the Edit net screen.

Note If any changes are made to a particular net, you cannot backannotate these changes automatically to the schematic.

5. Click **OK** to finish and return to the workspace, or click **Cancel** to close the screen.

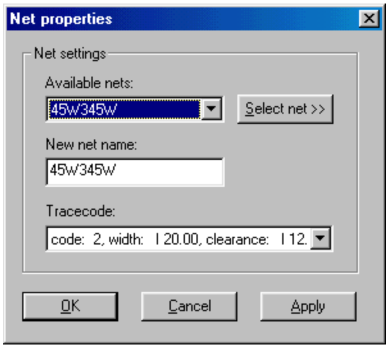
Return to Presentation

5.8.3 Net Properties

Use to change the name and/or the trace code of a selected net.

➢ To change the properties of a net:

1. Choose **Netlist/Properties**. The Net properties screen appears.



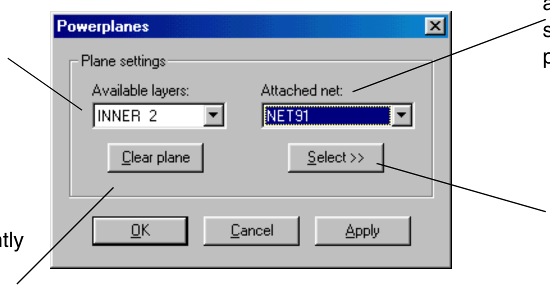
2. Select the net you want to rename. You can choose from the **Available nets** drop-down list, or click **Select net** to return to the workspace so you can click on a net to choose it.
3. Enter the new net name in the **New net name** field.
4. Select the trace code you want to associate to the newly named net from the **Tracecode** drop-down list.
5. To temporarily apply your changes, click **Apply**. To apply your changes and close the screen, click **OK**. To cancel your changes, click **Cancel**.

5.9 Working with Powerplanes

Ultiboard handles the creation of power and ground planes automatically. You must assign the layers you intend to use for power and ground planes.

➢ To create power and ground planes:

1. Choose **Netlist/Powerplanes**.The Powerplanes screen appears.



2. Select a layer from the **Available layers** drop-down list in the Powerplanes screen. The nets in that layer are listed in the **Attached net** drop-down list.
3. Select the net you wish to attach to the selected layer from the list, or click **Select** to choose a net from the workspace by clicking on it. The name of the net chosen from the workspace appears in the **Attached net** field.
4. Click **Apply** to apply the changes. Click **OK** to accept the changes and return to the workspace.

   This procedure assures connectivity of the pads on that layer because a thermal relief is automatically created.

Note You can only assign one powerplane to one layer. Use a copper area to assign more than one powerplane to a specified layer.

Powerplanes are only postprocessing settings for a layer, and are not visible in the design process. Your display will update and redraw faster using this function than if you use the Copper area function.

You need to generate two plot files in the postprocessing phase if you need to have traces buried in your powerplane (penetration of power layers). Generate one normal and one oversized negative (plane) layer for each powerplane.

Return to Presentation

Return to Presentation

Chapter 6
Autorouting

Return to Presentation

Return to Presentation

Chapter 6
Autorouting

6.1 About this Chapter

This chapter describes how to automatically route traces by using the single pass autorouter and the rip-up and retry autorouter* included with Ultiboard. In addition, you may want to consider the Ultiroute product also available from Electronics Workbench, which offers state-of-the-art autorouting for highly optimized routing of boards of all complexity.

*not available in the Personal Edition of Ultiboard.

6.2 The Single Pass Autorouter

6.2.1 About the Single Pass Autorouter

The single pass autorouter is intended for quickly autorouting fairly basic boards. The single pass algorithm places traces sequentially, without the ability to rip-up and replace traces. As a result, the autorouter will work adequately for boards with low density. If the single pass autorouter encounters a trace for which there is no available path to make its connection, it will stop and not be able to achieve 100% completion. If you encounter boards where the single pass autorouter cannot reach 100% completion, you will want to use the rip-up and retry autorouter (included in the Professional and Power Professional editions of Ultiboard—see “The Rip-up and Retry Autorouter” on page 6-10) or Ultiroute (available as a separate product from Electronics Workbench), both of which use rip-up and retry algorithms to clear space for traces that cannot be connected.

Return to Presentation

6.2.2 Pre-routing Critical Traces

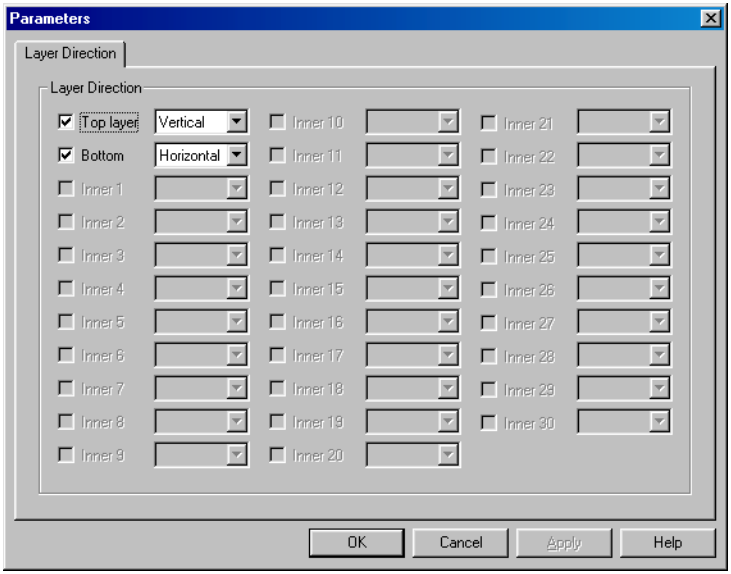
Any traces placed manually are not changed or removed during autorouting. Since most boards have thicker traces for the Power & Ground nets, and ICs are often decoupled with capacitors, we suggest routing those nets manually. Since critical traces often have wide trace widths because they may carry heavy currents, it is especially important to place these components to leave as much open area on the board as possible for the autorouter to route the remaining traces. Copper areas may also be placed before you use the autorouter.

Note If you pre-place copper areas before you route the board, you should update copper areas after the board is routed.

6.2.3 Setting Routing Layers and Directions

➢ To set the routing layers and directions:

1. Choose **Autouroute/Settings**. The Layer Direction screen appears. On the screen, choose from the available options.



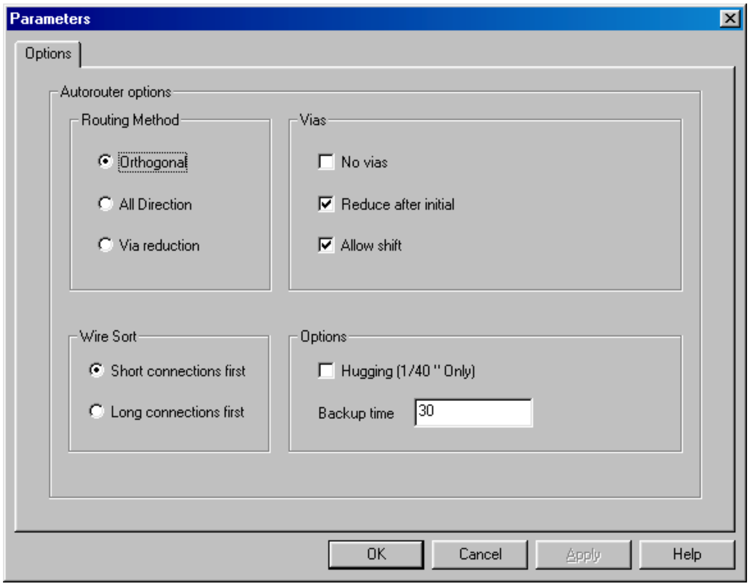
2. To set the layers on which you want the autorouter to operate, select the checkbox of the appropriate layer. If you do not want to route on a particular layer, deselect the layer in the checkbox.

Return to Presentation

3. To set the routing direction per layer, select the direction in the drop-down list for each layer you have selected. This will enable you to use different routing directions on alternate layers, thereby reducing signal interference. The routing direction can be set to Horizontal for horizontal routes, Vertical for vertical routes, North East for North East routes, and South East for South East routes. These are the directions as you view the design on the screen. The autorouter will attempt to maintain the specified directions when routing the different layers. Removing the check mark displayed in front of a layer will disable this layer for routing.
4. To temporarily apply your changes, click **Apply**. To apply your changes and close the screen, click **OK**. To cancel your changes, click **Cancel**.

6.2.4 Single Pass Routing Options

➢ To set single pass routing options, choose **Autouroute/Single Pass/Options.** The Options screen appears. On this screen, choose from the available options.



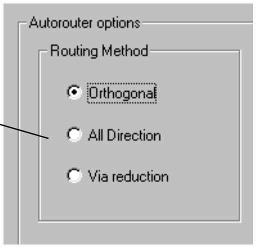
Return to Presentation

6.2.4.1 Routing Methods Options

You can use one of three routing methods when routing your boards:

- orthogonal
- all direction
- via reduction

Select the appropriate routing method



Orthogonal

The orthogonal method uses the preferred layer direction from the Layer Direction screen. This mode is normally used to route multi-layer boards that use more than one layer. In this mode, if the autorouter encounters a situation where it cannot adhere to the preferred direction for a given layer, it will insert a via and try routing on a different layer (to adhere to the preferred direction). As a result, while this method will help reduce signal interference between layers (by routing in different directions on alternate layers), it does so by increasing vias (which can add to your production costs).

All direction

Use this method if you want to ignore the routing directions set in the Layer Direction screen. The **All Directions** option is used to achieve the highest completion rates of all three routing methods since it does not have to adhere to any routing directions and is free to place vias when necessary. You may want to use this option for dense boards and for single-sided boards where the autorouter can route only on one layer.

Via reduction

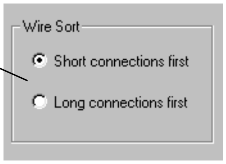
The via reduction option removes nets containing vias and tries to replace them without a via, cleaning up the design.

Note The autorouter will not affect any pre-placed (manually placed) traces. This means that vias on pre-placed nets will be unaffected by the autorouter.

6.2.4.2 Wire Sort Options

Wire Sort determines the order in which traces are routed: short traces first or long traces first. The length is based on the anticipated length of each net's traces, although the literal value is derived from the ratsnest distance between net pins. Usually the best results are obtained if you route the short connections before the long ones, and this setting is used by default.

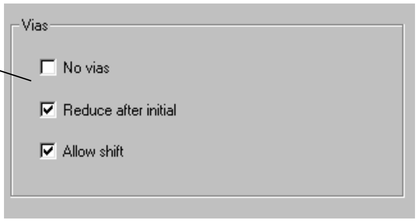
Select connections to be routed first



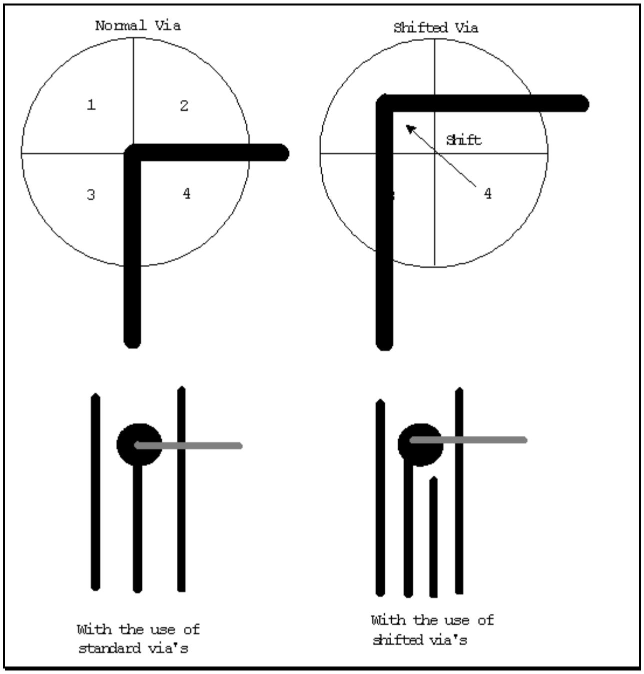
6.2.4.3 Controlling Vias

Via shift determines if vias should be shifted to a selected quadrant to offer optimum free space around vias for denser boards. Using the via shift option will create more space for routing dense areas of your boards. This function is only operational on a 25 mil grid.

Check appropriate selection



Via shift can free space near vias, as shown:

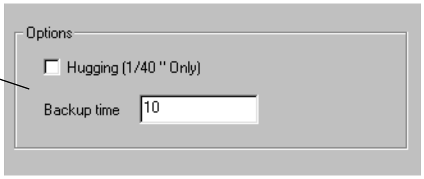


The **No vias** setting completely disables vias. It is also used for single sided boards.

The **Reduce Vias After Initial** setting activates a via reduction pass as the last action after a routing action is performed.

6.2.4.4 Other Options

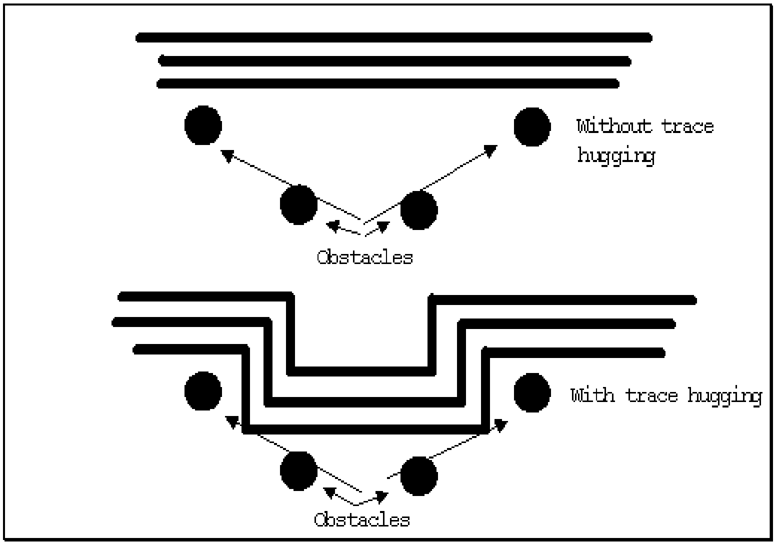
Select trace hugging and backup time, if desired



Return to Presentation

Trace Hugging

Trace hugging moves all routed traces to the nearest obstacle to clear the way as much as possible for routing other traces. Use trace hugging to place traces as close as possible to obstacles, thereby creating as much open area as possible on the board for routing. This is only possible on a 25 mil grid.



Backup Time

You can set the time interval between automatic backups of your design during autorouting by entering the desired time in the **Backup time** field.

6.2.5 Running the Single Pass Autorouter

The single pass autorouter can be run in several different ways:

- Route All
- Route Net
- Route Window
- Route Component
- Route Bus.

6.2.5.1 Route all

The **Route All** command routes all unconnected pins on the board.

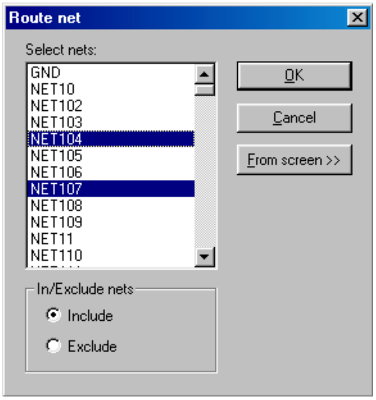
➢ To route all unconnected pins, choose **Autouroute/Single Pass/Route All**.

6.2.5.2 Route net

The **Route Net** command allows you to select one or more nets which you would like routed or excluded from routing.

➢ To select nets to be routed or excluded from routing:

1. Choose **Autouroute/Single Pass/Route Net**. The Route Net screen appears.



2. Select the nets you want to be routed. Enable the **Include** option. Only the highlighted nets are routed.

OR

2. Select the nets you do NOT want to be routed. Enable the **Exclude** option. All nets EXCEPT those that are highlighted are routed.

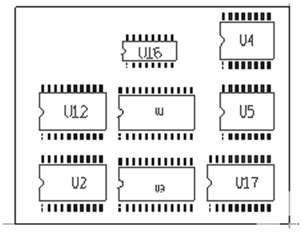
6.2.5.3 Route Window

The Route Window allows you to route all the pins within a user-defined window, which you define with the mouse.

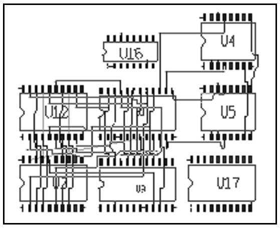
Note Nets connected to pins within the selection window that have connections outside the window are also routed.

➢ To route pins within a user-defined window:

1. Choose **Tools/Autouroute/Single Pass/Route Window**.
2. Place the mouse at the starting point of your window, and click to start drawing your window.
3. Drag the mouse to the end point of your window and click. Pins within that window are automatically routed.



A user-defined window before routing.



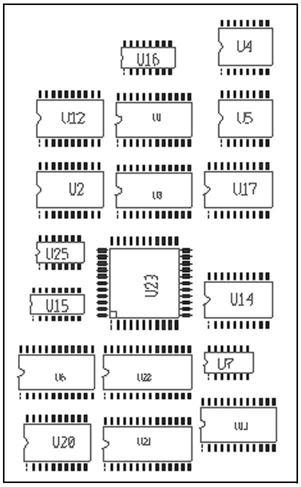
After window is drawn, pins are automatically routed.

6.2.5.4 Route bus

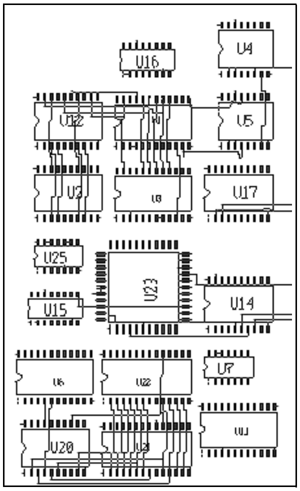
The Route Bus command searches through the design for “buses”. A bus is a number of connections that occur on the board (horizontally or vertically) with a maximum distance between them of two grid points.

Return to Presentation

➢ To route buses, choose **Tools/Autouroute/Single Pass/Route Bus**. Buses are automatically routed.



A board before busses are routed



Board after busses are automatically routed

6.3 The Rip-up and Retry Autorouter

6.3.1 About the Rip-up and Retry Autorouter

The rip-up and retry autorouter is a grid-based router using a powerful rip-up and retry algorithm to route traces. The program runs as a separate application from Ultiboard, so design files are automatically translated to and from the autorouter program.

Note It is recommended that you use a 50 mil mouse grid. The rip-up and retry autorouter is a grid-based router, which means that finer grids increase the amount of memory required to route the board, and decrease the router speed.

6.3.2 Pre-routing Traces

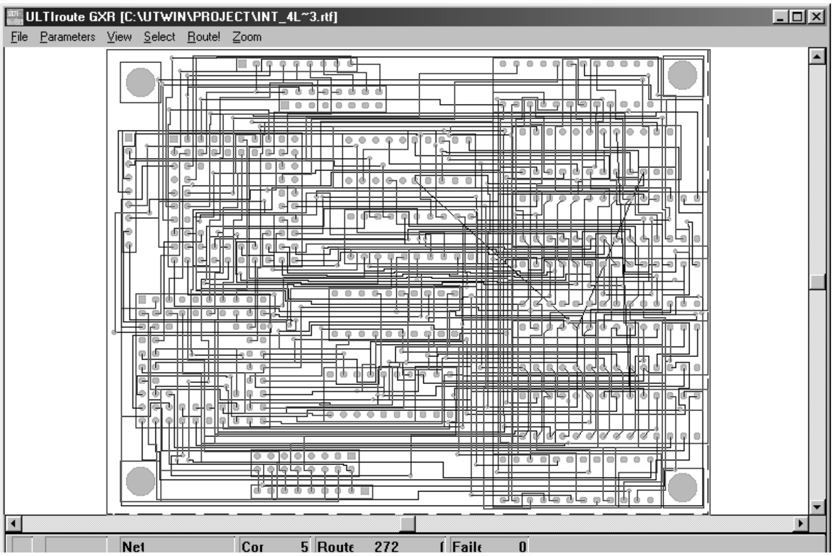
If you use thicker traces for power and ground, we suggest that you manually pre-place these traces before using the rip-up and retry autorouter. You might even pre-route the buses with the Route Bus feature of the single pass autorouter, which is a fast and accurate way of routing buses. If you are going to use copper areas in your designs, we suggest that you place these after you have routed the board using the rip-up and retry autorouter. Drawing the copper areas first will only introduce extra overhead (and these areas can usually be created just as easily after autorouting is completed).

6.3.3 Running the Autorouter

➢ To run the autorouter:

1. Choose **Autouroute/Ultiroute Rip-up and Retry Autorouter/To Ultiroute Rip-up and Retry Autorouter**. The Production Class screen appears and prompts you to choose the production class. (For information on Production Classes, see “About Production Classes (Technology Files)” on page 3-1.)
2. Choose the appropriate production class and click **OK**. The Component Placement Check screen appears and prompts you to verify component placement.

3. Click on the appropriate placement and click **OK**. The layout is automatically translated to a format which can be read into the autorouter. The file is loaded into the autorouter, and is opened in a new window:



6.3.4 Autorouter Options

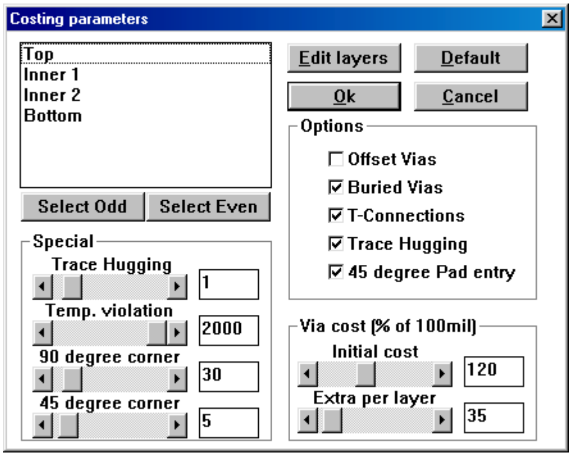
Several options may be set for the router:

- costs for horizontal, vertical and diagonal routing
- trace hugging On/Off and hugging “pull” cost
- temporary violation cost
- degree corner cost
- degree corner cost
- initial via cost
- additional via cost for crossing extra layers
- offset vias On/Off (for 25 mil grid only)
- buried vias On/Off
- allow T-connections
- allow 45 degree pad entries.

6.3.4.1 Costing Parameters

The grid-based router is cost-driven. In evaluating which of a number of possible trace placements to make, the router tries the option having the lowest cost. The costs associated with various options are defined in cost tables.

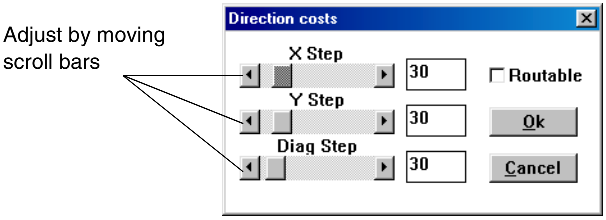
➢ To change costing parameters, choose **Parameters/Costing Parameters.** The Costing Parameters screen appears.



Costing for horizontal, vertical and diagonal routing

Each layer is assigned costs for a move in horizontal, vertical and diagonal directions.

➢ To edit the directional costs, select the layers from the list, and then choose **Edit Layers**. The Direction Costs screen appears.



When a layer will be used predominantly for horizontal traces, the X-step cost is lower than the Y-step cost. The diagonal step cost is normally set to the sum of the X-step and Y-step cost. If you prefer no diagonal trace bends, you may set the diagonal step cost higher. Disabling the **Routable** option off in the Direction cost screen disables the layer for routing. This is done automatically for power plane layers.

Trace Hugging

Trace hugging is a mechanism that packs traces close to each other so that free areas are not fragmented. It is recommended that normal and dense boards have trace hugging enabled. Hugging should be disabled only for boards that are not densely packed (such as many analog boards). This will save some time during the optimizing of the board.

Temporary violation

The rip-up and retry autorouter is capable of routing traces while crossing existing (fixed) traces. This is part of the rip-up and retry algorithm. However, crossing an existing trace causes a very high penalty. When this value is set too low, crossing a trace may be *cheaper* than making a detour or using vias. This can cause the undesirable situation where the global and local rip-up phases become unstable and only continue to create more violations instead of fewer.

90 degree corner

If the design does not have 90° corners, this value must be set to a high value. The default value (relative to the 45° corner cost) prefers 45° corners.

45 degree corner

If the design does not have 45° corners, this value must be set high. The default setting (relative to the 90° corner cost) prefers 45° corners.

Initial cost

This is the cost for a via, relative to a 100 mil detour. Each via gets this cost. If a via crosses multiple layers on a multi-layer board, the “extra cost per layer” is added for each layer-pair that will be crossed.

Extra layer

When a via crosses any layer, it takes up routing space on that layer. This cost will account for that space and, on multi-layer boards, will help to avoid useless layer crossing by vias. It only works when buried vias are allowed.

Offset Vias (for 25 mil grid only)

When this option is enabled, vias are shifted 1/2 grid step to the upper right direction. The via size must be chosen so that the via, including its clearance, is smaller than 2 grid cells.

Buried Vias

When this option is enabled, buried vias may be generated. This is only relevant for multi-layer boards.

T-Connections

When this option is enabled, T-connection or copper sharing is allowed. When deselected, connections are routed from Pin to Pin, not sharing any copper of the same net. For some high-speed applications (ECL or high-speed buses), this may be necessary.

45 degree Pad Entry

When this option is enabled, a trace may start and end on a pad with a 45° segment. For designers who prefer 90° entry and exit on pads, this option must be deselected.

6.3.4.2 Strategy Setting

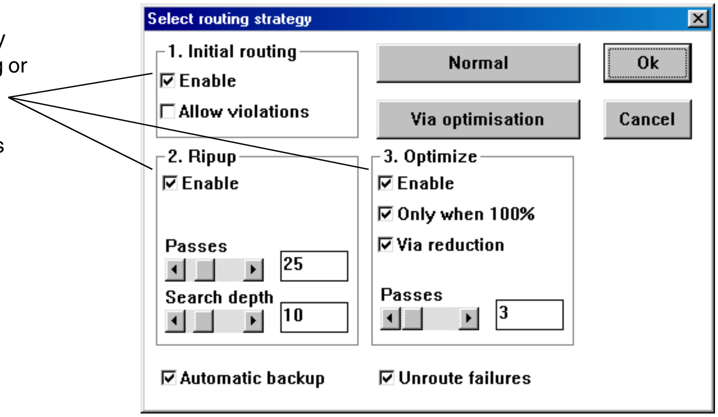
The rip-up and retry autorouter has a fixed sequence of routing strategies. The normal sequence is:

1. initial routing
2. rip-up
3. optimize
4. options
5. automatic backup
6. unroute failures.

You may also allow strategies to run separately by enabling or disabling each strategy.

➢ To run strategies separately, choose **Parameters/Strategy**. The Select Routing Strategy screen appears.

Set strategy by selecting or deselecting the Enable checkboxes



Initial routing

The initial routing strategy routes connections in a special order. This order is determined by the width, length and class of connection. Sorting is done by width, class (bus or non-bus) and length, approximately in that order. There are four initial sub-strategies: generation of via-fanouts, fast initial route (initial 1), intensive initial route (initial 2), and removal of superfluous via-fanouts followed again by an intensive initial route (initial 3). If the **Allow violations** option is enabled, this strategy will leave no unrouted connection, but may create some violations.

Rip-up

The rip-up strategy is used to resolve violations while optimizing the number of vias and the wire length. This is the most powerful algorithm within the rip-up and retry autorouter.

Very often, a board can only be completed as 100% failure-free when a number of vias are removed and the length of some wires is reduced.

The rip-up strategy looks for connections that might be improved by rerouting, and removes them. These connections are then scheduled for routing in a different order from that in which they were routed originally. After each pass, the total quality of the routing is computed and compared to the previous solution. When the new routing solution is an improvement over the previous one, it is kept as the current best solution. In this way, progress is guaranteed.

Optimize

The optimize strategy removes unnecessary bends and straightens detours caused by unnecessary trace hugging. This strategy is run last and should only be run when completion has reached 100% and no failures are left.

When **Via reduction** is enabled (by default), it will remove vias by allowing routing on all layers in all directions at the same cost.

Passes sets the number of iterations.

Automatic Backup

When this option is selected, the rip-up and retry autorouter saves the design to disk after the completion of a strategy. Note that this will not occur **during** execution of a strategy, even when a routing phase takes a long time to complete (like the rip-up strategy).

Unroute Failures

When this option is selected, the rip-up and retry autorouter focuses on violations or failures left by the rip-up and retry passes.

6.3.4.3 Notes on Routing Performance

The autorouter is influenced by certain factors. The production classes (for more information, see “About Production Classes (Technology Files)” on page 3-1) set these parameters automatically to obtain optimal routing speed for each class. If you want to change router settings, you should bear in mind the following factors that influence router speed.

Memory Requirements

The amount of memory required for routing is determined by the grid size and the number of routing layers.

The formula is:

Memory = NrOfLayers * Xsize * Ysize * (1000 / gridsize)2 * 2 bytes

In this formula Xsize and Ysize are given in inches and gridsize (usually 50) is given in mil.

Routing Speed

The speed of the router is not the same for the different strategies. Differences in speed are caused mainly by the time it takes to redraw the traces and pads in the map, which is necessary after each rip-up action.

Initial Routing Speed

This strategy is the fastest when only one redraw is necessary at the beginning of the routing pass. However, when traces wider than one grid cell need to be routed, the routing map is redrawn before and after this trace, which causes a significant slowdown of the initial routing.

Rip-up Speed

During Global and Local rip-up, the routing map is redrawn after some wires have been removed, allowing you to view the router’s progress but causing a slight delay. Another reason these strategies are slower than the initial strategy is that the router now rips up many wires which must be rerouted. This is done to find a better solution to the routing problem.

Optimize Speed

During the Optimize strategy, nets are removed and re-routed one by one. Especially on nets with only one connection, the redraw time of the router map may dominate.

Chapter 7
Preparing for Manufacturing

Return to Presentation

Return to Presentation

Chapter 7
Preparing for Manufacturing

7.1 About this Chapter

This chapter describes the procedures that you should perform to prepare your board for error-free manufacturing. The procedures include checking the board for any design rule errors, as well as optimizing your layout for improved manufacturing yields and reduced production costs.

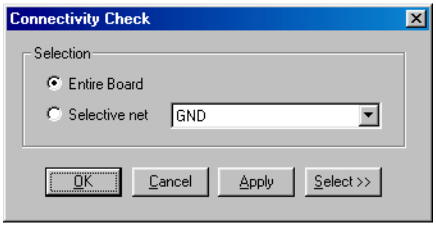
7.2 Checking for Errors

7.2.1 Connectivity Check

The connectivity check ensures that all logical (ratsnest) connections described in the netlist have physical trace connections on the layout. You can run the connectivity check for an entire board, or for just a single net.

➢ To run the connectivity check:

1. Choose **Tools/Connectivity Check**. The Connectivity Check screen appears.



Return to Presentation

2. To check the entire board, select **Entire Board**. To check a single net, select **Selective Net** and click on the desired net from the drop-down list.

OR

2. Click **Select** and then, in the workspace, highlight a pin or trace belonging to the net you would like checked.
3. To temporarily apply your changes, click **Apply**. To apply your changes and close the screen, click **OK**. To cancel your changes, click **Cancel**.

The results of the connectivity check are displayed both graphically and as a report written as a text file. Netlist connections with no corresponding physical connections in the layout are visually identified by white, straight lines between pins. The report is given the same name as your design, with the extension `.con`.

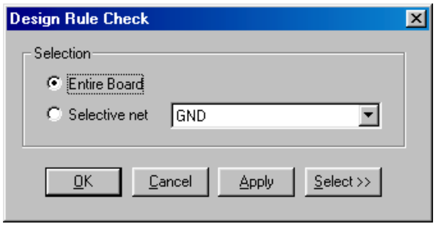
Note The connectivity check only looks for pins that are connected in the netlist, but that are not physically connected in the layout. It does not check for pins that are physically connected in the layout, but not connected in the netlist. Such connections are considered potential design rule violations, and are identified using the design rule check function.

7.2.2 Design Rule Errors

The design rule check looks for any design rule violations that may exist on your board. Successfully running a design rule check is essential for producing an error-free board. The check can be performed on the entire board, or just on a single net.

➢ To run the design rule check:

1. Choose **Tools/Design Rule Check**. The Design Rule Check screen appears.



2. To check the entire board, select **Entire Board**. To check a single net, select **Selective Net** and click on the desired net from the drop-down dialog box.

If errors are found the results are posted to a text file (`.err`). Errors are also displayed graphically by white circles indicating the coordinates where the error occurred.

Load the `.err` file to view the errors. Correct them and then rerun the design rule check. The white circles on the pins should disappear. Switch to the workspace screen and click to check a net for connectivity.

Note The design rule check described here is even more stringent in looking for errors than the real-time design rule check (described in “Real-Time Checks” on page 3-17). For example, the real-time design rule check will allow you to place a trace over a copper area of a separate net. This type of error will be detected with the **Tools/Design Rule Check** command. For this reason, it is critical that you always run a design rule check before you complete your designs.

The results of the design rule check are displayed visually in the screen, and are summarized in a report file with the same name as the design, with the extension `.err`. Design rule violations are identified on the screen as locations marked with small white circles.

7.3 Cleaning up the Board

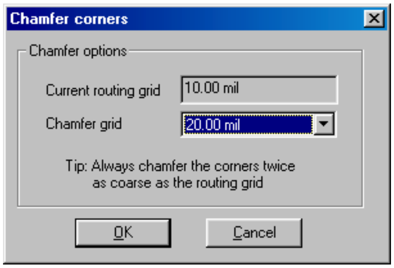
Ultiboard includes a number of features that clean up your boards before they go to production. Cleanup functions improve your production yields and result in more efficient layouts.

7.3.1 Chamfer (Miter) Corners

The chamfer corners command changes 90° routing angles to 45° angles to improve manufacturing. The chamfer grid (selected in the Chamfer Corners screen) refers to the legs of the chamfer area. This is the equal distance from an orthogonal corner back to where each vertex marking an end of the chamfer segment is located.

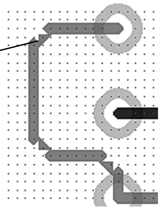
➢ To use the chamfer corners command:

1. Choose **Tools/Chamfer corners**. The Chamfer corners screen appears.



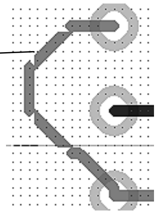
2. From the drop-down list, select the desired chamfers grid. The chamfers grid is used to define the orthogonal distance between two vertices of a chamfer segment.

Orthogonal distance between two vertices of a chamfer segment is two spaces.



Chamfer grid set to 20 mil

Orthogonal distance between two vertices of a chamfer segment is five spaces.



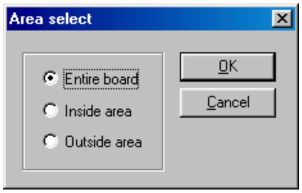
Chamfer grid set to 50 mil

Note By default, the chamfer grid is always set to double the routing grid (mouse grid). It is advisable to use this default setting.

7.3.2 Delete Open Trace Ends

The **Trace/Delete/Delete open trace ends** command removes any trace segments that do not have any terminating connections. As a result, all unnecessary copper is removed from your board. You may remove open trace ends over the entire board, inside a user defined area, or outside a user-defined area.

➢ To delete open trace ends, choose **Traces/Delete/Delete Open Trace Ends**. The Area select screen appears. Select the desired option from the screen and click **OK**.



7.3.3 Delete Unused Vias

The **Trace/Delete/Delete unused vias** command allows you to delete all vias that do not have any trace segments or copper areas connected to them. Use this command to clean up your board by removing unnecessary or excess vias on your board.

- To delete unused vias, choose **Traces/Delete/Delete Unused Vias.**

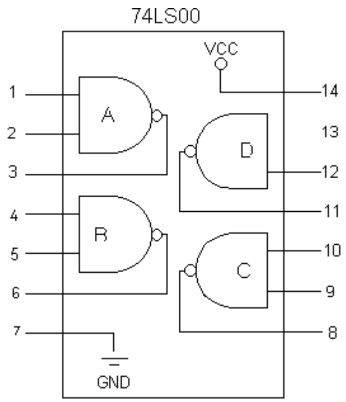
Note To remove vias that do have trace or copper connections, choose **Traces/Delete/Delete Single Via**.

7.4 Pin and Gate Swap

7.4.1 About Pin and Gate Swaps

When making pin connections at the schematic level, you choose the pin and gate connections which produce the most aesthetically pleasing, easy-to-read schematic diagram. At the PCB layout level, switching functionally equivalent pin and gate connections can often making routing easier.

Some, but not all, components will allow swapping. The 74LS00, for example, shown below, has four identical gates which can be swapped at the PCB level without changing circuit functionality.



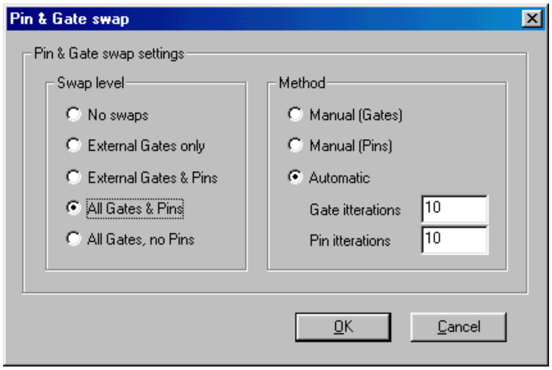
A pin swap exchanges pins with identical functionality. A gate swap exchanges gates with identical functionality. Ultiboard allows you to choose between different levels and methods of swapping. These options are described below.

Return to Presentation

7.4.2 Performing a Pin and Gate Swap

➢ To perform a pin and gate swap:

1. Choose **Tools/Pin & Gate Swap**. The Pin & Gate swap screen appears.



2. Select the level of swap you would like to use:

No swaps	Disallows swapping.
External Gates only	Swaps gates between identical components.
External Gates & Pins	Swaps gates and pins between identical components.
All Gates & Pins	Swaps gates and pins within a single component and between identical components.
All Gates, No Pins	Swaps gates within a single component and between identical components.

3. Under the **Method** options, select **Automatic** or **Manual** swap. Select **Manual (Gates)** or **Manual (Pins)** swap to specify the exact pin or gate that you wish to swap. Select **Automatic** swap to have Ultiboard automatically swap pins and gates that result in shorter or more efficient trace connections.
4. If you selected **Automatic** swap, enter the number of iterations the program should use in trying to perform available pin and gate swaps.
5. Click **OK** to perform the swap, or **Cancel** to cancel.

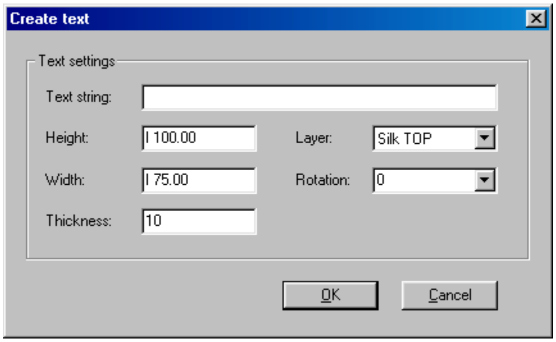
Return to Presentation

7.5 Working with Text

Ultiboard lets you place text on any layer of a design, including both the silkscreen and top routing layers.

➤ To place text:

1. Choose **Texts/Place**. The Create text screen appears.



2. Enter the text you want placed in the **Text string** field.
3. To adjust the size and appearance of the text, enter the desired value in the **Height**, **Width**, and **Thickness** fields.
4. From the **Layer** drop-down list, select the layer on which you would like the text to appear.
5. From the **Rotation** drop-down list, select the desired degrees of rotation to control the orientation of the text.

➤ To edit text, choose **Text/Edit** and then click on the text, or simply double click on the text you want to edit.

➤ To move text, choose **Text/Move**.

➤ To delete text, choose **Text/Delete**.

7.6 Renumbering Components

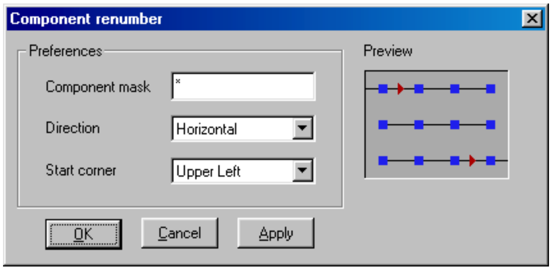
Renumbering components automatically renames all components in the order that you specify. It is easier to produce, service, and troubleshoot boards when components are ordered in a logical manner.

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.

Return to Presentation

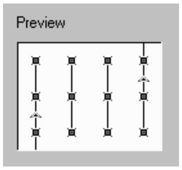
➢ To renumber components:

1. Choose **Tools/Renumber Components**. The Component renumber screen appears.

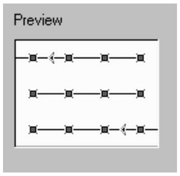


1. From the **Direction** drop-down list, select the desired direction.
2. From the **Start corner** drop-down list, select the desired start corner.

A preview of the renumbering strategy appears in the **Preview** area.



Preview of a renumbering strategy set to start in the lower left corner and renumber in a vertical direction



Preview of a renumbering strategy set to start in the lower right corner and renumber in a horizontal direction

3. To temporarily apply your changes, click **Apply**. To apply your changes and close the screen, click **OK**. To cancel your changes, click **Cancel**.

Return to Presentation

Chapter 8
Postprocessing

Return to Presentation

Postprocessing

Return to Presentation

Chapter 8
Postprocessing

8.1 About this Chapter

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

8.2 Introduction to Postprocessing

Postprocessing is the process of producing complete information describing how a finished board is to be manufactured. It is from this information that a manufacturer is able to produce a physical copy of the boards you design in Ultiboard.

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.

Note It is important to talk to your production house and identify all the files and formatting information they need to support their manufacturing process.

This command creates an Intermediate Plot File (`.ipf`) in the same folder as the design file. The Intermediate Plot File contains all data required to create your output. When the file is created, the Postprocessor starts.

If you use this command to print the powerplane layout and find that the copper coat is not shown on the hard copy, use **Traces/Copper area** instead. Make sure to enclose the entire board layer area.

Return to Presentation

8.2.1 Types of Output

Ultiboard will create output for the following:

- Photoplotters (Gerber compatible)
- Penplotters (both HPGL and DPMI compatibles)
- Windows printers (GDI)
- Excellon drill format
- Gerber file output
- DXF output.

Designs are usually sent to manufacturers as formatted graphics files. Gerber is the most common such format, and is read by Gerber and other photoplotters.

8.2.2 Overview of Steps

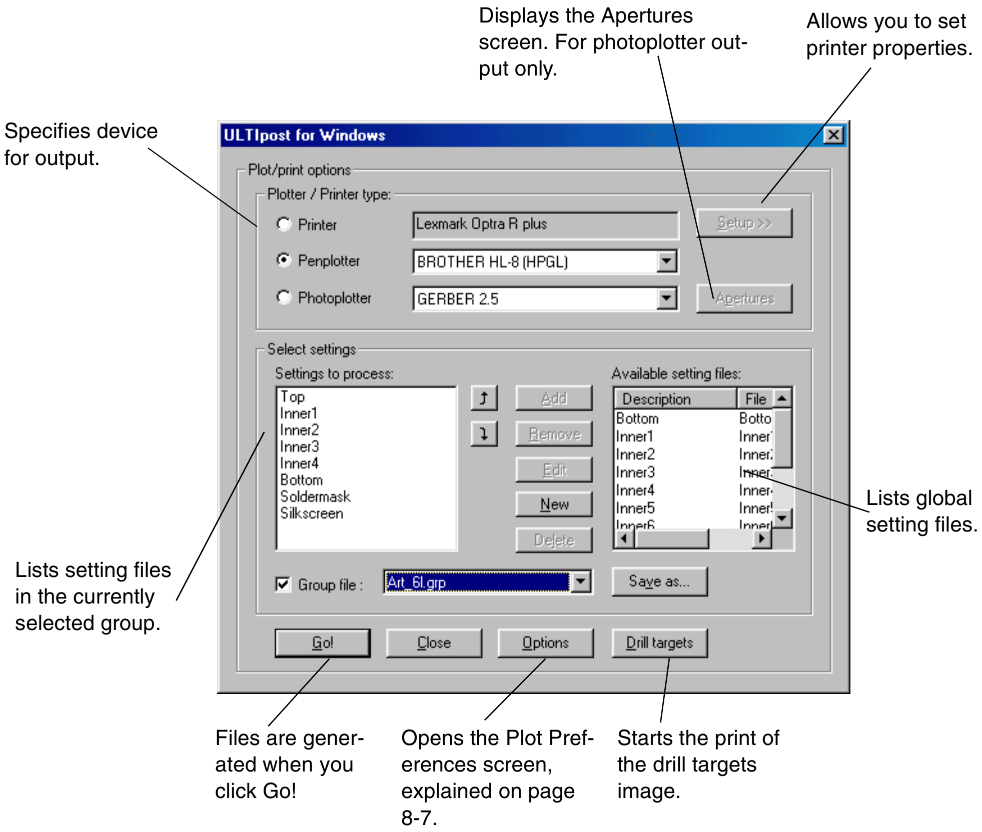
To produce output for your board, you need to do the following:

- display the postprocessing screen (see page 8-3)
- choose the device for which the output should be created and optionally change the properties for the device (see page 8-9)
- determine which layers (settings file) are to be output by choosing a group file (see page 8-3)
- optionally, modify the parameters for each settings file (see page 8-5)
- when your settings are complete, click **Go!**

Return to Presentation

8.2.3 The Postprocessing Screen

To create output, choose **File/Postprocessing**. The postprocessing screen appears.



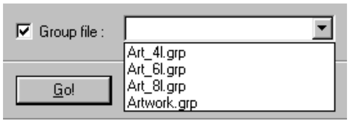
8.3 Working with Group Files

Ultiboard lets you produce separate output files/prints/plots for each layer. This gives you the ability to include or omit objects and properties (such as pads, traces, vias, and component values) on a per-layer basis. This mechanism is controlled through group files, which contain a series of settings files that represent the layers to be included in the output.

Return to Presentation

➢ To use Ultiboard's default settings files for creating output:

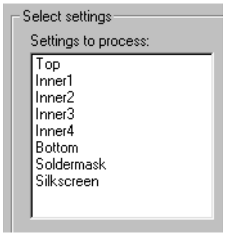
1. On the postprocessing screen, enable the **Group file** checkbox.



2. From the drop-down list, choose from among the following choices depending on the number of layers in your board:

Artwork.grp	for 2 layer boards
Art_4l.grp	for 4 layer boards
Art_6l.grp	for 6 layer boards
Art_8l.grp	for 8 layer boards

Each group file loads a file setting into the **Settings to process** list for the soldermask, silk-screen, top, and bottom layers (as well as all internal layers, depending on the total number of layers used). For example:



➢ To edit or create your own group files:

1. To add settings from the **Available setting files** list, click on the appropriate setting files, and choose **Add**. The selected files are moved into the **Settings to process** list.
2. To add new files to the **Available setting files** list, choose **New**.
3. To delete files from the **Available setting files** list, choose **Delete**.
4. Choose **Save as**. Name your file and click **Save**.

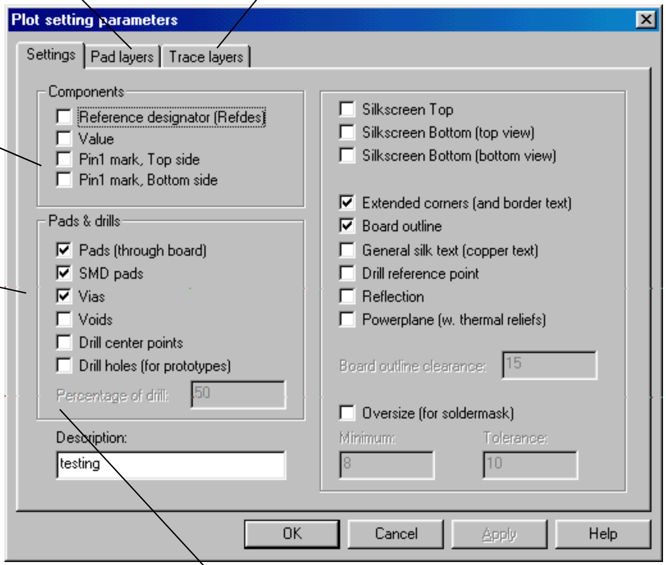
If you make any changes to files from a group file, Ultiboard will automatically ask you if you want to save your changes.

Return to Presentation

- To remove files from the **Settings to process** list, click on the appropriate file and choose **Remove**.
- To change the order of any file in the **Settings to process** list, click on the appropriate file and choose the up or down arrow button to move it in the direction you want.
- To save the current set of settings files as a group, enable the **Group file** option, click **Save as** and specify a name for the group.

8.4 Settings Files Parameters

- To view what will be output for each layer setting, double-click on the appropriate setting file in either the **Settings to process** or **Available setting files** list. The Plot Setting parameters screen appears.
- To create a settings file, click **New** on the postprocessor screen. The Plot Setting parameters screen appears.



Return to Presentation

The available parameters for each file are:

Extended corners	Places corner markers defining a rectangle which takes into account all items extending past the edge of the official board dimensions such as connectors, text etc. Useful for seeing the maximum area taken up by the board with any bits sticking out. Note that the extended corners are no reference for the outline of the board itself.
Board outlines	Toggles the board outline on/off.
Vias	Toggles vias on/off.
Pads (through board)	Toggles pads on/off.*
SMD pads	Toggles SMD pads on/off. (An SMD pad is a pad without a drill hole, and is also used for edge connectors.)*
Pin 1 mark, Top	Toggles marking of Pin 1 for the Top layer on/off.*
Pin 1 mark, Bottom	Toggles marking of Pin 1 for the bottom layer on/off.*
Voids	Generates a file with all drill holes of the specified layer.*
Drill reference point	Toggles the drill reference points on/off.
Drill center points	Enables the drill center holes. This will only create a dot at the place to be drilled, not a center hole in the pad. To create a real center hole, see the Drill holes option.
Drill holes	Generates drill holes in the pads for prototyping. Remember that this option has to be used together with the size (percentage of pad diameter) of the drill hole to set the center point.
Reflection:	Reflects the output. This is used to get the image printed so that the emulsion is on the correct side of the film.
Oversize	Toggles oversize on/off. Used for the soldermask.
Negative plane layer	Creates a negative plane layer with thermal reliefs. Used for power planes.
Silk, Top	Toggles the silkscreen for the top layer on/off.
General silk text	Toggles the general silkscreen text on/off, The silkscreen text is created within Ultiboard on the Silk Top and Silk Bottom layer.
Component name	Toggles the component name (U1, R5 etc.) on/off.

Return to Presentation

Extended corners	Places corner markers defining a rectangle which takes into account all items extending past the edge of the official board dimensions such as connectors, text etc. Useful for seeing the maximum area taken up by the board with any bits sticking out. Note that the extended corners are no reference for the outline of the board itself.
Component value	Toggles the component value (74LS00, 2k2 etc.) on/off.

* These items are only processed together with the Pad layers that you select.

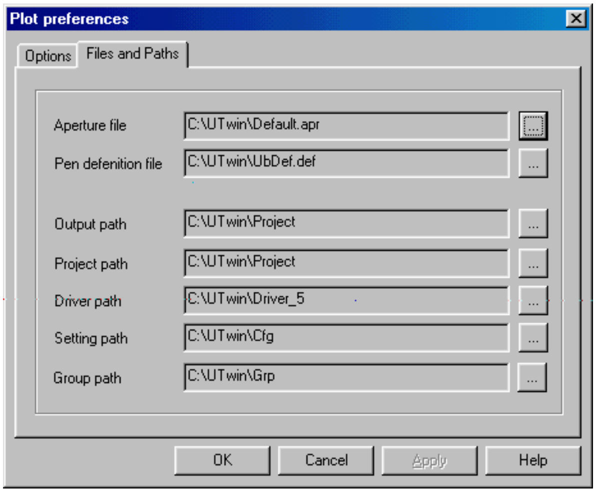
To edit any of the setting files, click on the appropriate file and choose **Edit**. Changes made to files in the **Available settings files** list are stored for later re-use. The settings that are processed are saved in the group file.

8.5 Plot Preferences

Use to change or set general printing parameters.

8.5.1 Files and Paths Tab

Use to set the location of files used for Postprocessing.

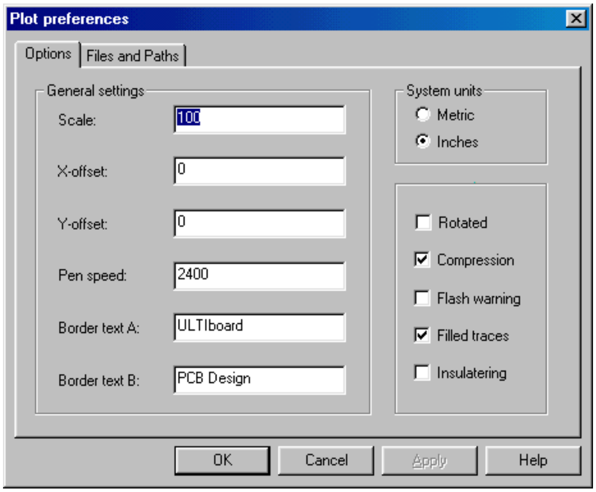


Return to Presentation

Aperture file	Selects the default aperture file (.apr) which is used to generate the Gerber output.
Pen definition file	Selects the default pen definition file (.def) used to generate pen plotter output.
Path settings	Sets the default directories for the different types of postprocessor files.

8.5.2 Options Tab

Use to set preferences for plotter or printer output.



Scale	Sets the scale for your plot/print output as a percentage of the actual design dimensions. The plot/print output may be scaled from 10 to 500%.
Offset	Sets the offset for the plot/print output. The offset must be specified in plotter units. You must be familiar with the X and Y directions and plotter units of your plotter to use this feature correctly.
Pen Speed	Sets the speed at which the plotter should draw your output. Pen speed is specified in plotter units per second. When the pen speed is set to 0, the pen speed setting from the pen definition file (.def) is used. In the definition file the speed is given separately for artwork and documentation files.

Return to Presentation

Border text	Enter the border text which is plotted next to the design on each plot. This message is split into 2 lines (A and B). If no value is assigned to these message strings, text A is “Ultiboard” and text B is “PCB design”. You may fill in your company name and/or department.
System units	Selects the units of measurement for the drill tool size, when drill targets are to be plotted/printed.
Rotated	When enabled, rotates the plot/print output 90 degrees.
Compression	Eliminates very small plotter steps (1 or 2 plotter units) from output to the plot file. This causes a reduction of the output file size, which may be useful. It also causes a very minute loss of accuracy which is apparent under a close examination of the plot quality. In some instances, depending on how many small steps your design creates, there may be some acceptable “jaggies”. You should be aware that close tolerance technologies, such as two or more traces between DIP pins and some SMT layouts, may not accept such minor variances.
Flash warning	When enabled, the system tries to flash all pads and draw all other objects with the exact size using the available flash and draw apertures in the selected aperture file (*.apr). If the aperture needed to flash or draw the object is not available in the aperture file, the system displays the aperture size that is not found and stops generating the output files. When the Flash Warning option is disabled, the system flashes the pads with an aperture that may be 4 mils smaller at most. If such an aperture is not available, the pad/object is drawn with the smallest aperture available.
Filled traces	Determines whether to plot/print traces as solid lines or two parallel lines. Disable this option for faster documentation prints/plots. This option does not influence the Gerber output.
Insulate ring	Generates powerplanes. The Postprocessor generates a full flash (size of the pad + clearance) for all pads which are not connected to the plane (default). It will generate insulation rings around those pads when this option is enabled. Be aware that the size of the Gerber plot file increases significantly when “insulation rings” are used. For other plot formats, this may not be a problem.

8.6 Output Devices and Formats

8.6.1 Printers

Ultiboard prints to the default Windows printer. To change the settings for this print, select **Printer**, click **Setup** and enter the desired changes on the screen.

Return to Presentation

8.6.2 Pen Plotters

For pen plotters, Ultiboard generates an output file for each layer of the board. The first file processed from the **Settings to process** list will have the extension `.p0`, the next file in the list will have the extension `.p1`, and so on.

8.6.3 Photoplotters

Photoplotters produce images of boards by projecting light through apertures (physical holes). Differently shaped apertures produce differently shaped images when light is transmitted from the plotter's lamp. Plotters use two basic operations: draw and flash. Draw produces a continuous line segment, while flash projects an instantaneous image.

Photoplotters come in two varieties: mechanical plotters and laser plotters. Mechanical plotters use a physical aperture wheel with a limited set of apertures. Laser photoplotters, on the other hand, can handle any aperture size.

8.6.3.1 Plotter files

When producing plotter output, Ultiboard creates a file with draw and flash commands to create an image of the board. Pads can be easily plotted by "flashing" the plotters' lamp on and off using an aperture with a shape that matches the pad. However, pads that have no matching aperture on the photoplotting device must often be built up by drawing the pad shape using smaller apertures. This process is considerably slower than producing pads using flashes, and results in larger data files since more commands are needed to draw, rather than flash, a pad.

8.6.3.2 Producing Plot Files with Ultiboard

When Ultiboard starts creating a plot file, it identifies which aperture sizes are required to produce the objects on your layout and attempts to locate the apertures in a list of all available apertures. Ultiboard provides two aperture files:

- `DEFAULT.APR`, which is used for mechanical photoplotters that use physical aperture wheels with a limited set of apertures.
- `LASER.APR`, which is used for laser photoplotters that can handle any aperture size.

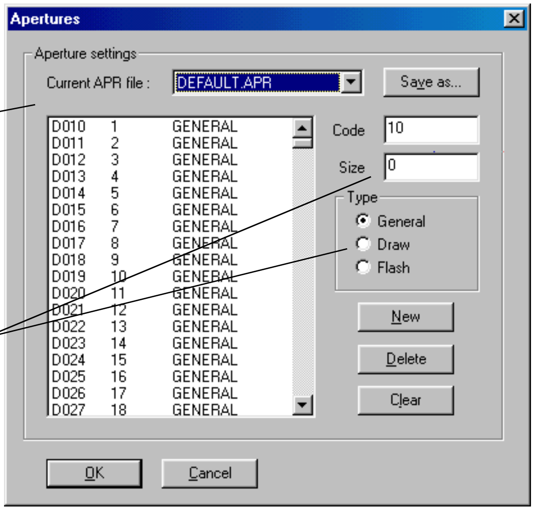
Suppose the `DEFAULT.APR` file is selected and Ultiboard is trying to locate a 60 mil aperture. If the 60 mil aperture does not exist, Ultiboard tries to find the nearest aperture size within a margin of 4 mil (smaller or larger). If no aperture can be found within 4 mil, Ultiboard selects the smallest available aperture to draw the pad.

Return to Presentation

➢ To change the list of available apertures, enable **Photoplotter** and click **Apertures**. The Apertures screen appears.

Select the desired aperture file. The apertures in the file are listed in the area below this field.

Information on the currently selected aperture. Size is in mils.



It is recommended that you always check with your photoplot manufacturer to see which pad sizes can be physically produced. You may want to create your own aperture files to match the apertures used by your photoplot manufacturer.

➢ To save the current aperture settings to a file, click **Save As** and specify a file name.

➢ To add an aperture code to the file, click **New**.

➢ To delete an aperture code from the file, click **Delete**.

➢ To clear all aperture codes from the file, click **Clear**.

The first file processed has an extension `.g0`, the next has extension `.g1`, and so on.

8.6.4 Gerber

8.6.4.1 About Gerber Files

Ultiboard produces Gerber files of your PCB layouts. Gerber files contain combinations of commands to select apertures, to move to an X-Y coordinate, and to perform other miscellaneous operations.

Return to Presentation

Below are some common Gerber commands:

Preparatory Functions (G*)

Preparatory functions are identified with a leading "G" and are followed by a 2-digit integer. Preparatory functions describe processing functions.

Examples:

```
G01 = Go to
G54 = Select aperture
```

Draft Codes (D*)

Draft codes consist of a leading "D" followed by two digits to open and close the aperture shutter and select one of the available apertures of an optical exposure head.

Examples:

```
D12 = Select Aperture 12
D15 = Select Aperture 15
```

Miscellaneous Codes (M*)

Miscellaneous codes consist of a leading "M" followed by two digits which may represent miscellaneous commands.

Examples:

```
M02 = Program stop
```

8.6.4.2 Producing Gerber Files with Ultiboard

Ultiboard can output to the two most popular Gerber formats: RS-274D and RS-274X. The RS-274D format requires a separate or previously defined aperture list, while the RS-274X (extended Gerber) format automatically includes appropriate aperture shapes for each artwork layer.

Note Check to see which format your manufacturer prefers.

The Gerber files you produce with Ultiboard can have different resolutions. Gerber M.N means that coordinate values are written using M integers followed by N decimal digits.

For example:

```
Gerber 2.3 = XX.XXX
Gerber 2.4 = XX.XXXX
Gerber 2.5 = XX.XXXXX
```

When the Gerber file is actually written, there are no decimal places inserted into the coordinate values. Decimal places are implied by the format of the Gerber file. For example, the

Return to Presentation

coordinate value 34275 with Gerber 2.3 represents the value 34.275. The same value with Gerber 1.4 would represent 3.4275.

Ultiboard also produces a report file with the project name and the extension `.rep`. This is useful to give to the manufacturer since it contains a complete list of apertures used, as well as the N.M format for Gerber coordinates.

8.6.5 Creating Drill Center Holes

You may wish to create drill center holes in the pads for in-house prototyping. You can create output for all Windows printers.

➢ To create holes in your output:

1. On the postprocessing screen, from the **Files to process** list, select the top layer and click **Edit**. The Plot Setting parameters screen appears.
2. In the Settings tab, under the Pads & drills options, enable **Drill holes (for prototypes)**. Leave the default percentage of drill setting at 50.

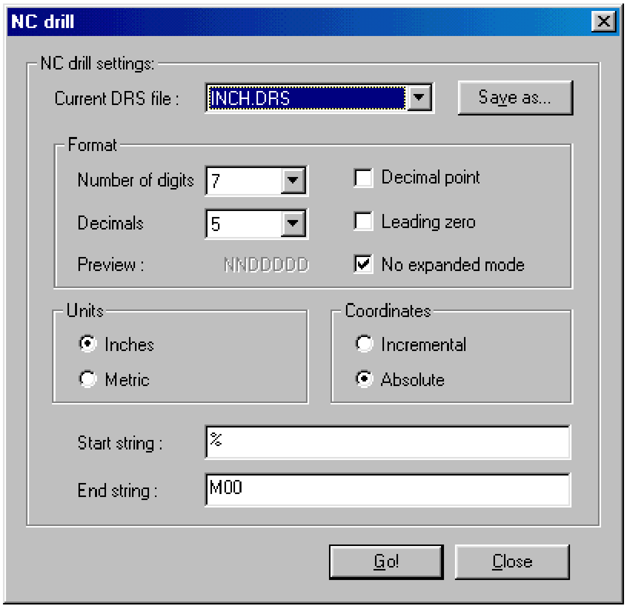
Photoplot/prototype service bureaus often need an excellon drill file in addition to Gerber files to create a finished board. Drill file settings vary with each prototype manufacturer. Contact your photoplot/prototype manufacturer before creating the output and obtain all the required information to produce your files.

Note You can save the settings for the format of the drill file in a drill setting (`.drs`) file and use the file to restore the settings for other designs.

Return to Presentation

➢ To create a drill file:

1. Choose **File/Drill**. The NC Drill screen appears.



The screen contains the following fields:

Current DRS File	Selects the drill setting file (.drs), containing the drill data output settings used to generate the Excellon drill file.
Save as...	Saves the current drill settings to disk.
Format	Sets the coordinate format of digits in the drill data output: total number of digits and number of decimals. You can preview your settings in the same screen, as you make the changes
Decimals	Toggles the decimal point in the drill data output on/off.
Leading zero	When enabled, adds leading zeros in the coordinates of the drill data output.
No expanded mode	Toggles the expanded mode on/off. Expanded mode means that all X- and Y-coordinates are output, even when they have not changed (otherwise those coordinates are omitted).
Units	Select metric (millimeter) or inch units of measurement for the drill data output.

Return to Presentation

Coordinates	Selects absolute or incremental coordinates for the drill data output. **Absolute** will output all drill positions relative to the origin (0,0) location in your design, **incremental** will output each drill position relative to the previous drill position.
Start/End string	Enter the starting and ending string for the drill data output. You should be familiar with Excellon code to change these from their defaults.

2. Click **Go!**

The `*.REP` file is appended with the drill data and the system creates the drill file (`*.D0`).

8.6.6 Mechanical CAD (.DXF) Files)

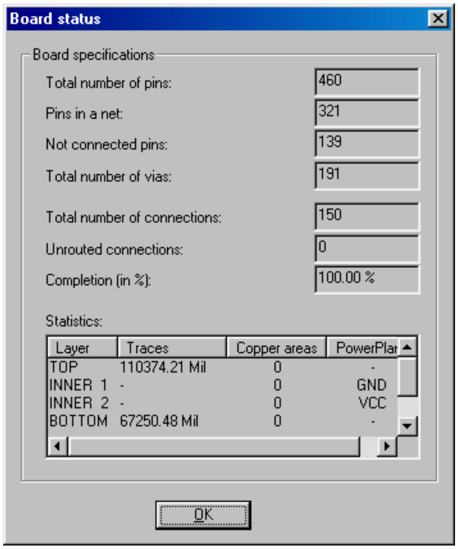
Ultiboard can produce a DXF file representing the physical layout of objects on your board. This option is often used to transfer your board layouts to other tools capable of reading DXF files.

➢ To create a DXF file, from the postprocessing screen enable **Penplotter** and select **DXF pcb layout** from the drop-down list.

8.7 Statistics and Report

Ultiboard can also produce general reports and statistical information on your PCB layouts.

➢ To obtain statistics on your board, choose **Tools/Info/Board Status**. The Board status screen appears.



Total number of pins	Total number of pins on the board.
Pins in a net	Number of pins that are connected in the net (logically rather than physically).
Not connected pins	Number of pins that are not connected in the net (logically, not physically).
Total number of vias	How many vias are on the board. This is useful for estimating production costs and for evaluating and comparing autorouting results to see how many vias are produced by an autorouting result.
Total number of connections	Total number of logical, not physical connections (ie. from the netlist).
Unrouted connections	Number of logical connections that do not have physical connections.
Completion	Number of routed connections divided by total number of connections. Useful for determining autorouting completion percentage.

Statistics	Total amount of copper used on the board. It gives you a summary value (for the whole board) and also splits up the values by layer. The traces value gives the total length of all traces. The copper areas value tells you how many separate copper areas are used. The power planes value tells you how many power planes are used. The traces value is useful for estimating production costs and also for evaluating and comparing different autorouting results to get a general feeling for the length of traces used.

Return to Presentation

Commands and Menus

Return to Presentation

Commands

Return to Presentation

Return to Presentation

Return to Presentation

Return to Presentation

Commands

Return to Presentation

Chapter 9
Commands and Menus

9.1 About this Chapter

This chapter describes the Ultiboard menus and their commands.

9.2 File Menu

9.2.1 File/New (CTRL+N)

Use to create a new design in Ultiboard. You are prompted to choose a production class.

For more on production classes, see “About Production Classes (Technology Files)” on page 3-1.

For more on this command, see “Starting a New Design File” on page 2-2.

9.2.2 File/Open (CTRL+O)

Use to open an existing Ultiboard design file. If you already have an Ultiboard design file open, the file is automatically closed. Only one design file may be open at a time.

For more on this command, see “Starting a New Design File” on page 2-2.

Return to Presentation

9.2.3 File/Close

Use to close the current Ultiboard design file. You are prompted to save any unsaved changes. If you have not saved the file before, you are prompted to specify a folder and file name.

For more on this command, see “Closing a Design” on page 2-3.

9.2.4 File/Save (CTRL+S)

Use to save the active design to its current name and folder. Use the Save As command if you want to change the name and folder of the active design before you save it. The first time you save a design you will be prompted for a folder and file name as well as a production class.

For more on this command, see “Saving a Design” on page 2-4.

9.2.5 File/Save As

Use to save the active design to a different location or with a different name than that of the original. You are prompted to enter the name and location of your design.

9.2.6 File/Post processing

Use to generate output to document the design so the PCB can be manufactured, tested and assembled.

For more on this command, see “Introduction to Postprocessing” on page 8-1.

9.2.7 File/Drill

Use to create the Excellon drill (`.d0`) file, containing the drill information for the PCB design.

For more on this command, see “Creating Drill Center Holes” on page 8-13.

Return to Presentation

9.2.8 File/Import

Use to import components, netlists, or design rules into Ultiboard.

9.2.8.1 File/Import/Components

Use to import either a components file (`.plc`) generated by a schematic capture program or a component placement file (`.cmp`) generated by Ultiboard.

Importing a component list is normally the fastest way to add components to a design. After the components list is imported, you are prompted to import the netlist file containing connectivity information for the imported components.

If, after importing your components to the workspace, you notice that some footprints are imported as question marks, this means there is no known shape found in the libraries for that symbol. You must create your own symbol using the Shape Editor. Ultiboard also produces an error file that tells you where errors have been encountered. You are prompted to view this error file if you wish.

For more information on the `.plc` file, see “Understanding .net and .plc files” on page 3-18.

For more about this command, see “From Component List Files” on page 4-3.

9.2.8.2 File/Import/Netlist

Use to import a netlist from a file.

You must import your components before importing a netlist or the command fails and produces the message: “Netlist Import errors”. After you select a netlist file to load/import, the system prompts you for the default trace code. This trace code is assigned to all nets that do not have a pre-defined width attribute in the netlist file. All netlist import errors are reported in a file named `import.log`.

Always import the component list `.plc` file first before loading the netlist (`.net`), otherwise the **Import Netlist** command fails.

Ultiboard saves the new netlist information with the extension `.nt0` and the new component information with the extension `.pk0` in the design folder during the netlist import/load. These files are used during the backannotation process.

For more about `.net` files, see “Understanding .net and .plc files” on page 3-18.

Return to Presentation

9.2.8.3 File/Import/Design Rules

Use to import design rules from a technology (`.tec`) file.

The design rules are stored with the design and describe all pad, via and trace properties used by the system. You can import all the design rules or just parts of the design rules from a technology file.

For more information on design rules, see "Setting Design Rules" on page 3-13.

9.2.9 File/Export

9.2.9.1 File/Export/Component List

Use to export the component list (`.csv`) file. The file is saved to the project folder by default.

The component list file contains data describing each part by reference designator, alias (value) and component shape. It can be used for generating a bill of materials.

9.2.9.2 File/Export/Placement

Use to export the component placement (`.cmp`) file. The file is saved to the project folder by default.

The component placement file contains data describing each part by reference designator, alias (value), component shape, rotation and location. It is used to recreate a PCB design placement. Placement data can also be used for interfacing to auto pick/place and auto-insertion machines.

9.2.9.3 File/Export/Netlist

Use to extract the netlist of the PCB design file and save it to the project folder as an ASCII file with an `.ntl` extension.

Use the netlist file in Ultiboard or other programs requiring a netlist input. The netlist file can also be useful for visual and computer checking of the design, depending on your needs and your schematic capture program.

9.2.9.4 File/Export/Documentation

Use to produce a status and summary report on the design database. The report is saved in the design folder as a documentation file (`.doc`). It contains information on board size, area, total trace length, components used, number of drill tools, number of holes and rules used in the design.

This report is useful for a PCB cost estimate.

9.2.9.5 File/Export/Centroid File

Use to export the component centroid (`.ctr`) file. The file is saved to the project folder by default.

9.2.9.6 File/Export/Design Rules

Use to export the design rules (pad, via and trace properties) of the currently loaded PCB design into other designs.

You can export all the design rules or just parts of the design rules to a named technology (`.tec`) file. By default, this technology file is saved in the Ultiboard project folder.

For more information on design rules, see "Setting Design Rules" on page 3-13.

9.2.9.7 File/Export/Design File v4.80

Use to export the active design in version 4.80 file format. This can be used to transfer a design to a previous version (lower than version 5.5) of Ultiboard. The older format only supports a database format of 1/1200 inch. This means that rounding errors could occur in the current design.

Ultiboard displays the Save As screen so you can name your design.

9.2.10 Most Recently Opened Files

To access recently opened files quickly, without using the **Open** command, you can choose from among the four most recently opened files listed at the bottom of the **File** menu.

9.2.11 File/Exit

Use to end your Ultiboard session. You are prompted to save any open documents with unsaved changes.

Return to Presentation

9.3 View Menu

9.3.1 View/Standard

Use to display or hide the Standard toolbar, which includes buttons for some of the most common commands in Ultiboard.

9.3.2 View/Display

Use to display or hide the Display Toolbar, which includes buttons for zoom functions and Redraw Screen.

9.3.3 View/Birdseye

Use to display or hide the Birdseye View.

The Birdseye View represents the total design area. The white rectangle within the Birdseye View represents the visible area of the workspace. The Birdseye View lets you tell at a glance exactly where your current workspace is located in the overall design. You can navigate quickly around the design by selecting a workspace area with the mouse.

For more about the Birdseye View, see “Birdseye View” on page 2-5.

9.3.4 View/Trace

Use to display or hide the Trace Toolbox. When the Trace Toolbox is visible you can quickly change trace properties (method, type, code, drawing angles and layer) and via properties (via code and via shift) for new traces and vias to be placed.

For in-depth information about traces, see “Drawing Traces” on page 5-4.

For more information about the Trace Toolbox, see “Trace Toolbox” on page 2-4.

9.3.5 View/Status Bar

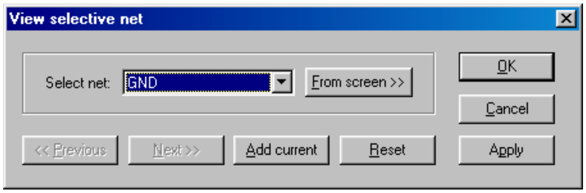
Use to display or hide the status bar.

9.3.6 View/Nets

Use to view all copper (traces, pads and copper areas) in one or more selected nets. This gives a good overview of how the nets are spread around the design. Ultiboard shows only the copper in selected nets.

➢ To select a net for viewing:

1. Choose **View/Nets**. The View selective net screen appears.



2. To select a net for viewing, do one of the following:

- Click **From screen** and, on the workspace, click on a net. The selected net now appears in the drop-down list. Click **Apply**, and only the selected net is displayed in the workspace.

OR

- Select the net from the drop-down list of available nets.

3. To view more than one net, click **Add current** and select the next net to view.

Click **Previous** and **Next** to display each successive available net.

Click **Reset** to return to the full screen view of your board.

9.3.7 View/Zoom Full (F7)

Use to adjust the view of the workspace to include the entire design.

Return to Presentation

Commands

9.3.8 View/Zoom In (F8)

Use to reduce the view of the workspace, using the mouse cursor as center position. Provides a closer look at the current design.

9.3.9 View/Zoom Out (F9)

Use to increase the view of the workspace, using the mouse cursor as center position.

9.3.10 View/Zoom Window

Use to magnify a selected part of the board. Ultiboard prompts you to click the opposite corners of a rectangular area on the board, and then zooms in on that section.

Use this command when you want precise control over the workspace view.

9.3.11 View/Redraw Screen

Use to redraw the current workspace.

9.3.12 View/Top

Use to toggle the view of the top layer on and off.

9.3.13 View/Inner

Use to toggle the view of inner layer # on and off.

Return to Presentation

9.3.14 View/Bottom

Use to toggle the view of the bottom layer on and off.

9.4 Traces Menu

9.4.1 Traces/Place

Use to start routing traces manually.

You can remove a trace segment with the **Trace/Delete** command or by overdrawing the trace segment with the **Trace/Place** command. If you cancel the new trace by clicking the right mouse button or pressing ESC, you are automatically able to move traces.

For more on this command, see “Drawing Traces” on page 5-4.

9.4.2 Traces/Move

Use to move traces. As you adjust the routing in a design, the “reroute while move” feature automatically drags traces and maintains spacing between traces.

For more on this command, see “Moving, Dragging, and Deleting Traces” on page 5-9.

9.4.3 Traces/Delete

For more detail on the delete trace commands, see “Deleting Traces” on page 5-11.

9.4.3.1 Traces/Delete/Delete Segment

Use to delete only one segment. Click on the trace segment to select it. When the trace segment is selected and you move the mouse, a rubber band line appears between the cursor and the point of selection. Click one more time to delete the trace segment. To cancel the deletion, press ESC.

This command works only on traces that are in the current active layer.

If you want to delete an entire trace at a time, use **Traces/Delete/Delete Single Net** instead.

Return to Presentation

Note This command will not delete associated vias. These vias should be deleted using **Traces/Delete/Delete Single Via** or **Traces/Delete/Delete Unused Vias**.

9.4.3.2 Traces/Delete/Delete Open trace ends

Use to delete all open trace ends in the design. Use this command to clean up the design after design completion.

9.4.3.3 Traces/Delete/Delete Single net

Use to delete all traces of a single net. Select the net from which all traces should be deleted from the list box or by clicking on a trace-segment or component pin that belongs to the specific net.

9.4.3.4 Traces/Delete/Delete All traces

Use to delete all traces in a specified area on the PCB. You are prompted to choose if you want to delete all traces on the entire board, all traces inside a specified area or all traces outside a specified area.

Note This command will delete all traces on *all layers* within the specified area.

9.4.3.5 Traces/Delete/Delete Unused vias

Use to delete all vias that do not have any trace-segments or copper areas connected to them. Use this command after **Traces/Delete/Delete open trace ends** to clean up the design.

9.4.3.6 Traces/Delete/Delete Single via

Use to delete vias. Simply click on the via to be deleted. You are prompted to confirm the deletion.

9.4.4 Traces/Edit

Use to modify a selection of traces. Displays the Edit Traces screen, where you enter the new trace code or trace type for the selected traces.

For more information, see “Editing Traces” on page 5-2.

Return to Presentation

9.4.5 Traces/Continue

Use to continue a previous trace or arc action (depending on which action was used last) at the exact coordinates where you cancelled the action.

9.4.6 Traces/Arc

Use to construct arc segments in traces.

Locate three points to draw an arc. Click at the point where you wish to anchor the starting point of the arc (point on the arc), then click at the point you want to use to anchor the “origin” of the arc's radius (arc center point) and finally, define the arc's angle.

Press SHIFT+F3 to switch to the new trace.

9.4.7 Traces/Copper Area

Use these commands to place, delete, edit, and update copper areas. Copper areas are filled copper areas on a finished PCB. Pads, vias or traces from a different net may penetrate these copper areas.

For more about copper area commands, see “Using Copper Areas” on page 5-12.

9.4.7.1 Traces/Copper Area/Place

Use to create copper areas in a specified net. Ultiboard automatically creates voids to pads, vias and traces in different nets and adds thermal reliefs (if selected) to pads and vias in the same net.

When you decide to add traces through the copper area after the copper area has been placed, use **Traces/Copper Area/Update Single Copper area** to create new voids and thermal reliefs in the copper area.

9.4.7.2 Traces/Copper Area/Delete

Use to delete copper areas. Select the copper area to be deleted by clicking on it. Cancel the deletion by pressing ESC.

This command works only on copper areas that are in the current active layer.

9.4.7.3 Traces/Copper Area/Edit

Use to change the settings or the outline of an existing copper area. Select the copper area by clicking over it. Only copper areas that are in the current active layer can be selected.

9.4.7.4 Traces/Copper Area/Update All

Use to update all copper areas. Use this command after adding traces, vias or pads within copper areas. Ultiboard will recalculate the necessary voids to pads, vias and traces in different nets and thermal reliefs to pads and vias in the same net for each copper area in the design.

This command does not affect the copper area settings or the copper area outline.

9.4.7.5 Traces/Copper Area/Update Single

Use to update a single copper area. Use this command after adding traces, vias or pads within a copper area.

Select the copper area to be updated by clicking on it. Ultiboard will recalculate the necessary voids to pads, vias and traces in different nets and thermal reliefs to pads and vias in the same net for the selected copper area.

This command does not affect the copper area settings or the copper area outline.

This command works only on copper areas that are in the current active layer.

9.4.8 Traces/Layer Push

Use to push a complete trace segment from one active layer to the other. Ultiboard automatically inserts or deletes vias where needed.

Click the trace segment to push to the other layer. Only trace segments placed on the current active layer can be pushed to the other active layer. You can switch the active layers by pressing F2. You can change the active layers in a multi-layer design with the + and - keys on the numeric keypad.

9.4.9 Traces/Highlight Net

Use to highlight all pads (component-pins) that belong to a selected net. Displays the Highlight Net screen, where you select the net to highlight.

Return to Presentation

Ultiboard places a yellow “X” on the appropriate pads.

9.4.10 Traces/Set Active Layers

Use to select layer pairs to work with in manual routing. Displays the Set active layers screen, on which you select the two active layers from a list of all layers in the board.

Press F2 during routing to toggle between these layers while digitizing a trace or by pressing the + and - keys on the numeric keypad. When you start Ultiboard, the default active layers are set to Top and Bottom.

Note If you want to route a single-sided board, choose a double-sided board and simply do not use the second layer.

9.5 Components Menu

9.5.1 Components/Place

Use to create a new component on the design manually. Normally you would import the component from a component list from the Electronics Workbench Schematic Capture program, but this command is often used for non-electrical components such as mounting holes.

For more on this command, see “Editing Components” on page 4-13.

Note If you add components manually rather than importing them from a schematic you must manually assign each pin of that component to a net, using **Netlist/Edit**.

9.5.2 Components/Move

Use to move a selected component. The component you want to move may either be moved by clicking a component on the workspace, by selecting the name of the component from the drop-down list, or by entering the name of a component.

For more on this command, see “Moving Components” on page 4-7.

Return to Presentation

9.5.3 Components/Delete

Use to delete a component. You are prompted to confirm that the component should be deleted. When you delete a component you automatically delete its connections from the nets it belongs to.

To delete more than one component at once, use the **Group/Delete** command, first checking the Groups tab of the **Tools/Options** screen to ensure you have the proper group settings.

Note Once you delete a component you cannot restore it.

For more on this command, see "Deleting Components" on page 4-11.

9.5.4 Components/Edit

Use to edit component settings. Click on the component to be changed, or select it from the the drop-down list. The Edit Component screen appears, with information about the selected component.

For more on this command, see "Editing Components" on page 4-13.

9.5.5 Components/Attributes

Use to assign attributes (keywords or strings) to a component. Select the component from the list.

9.5.6 Components/Text Position

Use to change the "refdes" (reference designator) text position and/or "value" text position of a component. Click to select the text string. The component to which the selected text belongs will be highlighted in the preferred marked color as set in the Layers & Colors tab of the **Tools/Options** screen. Press F2 to rotate the text while it is being moved.

Position the text by clicking the left mouse button or by entering exact coordinates. To use exact coordinates, press the asterisk key (*) on the numeric keypad and use the Specify coordinate screen.

Return to Presentation

9.5.7 Components/Text Size

Use to display the Set global component text size screen, which you can use to change the height, width and thickness of the component text (value and reference designator) for all components in the design. General text elements will not be resized using this command.

9.5.8 Components/Drag

Use to drag (move) selected components while maintaining all possible connections. Autorouter routines are used to drag the traces, so the real-time design rule check is active during the operation. As a result, if dragging or moving a component creates short circuits or clearance errors, the connections that cause the problem are lost.

For more on this command, see “Dragging Components” on page 4-9.

9.5.9 Components/Group Move

Use to move a selected group of component10

s around the design.

For more on this command, see “Moving Components by Group” on page 4-10.

9.5.10 Components/Group Lock

Use to lock the components in a selected group to the board. As soon as you try to move or drag a locked component an error message appears. Although locked components can not be moved or dragged, locked components can be unlocked, deleted or changed.

For more on this command, see “Locking and Unlocking Components” on page 4-11.

9.5.11 Components/Group Unlock

Use to unlock locked components in a selected group.

For more on this command, see “Locking and Unlocking Components” on page 4-11.

9.5.12 Components/Shape

Ultiboard provides a Shape Editor to make creating, editing and copying shapes fast and easy. Start with a predefined shape and make alterations, or start from scratch. To invoke the Shape Editor, use these commands.

For more about using the Shape Editor for working with components, see “Editing Shapes” on page 4-14.

9.5.12.1 Components/Shape/Edit shape

Use to edit shapes in the Ultiboard Shape Editor window. Choose the shape to be edited from the drop-down list, by clicking on it, or by manually entering its name.

9.5.12.2 Components/Shape/Copy shape

Use to copy an existing shape and edit the copied shape in the Shape Editor. This command is especially useful when creating shapes which differ slightly from each other (for example, in the number of pads, pad code, outline etc.).

9.5.12.3 Components/Shape/Delete unused shape

Use to delete shapes from the local library. Select the shape to be deleted from the drop-down list. Only shapes that are not in use in the design can be deleted from the local library.

9.5.12.4 Components/Shape/List shapes

Use to list and preview all shapes in the local library, an external design file, or an external library file.

9.5.13 Components/Find

Use to search for a component by its refdes. Select the component’s refdes from the drop-down list or enter the refdes into the Component field.

For more on this command, see “Finding Components” on page 4-17

Return to Presentation

9.5.14 Components/List

Use to list all the components that are present in the design. Components are listed by refdes and value.

For more on this command, see “Listing Components” on page 4-17.

9.6 Texts Menu

Use the commands in this menu to insert, adjust and delete notations on your design. For more on these commands, see “Working with Text” on page 7-7.

9.6.1 Texts/Place

Use to create a general text element. Displays the Create Text screen.

Press F2 to rotate the text element while it is attached to the mouse. Click at the point where you want to place the text element, or enter exact coordinates using the asterisk key (*) on the numeric keypad. You can place a text element in multiple positions on your design.

For more on this command, see “Working with Text” on page 7-7.

9.6.2 Text/Move

Use to move a selected text element on the active layer. The selected text element is highlighted in the “marked” color set on the Layers & Colors tab of the **Tools/Options** screen.

Press F2 to rotate the text element while it is attached to the mouse. Click at the point where you want to place the text element or enter the exact coordinates using the asterisk key (*) on the numeric keypad.

9.6.3 Texts/Delete

Use to delete the selected text element on the active layer. Choose **Texts/Delete**, then select the text on your design.

The selected text element is highlighted in the “marked” color set on the Layers & Colors tab of the **Tools/Options** screen. You are prompted to confirm the text deletion.

9.6.4 Texts/Edit

Use to change the selected text element on the active layer.

The selected text element is highlighted in the “marked” color set on the Layers & Colors tab of the **Tools/Options** screen. Displays the Change Text screen, which contains the following fields:

Text string	Text string to be displayed on the design.
Text size	Size of the text element. The text height and text width are given in the default unit of measurement. The text thickness is given as a percentage of the text height.
Text layer	Layer where the text element should be placed.
Text rotation	Rotation of the text element. The rotation can be selected from the drop-down list or entered manually.

9.7 Group Menu

9.7.1 Group/Settings

Use to select the objects (components, traces, texts, copper areas) that you want to include in a group copy, group delete, group export or group move.

Displays the Group Options screen.

Changes made to the group settings are automatically stored and become the default settings for new Ultiboard sessions. Each object that is partially positioned inside a selected group is included in the selected group.

9.7.2 Group/Move

Use to move a selected group. Press F2 to rotate the group counter-clock wise around its origin in steps of 90º degrees before placing it.

Return to Presentation

See "Moving Components by Group" on page 4-10.

9.7.3 Group/Delete

Use to delete a selected group or to delete more than one component at a time. Note that once you delete a group you cannot restore it.

You are prompted to confirm that the group should be deleted. When you delete a group you automatically delete its connections.

For more on this command, see "Deleting Components" on page 4-11.

9.7.4 Group/Copy

Use to copy the selected group. Press F2 to rotate the group around its origin in steps of 90º before you paste it. After you paste the group, its components and nets are renumbered.

9.7.5 Group/Continue

Use to repeat a group move, delete or copy. The **Group/Continue** command can only be executed directly after a group move or group copy.

Group Continue after a Group Copy

The most recently defined group will be copied again and can be pasted in a new position.

Group Continue after a Group Move

A selected group is picked up and can be moved to a new position using the mouse.

9.7.6 Group/Undo

Use to undo a group move, copy, delete or continue.

You can only undo the last group operation. The **Group/Undo** command can only be executed immediately after another group command.

9.7.7 Group/Import

Use to import a previous exported group file. Displays the Select Group file screen, from which you select the group file (`.blk`) that you want to import. You can choose to renumber nets and components. Before final placement, the imported group can be rotated counter-clockwise in steps of 90° by pressing the F2 key.

9.7.8 Group/Export

Use to store a selected group in a file.

➢ To store a selected group in a file:

1. Choose Group/Export.
2. Click to select the first corner of the area to be exported.
3. Click to select the second coordinate when prompted.
4. Enter a file name for the group you wish to export. The file you create will have the extension `.blk`.

9.8 Netlist Menu

9.8.1 Netlist/Load

Use to import a netlist from a file. Before reading in a netlist, you must have your components loaded. After you select the netlist file to load, the system prompts you for the default trace code. This trace code will be assigned to all nets that do not have a predefined WIDTH attribute in the netlist file.

All netlist import errors are reported in a file named `import.log`.

See also **File/Import/Netlist**.

9.8.2 Netlist/Create

Use to create a new net or series of new nets. Name the net, add pins and select a trace code for the net.

For more on this command, see “Creating Nets” on page 5-16.

9.8.3 Netlist/Edit

Use to add pins to and delete pins from a given net. Select the net you want to modify from a list or from the workspace of an active design.

For more on this command, see “Editing Nets” on page 5-17.

9.8.4 Netlist/Properties

Use to change the properties of a net. Select an existing net from the screen or from the list of available nets, then change the trace code or name.

For more on this command, see “Net Properties” on page 5-18

9.8.5 Netlist/Powerplanes

Use to define layers as Power/Ground planes or to clear layers that were previously defined as Power/Ground planes.

For more on this command, see “Working with Powerplanes” on page 5-18.

9.8.6 Netlist/Lists

Use to display a list of:

- All current netlist names. This assists you when selecting a net name.
- All unconnected (unassigned) pins. The unconnected pins will be listed by the components’ reference designators.
- All unconnected pins of a specified component.

Return to Presentation

9.8.7 Netlist/Compare Netlist

Use to compare a netlist file on disk with the netlist for your current design. This is helpful in identifying how the nets in your design may have changed since the design was imported from the schematic.

For more on this command, see “Identifying Netlist Changes Since Previous Import” on page 3-19.

9.9 Tools Menu

9.9.1 Tools/Production Class

Use to set the pre-defined production class level for a new board design. You must assign a class to each board.

For details about production classes, see “About Production Classes (Technology Files)” on page 3-1.

9.9.2 Tools/Connectivity check

Use to check if all copper is connected from pin to pin as specified by the netlist of the design. Ultiboard prompts you with the Connectivity Check screen.

The connectivity check can be run for the entire board or just for a specified net. If errors are found, Ultiboard notifies you, and the results are posted to a text file (`.con`).

For more on this command, see “Connectivity Check” on page 7-1.

9.9.3 Tools/Design rule check

Use to invoke an automatic check of the design against the current rule sets. Displays the Design Rule screen.

The design rule check can be run for the entire board or just for a specified net. If errors are found the results are posted to a text file (`.err`). Errors are also displayed graphically by white circles indicating the coordinates where the error occurred.

For more on this command, see “Design Rule Errors” on page 7-2.

9.9.4 Tools/Renumber components

Use to make it easier to locate components when assembling, testing, and servicing the printed circuit boards you design.

For more on this command, see “Renumbering Components” on page 7-7.

9.9.5 Tools/Pin & Gate swap

Use to swap pins and gates to create a more easily routable board. Swapping involves interchanging two assigned nets for electrically equivalent elements, gates or pins. Swapping creates allowable differences in pin assignments that should be backannotated to your schematic.

For more on this command, see “Pin and Gate Swap” on page 7-5.

9.9.6 Tools/Pad stack

Use to enable or disable pads and vias on selected layers. Displays the Pad stack screen, which you can use to change the padstack for all pads and/or all vias for the top layer, bottom layer or all inner layers at once. You can also change the pad stack for a single pad or via by clicking **From Screen** and then clicking on it. If you select a single pad or via you will be able to enable or disable each separate layer (including inner layers) in the design.

9.9.7 Tools/Chamfer corners

Use to add chamfered corners (i.e. corners at an increment of 45º) to all the orthogonal routed traces.

For more on this command, see “Chamfer (Miter) Corners” on page 7-3.

9.9.8 Tools/Board outline

Use these commands to create board shapes.

9.9.8.1 Tools/Board Outline/Edit

Use to launch the Shape Editor to edit the existing board outline.

9.9.8.2 Tools/Board outline/Define by rectangle

Use to draw the board outline as a rectangle. You are prompted to select the two opposite corners of the rectangular board outline. Use the mouse, or type the exact coordinates using the asterisk (*) on the numeric keypad.

For more on this command, see “Defining a Board Outline as a Rectangular Shape” on page 3-5.

9.9.8.3 Tools/Board outline/Define by polygon

Use to draw the board outline as a polygon. Click at the starting position of the board outline, then click at the desired outline vertices. Alternatively, type the exact coordinates using the asterisk (*) on the numeric keypad. When you are finished placing the vertices, click the ESC key to close and place the board outline.

For more on this command, see “Defining a Board Outline as a Polygon” on page 3-6.

9.9.8.4 Tools/Board outline/Import from file

Use to import a board outline from another design file. You are prompted to delete the old board outline. The imported board outline is centered in the workspace.

For more on this command, see “Loading a Board Outline from a File” on page 3-11.

9.9.9 Tools/Reference point

Use to set the reference point (origin) for your design. This is extremely important for relating physical dimensions to PCB layouts, since all measurements are shown relative to the origin. Ultiboard prompts you with the Board Reference Point screen.

Enter the precise coordinate for the new reference point, or set the new reference point using the mouse. The new reference point is highlighted with a special symbol, a circle with a cross.

The mouse grid is related to the exact center point of the total workspace, not the reference point. You can set the reference point of your design to the center point of the workspace by pressing CTRL-Q.

9.9.10 Tools/Relative mode

Use to toggle between absolute mode and relative mode methods of measuring the distance between different points on the PCB.

Click to select the temporary origin for the relative mode. In relative mode, the mouse cursor position is displayed relative to the temporary origin in the status bar. The character "R" displayed in the status bar indicates Relative mode.

You can also toggle between the absolute mode and relative mode by pressing CTRL-R. In this case the current mouse pointer position will be used as the temporary origin.

9.9.11 Tools/Info

9.9.11.1 Info/Object Status

Use to show relevant information about selected design objects. Click on a design object to select it.

Possible objects to show information about are:

- pads
- vias
- components
- copper areas
- trace segments.

9.9.11.2 Info/Board Status

Use to show relevant board and trace statistics about the current design.

For more on this command, see "Statistics and Report" on page 8-15.

9.9.11.3 Info/Net Status

Shows relevant trace information about a selected net. Click on a component pin or trace segment to select the net.

9.9.12 Tools/Options

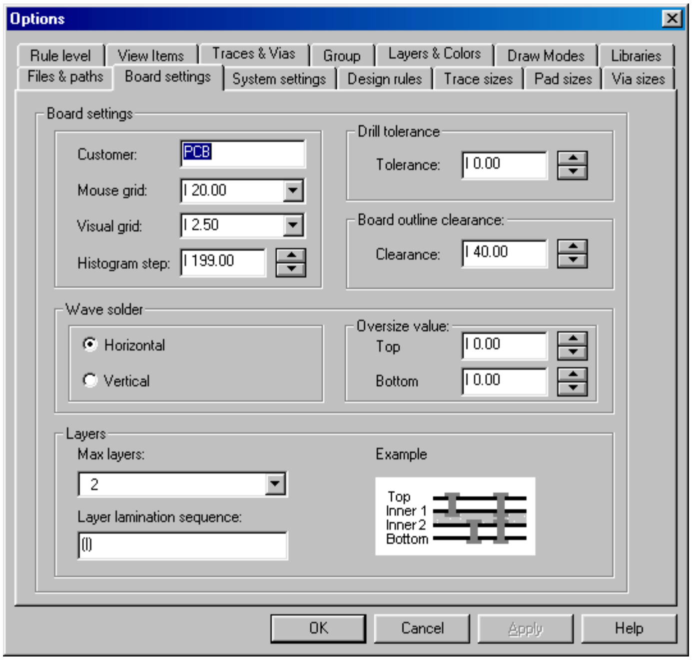
Displays the Options screen, which consists of a series of tabs used to control settings for different aspects of Ultiboard. Any changes made here are automatically stored as default settings and reused in a new Ultiboard session.

9.9.12.1 Files and Paths Tab

Use to set the default location of files used by Ultiboard.

9.9.12.2 Board Settings Tab

Use to specify general information about your board.



Return to Presentation

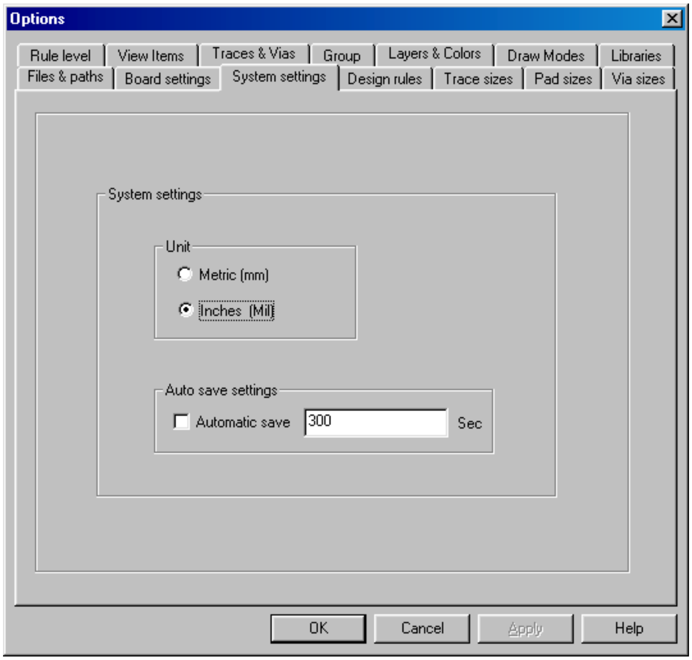
The Board settings options are:

Customer	Customer name (or other name such as project, designer, etc.) which identifies the current design. The customer name is plotted/printed with the output files for this design. Required.
Mouse grid	Minimum grid for design activities. You can use this to have a coarse placement grid for components, then later change it to a finer grid for routing.
Visual grid	Horizontal/vertical spacing of the visual grid, which is displayed as dots on the screen if the **visual grid** option is enabled (on the View items tab).
Wave solder	Specifies the “planned” wave solder direction of the board, horizontal or vertical.
Drill tolerance	The +/- tolerance value applied to drill sizes for the current design. The drill tolerance tells the system about the inaccuracy of the drill position and, therefore, that extra clearance must be provided, and accounted for, around a drilled hole. The drill tolerance is stored with the design.
Board outline clearance	The area from the edge of the PCB, in which you cannot place any copper objects.
Oversize value	The tolerance oversize value for the top and bottom layer. These values are used to increase the allowable pad-to-pad, pad-to-via and via-to-via clearances in the specified wave solder direction.
Max layers	The number of physical trace/plane layers your design can use. Ultiboard is a layer pair-based system so you need to use even numbers of layers, although for special designs you might opt not to place anything on a layer or not to fabricate it.
Layer lamination	The lamination sequence for boards with at least two sequence layer pairs (Max layers >=4). The lamination sequence defines the possible blind and buried vias which can be used in the design. For example: 10 layer board (5 layer pairs)((l + l) + l + (l + l)). This specification means: first handle (drill and metalize) the layer pairs 1 through 5 individually, then laminate 1 and 2 together and 4 and 5 and handle these half-products (drill and metalize). Finally laminate 3 to the half products (drill and metalize). In this example, when routing on layers Inner2 and Inner6, the system must create a through-hole via. When routing on Top and Inner3, a second order buried via is used.

Return to Presentation

9.9.12.3 System Settings Tab

Use to set the units of measurement and auto-save time.



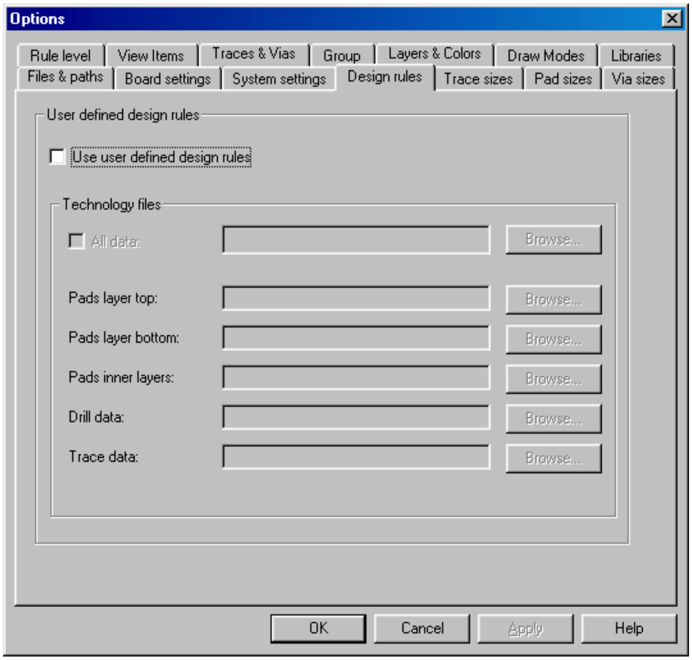
The System settings options are:

Unit	Choose from inch unit of measurement in integer values of 1 mil (1/1000 inch) or metric unit of measurement.
Automatic save	Enables or disables the autosave function. When enabled, the system will automatic save the current design with the extension .bak every specified time interval.
Sec	The time interval for the autosave function.

Return to Presentation

9.9.12.4 Design Rules Tab

Use to specify the location of design files used by Ultiboard.



The Design rules options are:

Class (All data)	The default technology file which is used to import the default class rules for your designs. Class rules include all rule data for the different rule sets: Pads layer top, Pads layer bottom, Pads inner layers, Trace and Drill. If the Class option is disabled you can select different technology files for the different rule sets.
Pads layer top	Select the default technology file which is used to import the default pads layer top rule data for your designs.
Pads layer bottom	Select the default technology file which is used to import the default pads layer bottom rule data for your designs.
Pads inner layers	Select the default technology file which is used to import the default pads inner layers rule data for your designs.
Drill data	Select the default technology file which is used to import the default drill rule data for your designs.

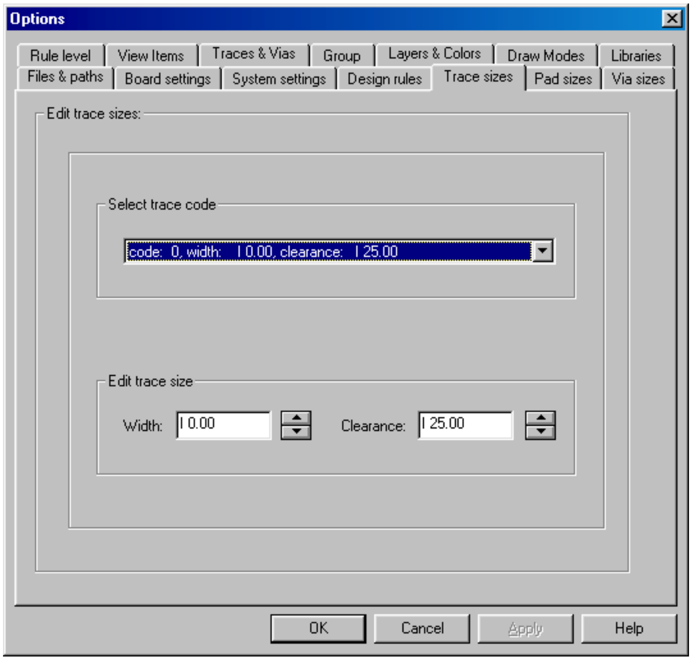
Commands

Return to Presentation

Trace data	Select the default technology file which is used to import the default trace rule data for your designs.

9.9.12.5 Trace Sizes Tab

Use to edit trace size codes.



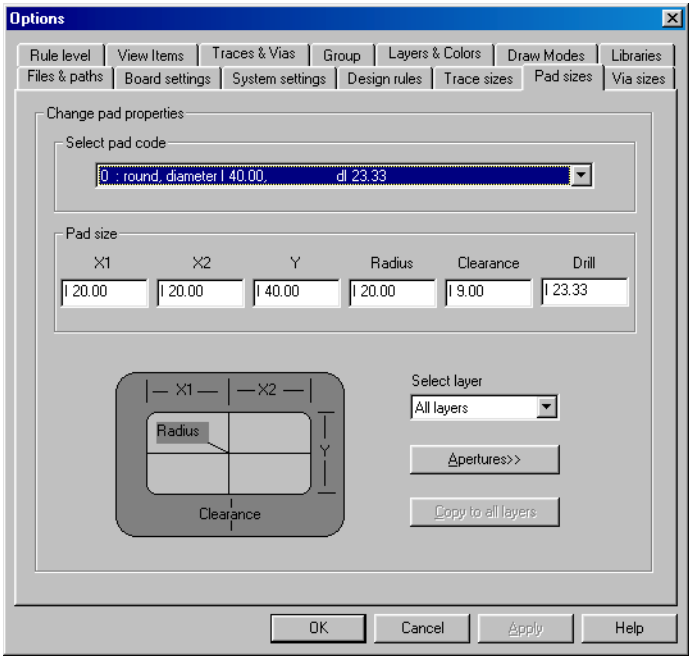
The Trace sizes options are:

Select Trace code	The trace code for which you want to edit the properties.
Width	The new trace width for the selected trace code.
Clearance	The new trace clearance for the selected trace code. The clearance is measured from each edge of the trace to objects on its layer.

Return to Presentation

9.9.12.6 Pad Sizes Tab

Use to edit pad codes.



The Pad sizes options are:

Select pad code	The pad/via code for which you want to edit the properties.
Pad size	The values **X1**, **X2**, **Y** and **Radius** are used to describe the pad shape. The distances **X1** and **X2** start from a common origin in the X axis. The **Y** value is evenly centered (½ **Y**) on each side of this origin point. The **Radius** describes the corner radius of the pad's shape. Pad and via codes have to meet the following conditions: X1 + X2 >= Y, Radius <= Y/2, Radius <= X1, Radius <= X2.
Clearance	The new pad clearance for the selected pad code and the selected layers. The clearance is measured from the edge of the pad/via to objects on its layer.
Drill	The drill size for the selected pad code. If you enter 0 (no drill), this pad code will represent a SMD pad.

Commands

Return to Presentation

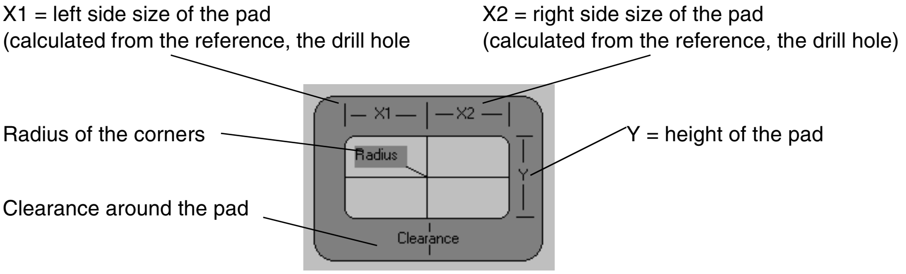
Select layer	Select the layers for which you want to change the properties of the selected pad code: • For SMD pads or when the top size varies from the other layers, use **Top layers.** • For SMD pads or when the bottom size varies from the other layers, use **Bottom layers.** • For the pads on all the inner layers, use **Inner layers.** • For feed-through pads, use **All Layers**.
Apertures	Pre-set the apertures for the selected pad code by entering a D-code for the horizontal and vertical pads and for the horizontal and vertical terminal breaks. If you choose "0", the apertures are selected when running the Postprocessor program. Only use this option for non-standard apertures where you wish to bypass regular Postprocessing. **Note:** You may not use D-codes which are used to pre-set apertures for pad and via codes to generate the Gerber files from the Postprocessor.

Note Pad codes 0-100 and 160-240 are used by components in the libraries and should be left at their default settings. Changes to these pad codes may cause inconsistencies. To define your own pads, use pad codes 101 through 159.

Pads fall into the following categories:

- round pads (with or without drill holes)
- square pads (with or without drill holes)
- rectangular pads (with or without drill holes)
- rectangular or square pads with round corners (with or without drill holes).

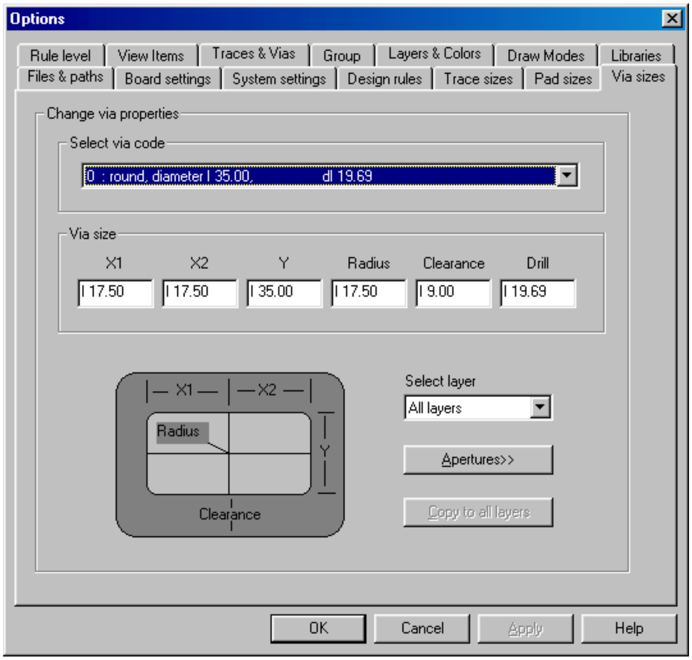
A pad is built from a middle point (also the drilling point), which is the point around which the pad gets rotated.



Return to Presentation

9.9.12.7 Via Sizes Tab

Use to edit via sizes.



The Via sizes options are:

Select via code	The via code for which you want to edit the properties.
Via size	The values **X1**, **X2**, **Y** and **Radius** are used to describe the via shape. The distances **X1** and **X2** start from a common origin in the X axis. The **Y** value is evenly centered (**½ Y**) on each side of this origin point. The **Radius** describes the corner radius of the via's shape. Pad and via codes have to meet the following conditions: X1 + X2 >= Y, Radius <= Y/2, Radius <= X1, Radius <= X2.
Clearance	The new via clearance for the selected via code and the selected layers. The clearance is measured from the edge of the pad/via to objects on its layer.
Drill	The drill size for the selected via code. If you enter 0 (no drill), this via code will represent a SMD pad.
Select layer	The layers for which you want to change the properties of the selected via code: **Top layer**, **Bottom layer**, **Inner layers** or **All layers.**

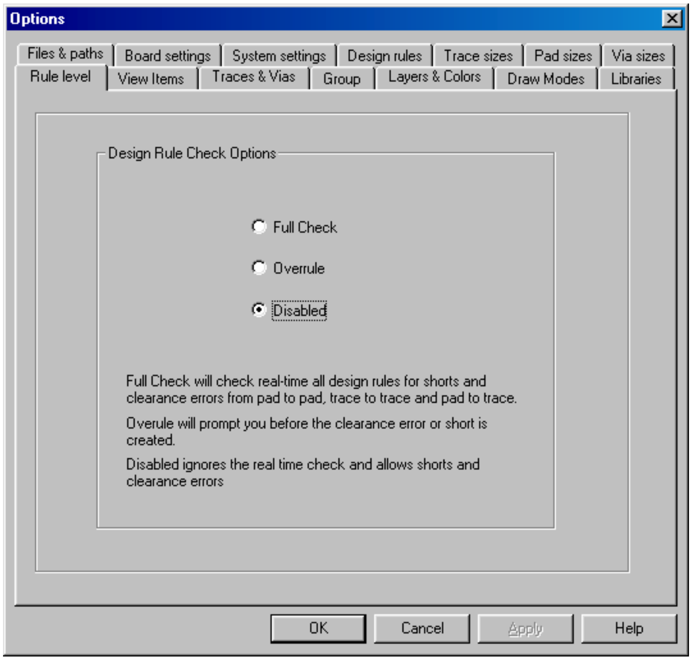
Commands

Return to Presentation

Apertures	Pre-set the apertures for the selected via code by entering a D-code for the horizontal and vertical vias and for the horizontal and vertical terminal breaks. If you choose "0" the apertures are selected when running the Postprocessor program. Only use this option for non-standard apertures where you wish to by-pass regular postprocessing. Note: You may not use D-codes which are used to pre-set apertures for pad and via codes to generate the Gerber files from Postprocessor.

9.9.12.8 Rule Level Tab

Use to set design rule level options.

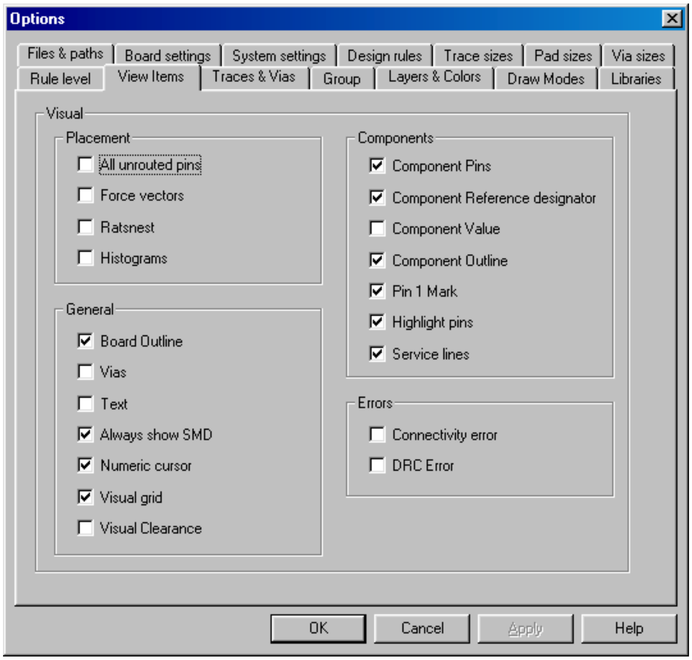


The Rule level options are:

Full check	Checks all design rules for shorts and clearance errors from pad to pad, via to via, trace to trace, pad to trace, pad to via and trace to via. Never allows you to create clearance errors or shorts.
Overrule	Prompts you before the clearance error or short is created.
Disabled	Ignores the real time check and allows shorts and clearance errors.

9.9.12.9 View Items Tab

Use to determine the items that are displayed in the workspace.



The View items options are:

All unrouted pins	Toggles the visibility of unrouted pins on/off. Unrouted pins are pins which do not have nets assigned to them and therefore are left “unrouted”. The visibility of component pins must also be toggled on to view the unrouted pins.
Force vectors	Toggles the visibility of force vectors on/off. Force vectors are used to indicate the direction in which a component could be moved to reduce wire length. The vectors are calculated from the number of connections, their direction and their straight line length. This is known as averaging the vectors.
Ratsnest	Toggles the ratsnest display during component actions on/off. The ratsnest, also called air-lines, shows the shortest pin-to-pin connections for all pins belonging to a common net. It is recalculated each time a component is moved.
Histograms	Toggles the histogram display on/off. The histogram display is used as an aid during placement to visualize the connectivity density.
Visual Clearance	Toggles the visibility of pad, via, trace, polygon and board outline clearance on/off.

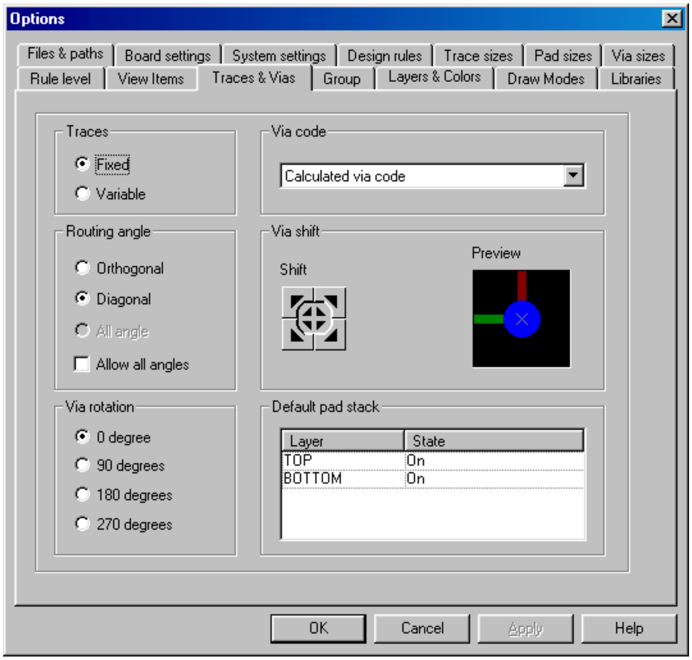
Return to Presentation

Component Pins	Toggles the visibility of component pins (pads) on/off.
Component Reference Designator	Toggles the visibility of the component reference designator (refdes) on/off.
Component Value	Toggles the visibility of the component value on/off.
Pin 1 Mark	Toggles the visibility of the components' pin 1 mark on/off.
Highlight pins	Causes the pins in a net to be highlighted with a large "X" when you start or continue a route on that net.
Service lines	Toggles the ratsnest for the display of a net during routing of that net on or off.
Board Outline	Toggles the visibility of the board outline on/off.
Vias	Toggles the visibility of vias on/off.
Text	Toggles the visibility of text elements on/off.
Always show SMD	When enabled, displays SMD pads independent of the View Component Pins option.
Numeric Cursor	Toggles the mouse pointer position, displayed in the status bar, on/off.
Visual grid	Toggles the visual grid on/off. The visual grid spacing must be set in the Board Settings tab.
Connectivity errors	Toggles the visibility of connectivity errors on/off. If disabled, no errors are displayed, although errors may exist.
DRC Error	Toggles the visibility of design rule errors on/off. If disabled, no errors are displayed, although errors may exist.

Return to Presentation

9.9.12.10 Traces & Vias Tab

Use to specify default trace and vias, and pad stacks.



The Traces & Vias options are:

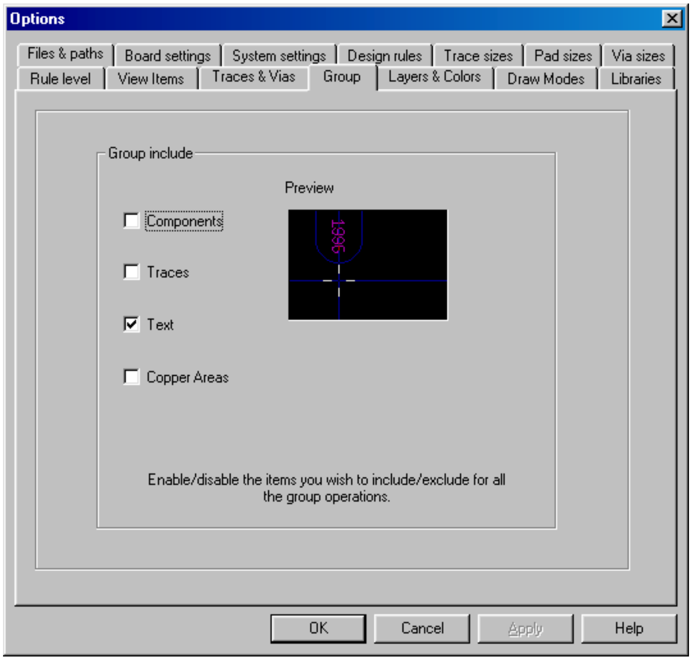
Traces	The default trace type: Fixed or Variable. Fixed traces are not affected by the autorouter; variables traces are.
Via code	The current via code to use during routing. The code represents the via shape assigned to the number. During routing the via code can be changed by pressing SHIFT-F5.
Routing angle	The default routing mode: Orthogonal, 45 degree or Allow all angles. During routing you can toggle between the allowed modes using the space bar.
Shift	The current via shift. Vias can be moved off the main routing path into one of four quadrant locations at an adjacent grid intersection. Via shift is only allowed on a grid smaller than 1/20 inch. During routing the via shift can be changed by pressing CTRL-F5.
Preview	Preview of the specified via shift.
Via rotation	Sets the current via rotation to 0, 90, 180 or 270 degrees.

Return to Presentation

Default pad stack	The default pad stack for vias. A layer is toggled on/off by double-clicking on it. These settings control whether the via will be through-hole, blind or buried.

9.9.12.11 Group Tab

Use to specify the items a group is to include. Items selected on this tab are used by group commands.



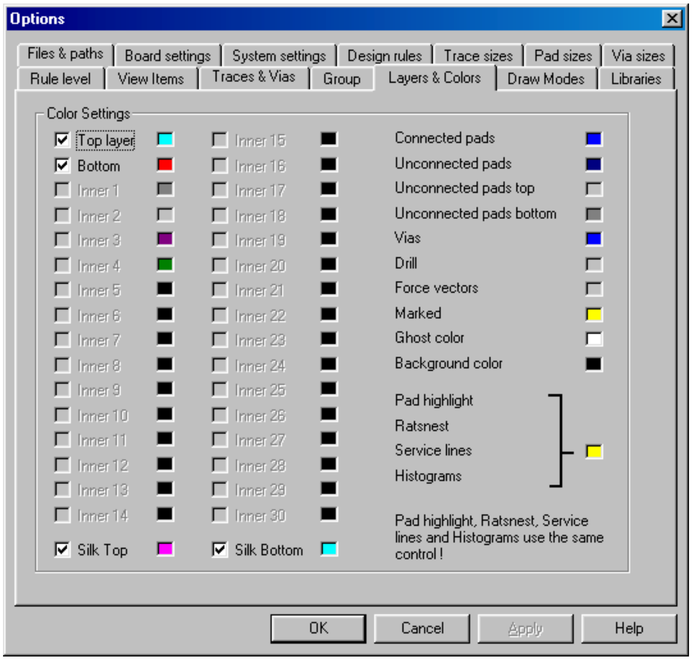
The Group options are:

Preview	Previews the enabled objects in the group that are included in any group actions.
Components	Includes or ignores components during a group action.
Traces	Includes or ignores trace-segments during a group action.
Text	Includes or ignores text elements during a group action.
Copper Area	Includes or ignores copper areas during a group action.

Return to Presentation

9.9.12.12 Layers & Colors Tab

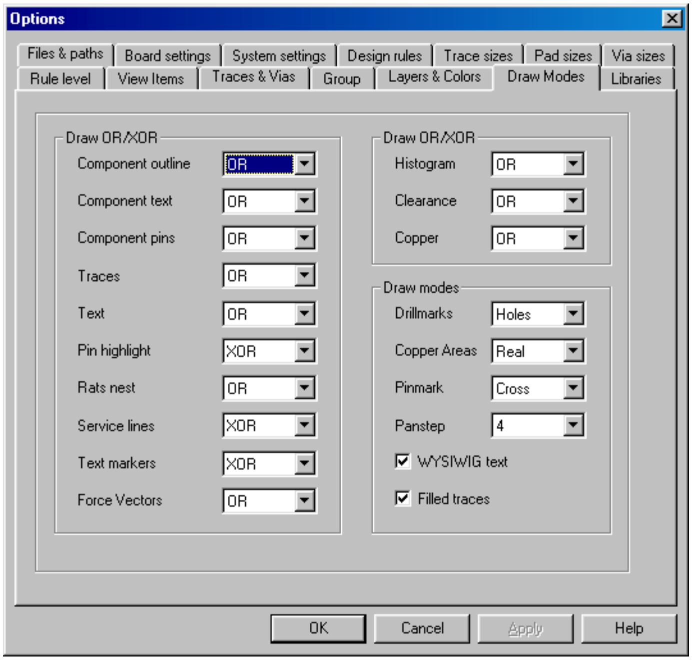
Use to set the colors for all major elements of the on-screen display of your design. These are: layers, pads, vias, silk top and bottom, drill, force vectors, marked, ghost color, background color, pad highlight, ratsnest, service lines, and histograms.



Return to Presentation

9.9.12.13 Draw Modes Tab

Used to set draw modes.



The Draw modes options are:

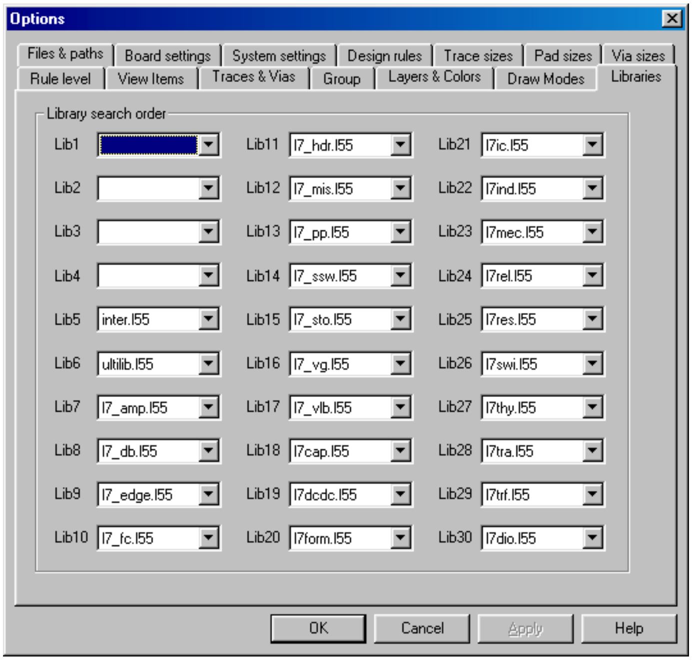
Draw OR/XOR	Draw mode for the different elements available, choosing OR/XOR for each element.
Drillmarks	Determines how drilled component pins and vias are displayed on the screen. Possible settings are: Cross, Hole or None. The visibility of component pins must also be toggled on to display the drill marks.
Polygons	Determines how polygons are displayed on the screen. Possible settings are: Real, Minimized or None.
Pinmark	Determines how each pin in a net is marked during manual routing of that net. Possible settings are: Cross or Square.
Panstep	The pan step (2,...,10). During the design process you can pan the current workspace in steps equal to the selected pan step by holding down the SHIFT key while moving the mouse cursor in the desired direction towards the edge of the workspace.

Return to Presentation

WYSIWYG text	Determines how component texts (value and refdes) are displayed on the screen. When enabled, all component texts are displayed exactly the same as they are plotted/printed. Disable this option to make all text elements readable. The non-component text elements will always be displayed as WYSIWYG text.
Filled Traces	Determines whether to display traces as solid lines or two parallel lines.

9.9.12.14 Libraries Tab

Use to set library sequence order. These libraries are searched automatically for a shape by name, in the sequence presented. (Lib1 will be searched first, Lib2 next, etc.) Choose a library from the drop-down list for each field. Once a shape (name) is found the search process will stop. If a shape exists in more then one library the first shape found is loaded.



Return to Presentation

9.10 Autoroute Menu

9.10.1 Autoroute/Settings

Use to change/set the routing parameters used by the autorouters. These settings affect all the available autorouters. Set these parameters in the Autorouter Parameters screen.

For more on this command, see "Setting Routing Layers and Directions" on page 6-2.

9.10.2 Autoroute/Single Pass

For more on these commands, see "The Single Pass Autorouter" on page 6-1.

9.10.2.1 Autoroute/Single Pass/Route All

Use to start the single pass autorouter. It attempts to route all unrouted nets in the design using the specified autorouter options. The autorouter can be interrupted by pressing the CTRL-BREAK keys.

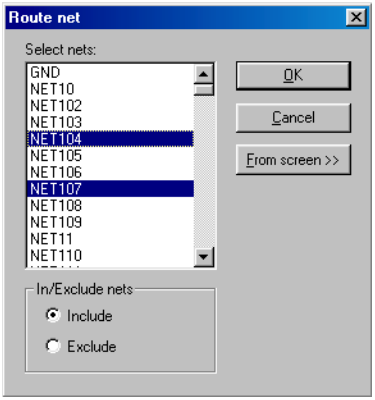
For more on this command, see "Route all" on page 6-8.

9.10.2.2 Autoroute/Single Pass/Route Net

Use to start the single pass autorouter. It attempts to route the selected nets using the specified autorouter options. The autorouter can be interrupted by pressing the CTRL-BREAK keys.

Return to Presentation

Specify the nets to be routed in the Route Net screen.



With the **Include Nets** option enabled all selected nets will be routed by the single pass autorouter. With the **Exclude Nets** option enabled all nets except the selected nets will be routed by the single pass autorouter.

For more on this command, see “Route net” on page 6-8.

9.10.2.3 Autoroute/Single Pass/Route Window

Use to start the single pass autorouter. It attempts to route the workspace you specify. This command routes all nets with connections within the workspace to the nearest connection outside the workspace. The autorouter can be interrupted by pressing the CTRL-BREAK keys.

Specify the boundaries of the workspace using the mouse or by direct coordinate entry using the asterisk key on the numeric keypad.

9.10.2.4 Autoroute/Single Pass/Route Component

Use to start the single pass autorouter. It attempts to route all unrouted nets of the specified component. Select the component to be routed from the drop-down list or by using the mouse. The autorouter can be interrupted by pressing the CTRL-BREAK keys.

9.10.2.5 Autoroute/Single Pass/Route Bus

Use to start the single pass autorouter. It attempts to route all unrouted nets within a distance of 10 mils, only horizontal or vertical. The autorouter can be interrupted by pressing the CTRL-BREAK keys.

For more on this command, see “Route bus” on page 6-9.

9.10.2.6 Autoroute/Single Pass/Options

Use to set the routing parameters used by the single pass autorouter.

For more on this command, see “Single Pass Routing Options” on page 6-3.

9.10.3 Autoroute/Ultiroute Rip-up and Retry Autorouter

For more about these commands, see “The Rip-up and Retry Autorouter” on page 6-10.

To Ultiroute Rip-up and Retry Autorouter

Starts the rip-up and retry autorouter with the current design. The design will be converted by the Ultiroute rip-up and retry translator, then will be automatically loaded within the rip-up and retry autorouter. The rip-up and retry autorouter options should be set within the rip-up and retry program.

From Ultiroute Rip-up and Retry Autorouter

Converts the design routed with the rip-up and retry autorouter (`.rtf`) to Ultiboard. This conversion is done by the Ultiroute rip-up and retry translator.

9.10.4 Autoroute/Specctra

9.10.4.1 Autoroute/Specctra/To Specctra

Starts the Ultiboard to Specctra conversion utility. The conversion progress is shown in a screen. As soon as the conversion process is finished, click **OK**. The Specctra Preferences screen is displayed.

Select the mode in which you want Specctra to start:

Smart Route	Specctra immediately starts routing the converted design in the Smart Route mode with the specified settings for Wire grid and Via grid.

Return to Presentation

User Settings	Specctra starts routing the converted design in the Smart Route mode with the specified settings for Wire grid, Via grid and Routing. First the selected Preprocessing options are performed and then, when the design is 100% completed, the selected Postprocessing options are performed.
No Routing/Placement mode	Specctra is started in the interactive routing mode. You can manually start the desired action, for example Init-Place (Auto placement) or Smart Route.

Note To avoid rounding errors, all units in Specctra will be displayed as standard Ultiboard database units (1nm'). So, 1 unit in Specctra should be considered as 1 Ultiboard unit (5/6 mil).

9.10.4.2 Autoroute/Specctra/From Specctra

Starts the Specctra to Ultiboard conversion utility.

The conversion progress is shown in a screen. As soon as the conversion process is finished, click **OK**. The new placement and wires are saved in your Ultiboard design.

9.10.5 Autoroute/Ultiroute

Starts the Ultiroute autorouter with the current design. The current design is automatically loaded within the Ultiroute autorouter. Autorouter options should be set in the Ultiroute program.

When Ultiroute is closed and the design has been changed by the router, the system asks to import the routing and placing results.

9.11 Help Menu

9.11.1 Help/Help Topics

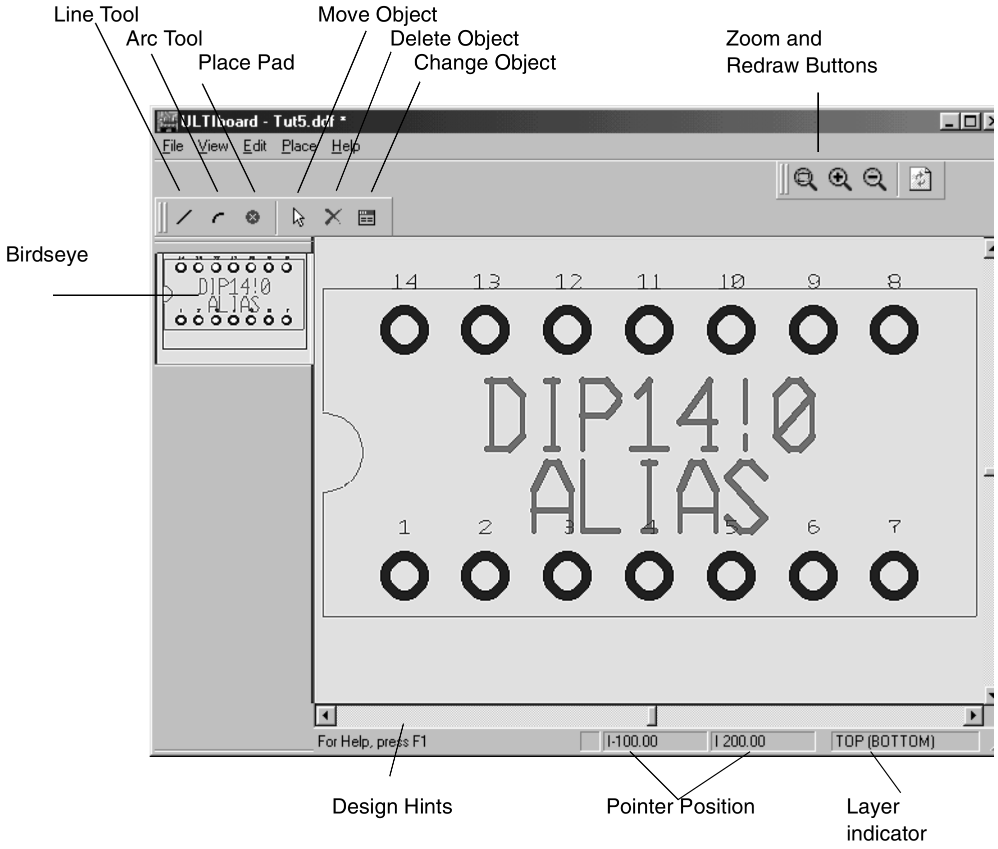
Displays the Ultiboard help file. You can use the index to type in a keyword or refer to the Table of Contents.

9.11.2 About Ultiboard

Return to Presentation

Use this command to display the copyright notice and version number of your copy of Ultiboard.

9.12 The Shape Editor

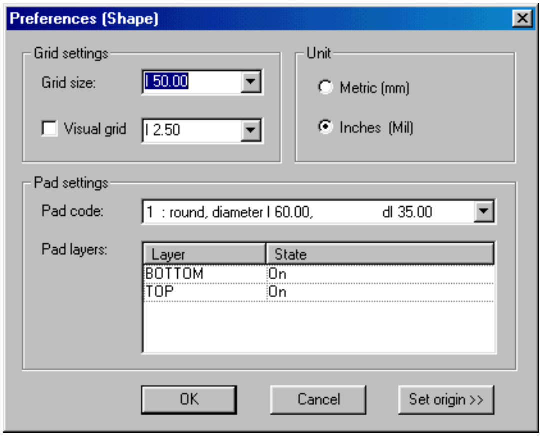


Return to Presentation

9.12.1 Shape Editor File Menu

9.12.1.1 Shape Editor/Preferences

Displays the Shape Preferences screen.



Set the grid sizes and reference point for building the shape. The dimensions of shapes need to be precise. You can get these dimensions from a databook, but the units might be different from what you were using when placing components and traces. You can change the unit of measurement in this screen.

You may also want to change or set the origin, which affects how relative coordinate values are displayed.

Grid Settings	Sets the mouse grid and the visual grid size for the Shape Editor. Select values from the drop-down list or enter them manually. The visual grid can be toggled on or off.
Unit	Set the default unit of measurement used in the Shape Editor. Choose from inch unit of measurement in integer values of 1 mil (1/1000 inch) or a metric unit of measurement.

Return to Presentation

Pad Settings	Set the pad code and the pad layers. The selected pad code and pad layers are used as the default when placing pads. Double-click on a specific layer to toggle the pad on or off for this layer. You can only toggle the pad layers on or off for the layers which are available in your design. The other pad layers are automatically toggled on. When you place pads, you select the *type* of pads to place. You can also place a number of pads automatically. For example, a DIP20 shape has 2 rows of identical, equally spaced pads. To automatically place a row of 10 pads, enter the starting pad number (1), and the final pad number “up to” (10). Then specify how far each subsequent pad should be placed from the last. A positive X value means that the next value will be placed X to the right of the previous pad. A negative X value means that a pad will be placed X to the left of the previous part.
Set Origin	Sets the shape origin (shape reference point). Click the desired position in the Shape Editor workspace to select the new shape origin.

9.12.1.2 Shape Editor/Exit

Use to close the Shape Editor and return to Ultiboard.

9.12.2 Shape Editor View Menu

9.12.2.1 Standard

Use to display and hide the Standard toolbar, which includes buttons for some of the most common commands in Ultiboard’s Shape Editor.

9.12.2.2 Birdseye

Use to toggle the birdseye view on or off.

9.12.2.3 Status Bar

Use to display and hide the status bar.

Return to Presentation

9.12.2.4 Info Object

Use to show relevant information about selected shape objects. Click on the shape object to select it. Possible objects to display information about are:

- shape pads
- shape name
- shape alias
- shape line vertices
- shape arcs.

9.12.2.5 Zoom Full (F7)

Use to adjust the view of the workspace to include the entire design.

9.12.2.6 Zoom In (F8)

Use to reduce the view of the workspace, using the mouse cursor as the center position. Provides a closer look at the current design.

9.12.2.7 Zoom Out (F9)

Use to increase the view of the workspace, using the mouse cursor as the center position.

9.12.2.8 Zoom Window

Use to select a new workspace. Click the two points of a rectangle that will encompass the area to become the new workspace.

Use this command when you want precise control over the workspace.

9.12.2.9 Redraw Screen

Use to redraw the current workspace.

Return to Presentation

9.12.3 Shape Editor Edit Menu

9.12.3.1 Move

Use to move selected shape objects. Click on the shape object to select it. The object you select is highlighted or marked with a cross. Press F2 while moving the object to rotate it. Click at a position to temporarily anchor it. Press ESC to finally place the shape object.

Possible objects to move are:

- shape pads
- shape name
- shape alias
- shape line vertices
- shape arcs.

9.12.3.2 Delete

Use to delete selected shape objects. Click on the shape object to select it. The selected object is highlighted or marked with a cross. You are prompted to cancel or confirm the deletion.

Possible objects to delete are:

- shape pads
- shape line vertices
- shape arcs.

9.12.3.3 Edit

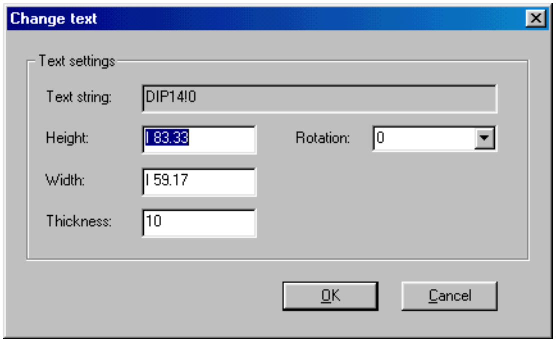
Use to change the properties of shape texts (name or alias) or shape pads. The selected object is highlighted or marked with a cross in the workspace.

- Change the text values for Height, Width, Thickness and Rotation. Thickness must be entered as a percentage of Height. The text string itself can not be changed.
- Change the pad number/name, pad code, pad rotation and pad layers.

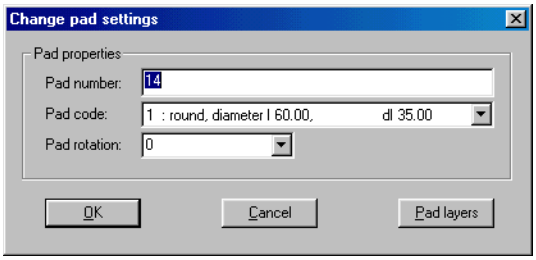
Click on the shape object to select it and display the Change text screen or Change pad settings screen. The current information about the selected object appears in the screen.

Return to Presentation

The Change text screen:



The Change pad settings screen:



9.12.4 Shape Editor Place Menu

9.12.4.1 Line

Use to draw line segments in the shape. Click at the start of a line to anchor the first vertex at the start of the line. Click to create successive anchor points along the line until the line is defined. Every time you anchor a point and move the mouse cursor, a rubber band line is displayed to show you the path of the next segment. The line is drawn when the next segment is anchored. Press ESC to cancel the **Line** command.

Return to Presentation

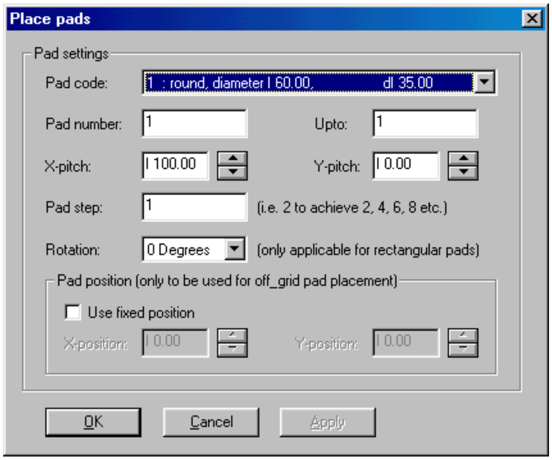
9.12.4.2 Arc

Use to place arcs in the shape.

Draw three points to locate an arc. Click a starting point to anchor the arc (point on the arc), then click to anchor the "origin" of the arc's radius (arc center point) and finally define the arc's angle.

9.12.4.3 Pads

Use to place pads in your shape. Displays the Place Pads screen.



Choose the pad code and rotation for the pads you want to place. The default pad code is set to the code defined in the shape preferences. For more information on pads, see "Setting Pad Sizes" on page 3-14.

To place a single pad enter the pad number/name of the pad to be placed in the pad number box and make sure the **Upto** box contains the same pad number/name. Enable the **Use fixed position** option to place the pad at the specified (X,Y) position (for off grid placement).

9.12.4.4 Attributes

Use to add or remove self-defined shape attributes.

Return to Presentation

Appendix A
Glossary

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.

Birdseye

The small window at the top left of the screen that lets you select the portion of your layout that should appear in the workspace.

Blind Via

A via that connects the top or bottom layer with any inner layer.

Block

A group of items to be moved together. You specify what types of items can be included in a block using the Group tab of the **Tools/Options** screen. When selecting a block, even items that are only partially within the selection are included.

Buried Via

A via that connects inner layers.

Chamfer Corners

Corners at an increment of 45º on the trace routes.

Drop-down list

List of components or shapes that appears when you want to modify a component or shape.

Return to Presentation

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.

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 Mode

Distances are calculated based on the board's reference point.

Orthogonal Routing

Routing performed at 90° degrees only on pairs of board layers.

Pad Stack

The connections between layers of the board.

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 **Tools/Reference Point**. Represented in the viewing window as .

Return to Presentation

Relative Mode

Distances are calculated based on the distance between the last selected coordinates and the current mouse pointer coordinates.

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. Runs as a separate interface. Slower than the single pass autorouter, but will achieve higher completion rates on very dense boards.

SMD Pad

A pad without a drill hole (drill diameter property in the pad code set to 0).

Shape Editor

Screen used to edit shapes. Appears when you modify a shape.

Single Pass Autorouter

Based on the Lee Algorithm. Gives full control over the routing methods (orthogonal, all directions, via reduction), wire sort (short connections first, long connections first), vias (no vias, reduce after initial), and options (hugging, backup time). Does not generate the most optimal design. Faster than the rip-up and retry autorouter.

Thermal Relief

Area around a pin where no copper appears, but which is crossed by copper lines to make connections.

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.

Trace Toolbox

Display of options on the left of the window. Used to speed up access to common trace functions.

Return to Presentation

Unit of Measurement

Affects status bar display. To set, either use the System Settings tab of the **Tools/Options** screen or use one of the following:

- ALT-I (to change to inches)
- ALT-M (to change to millimeters).

Visual Grid

Displays grid lines behind your circuit, for reference purposes.

Workspace

The part of the screen where your design layout appears.

Appendix B
File Types and Extensions

File Type	Extension	Description
Aperture files	.apr	Used to generate Gerber output.
Block files	.blk	Contain a group of components to be imported into another file.
Component files	.plc	Store component information. Automatically created when you export from Ultiboard.
Component Placement files	.cmp	Describe each component by reference designator, alias (value), component shape, rotation and location.
Design file	.ddf	Include design properties (options).
Difference file	.dif	Store results of netlist comparison.
Drill Setting files	.drs	Contain the drill settings.
Error files	.err	Record design rule check errors.
Excellon files	.do	Contain instructions for the Excellon drill.
External Data files	.edf	Contain instructions for autorouters.
Extracted Netlist files	.ntl	Store netlist information. Created when you export the netlist from Ultiboard.
Gerber/Postscript output	.G#	The # represents the contents of the file. For example, .G0 contains pads and traces of the top layer, .G1 contains pads and traces of the bottom layer, reflected, .G2 contains the soldermask, .G3 contains the silkscreen.
Group files	.grp	Specify what layers are to be included in the output.

Return to Presentation

Imported component files	.pk0	Created when you load a netlist. Used for backannotation.
Imported netlist files	.nt0	Created when you load a netlist. Used for backannotation.
Netlist files	.net	Store netlist information. Automatically created when you export from Ultiboard.
Pen Definition files	.def	Contain instructions for the pen plotter.
Penplotter output	.P#	The # represents the contents of the file. For example, .P0 contains pads and traces of the top layer, .P1 contains pads and traces of the bottom layer, reflected, .P2 contains the soldermask, .P3 contains the silkscreen.
Report files	.rep	Contain information for the board manufacturer (drills used, all apertures). Should be sent with Gerber and drill files.
Setting files	.dat	Contain a list of the items to be plotted.
Technology files	.tec	Contain design rules.
Text files	.con	Contain errors from the connectivity check.

Return to Presentation

Appendix C
Keystroke Commands

F1	• Display context-sensitive help about the selected menu command, dialog box or window. Position the cursor over the item you want help on and press F1.
F2	• Rotate a text string while it is still attached to the mouse. • Select the other active layer, while working with polygons and trace segments. • Rotate a block counter-clock-wise around its origin in steps of 90 degrees before you paste it in move block mode.
F5	Change the trace code when creating or working with traces in unspecified nets (manually placing traces).
F7	Zoom to full screen view.
F8	Zoom in on the current design view.
F9	Zoom out from the current design view.
F10	Pan the view of a design to the mouse pointer position.
ESC	• Cancel the current action or function. • Initiate the move command on the Traces menu while you are in the new trace or continue trace mode. • Leave move trace mode. • Close and place a polygon after placing the vertices (with the new polygon command) • Place a shape object after moving it in the Shape Editor.
ALT-F4	Exit from Ultiboard. You are prompted to save any open documents with unsaved changes.
ALT-I	Select inch units of measurement in integer units of 1 mil (1/1000 inch).
ALT-M	Select metric units of measurement, with 2 place accuracy (0.01mm).
Shift-mouse	Pan the current viewing window in steps equal to the selected pan step.
Shift-F1	Display context-sensitive help about an item. Available for toolbar buttons, dialog box options, tools in the toolbox, and other parts of the window.

Return to Presentation

Shift-F3	Switch to the New Arc command when using the Traces/New command, and to the Traces/New command when using the Arc command.
Shift-F5	Change the current via code during routing.
Ctrl-F5	Change the via shift during routing.
Ctrl-Break	Interrupt the autoroute.
Ctrl-N	Open a new file.
Ctrl-O	Open an existing file.
Ctrl-P	Print a design.
Ctrl-	Set the reference point of your design to the exact centre point of the total work area.
Ctrl-R	Toggle Relative mode on/off.
Ctrl-S	Save the current design in a file.
t	Toggle Top layer on/off.
b	Toggle Bottom layer on/off.
1….9	Toggle Inner layer 1...9 on/off.
Arrow keys	Move the mouse cursor in the desired direction in steps equal to the mouse grid.
Numeric keypad + and -	Change the active layers in a multi-layer design.
Space bar	Toggle between orthogonal routing, 45 degree routing and all angle routing when creating a new trace.
Right-click	• Leave new trace mode. You will automatically enter move trace mode. • Leave move trace mode. • Open move trace mode while you are in continue trace mode. • Close and place a polygon after placing the vertices (with the new polygon command). • Release and place components while in new component or move component mode. • Release and place a text element in move text or new text mode. • Click in the birdseye window at the point you want to use as the centre of the current viewing window.

Return to Presentation

Appendix D
Extended Ultiboard Libraries

D.1 About the Ultiboard Libraries

Ultiboard includes Standard Libraries containing approximately 450 shapes, as well as Extended Libraries providing approximately 3250 additional shapes. You can also create your own shape libraries.

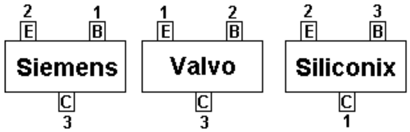
This appendix provides illustrations of every shape included in the Ultiboard Libraries.

D.2 Conventions

For compatibility between the symbol pin on the schematic and the pad of the physical housing on the PCB, pin names, rather than pin numbers, are used for the following shapes:

- Electrolytic capacitors: +, -
- Diodes: A, K
- Transistors: E, B, C
- Bridge rectifiers: AC1, AC2, +, -
- DIN Connectors: 1Ann, 1Bnn, 1Cnn

Example:



Shapes that have mounting holes use the pin names B01, B02.

Appendices

Return to Presentation

D.3 Rules

Padcodes 0 - 100 and 160 - 239 are reserved for Ultiboard shapes.

Padcodes 101 - 159 are reserved for user-defined shapes.

If you have defined shapes of your own, using rules of your own, you might need to change the padcodes of these shapes in your library source files to use an equivalent padcode from the extended Ultiboard rules.

Index

Return to Presentation

Return to Presentation

D

Return to Presentation

Return to Presentation

Return to Presentation

Return to Presentation

Q

R

Return to Presentation

S

Return to Presentation

T

Return to Presentation

U

V

Return to Presentation