Advanced PCB

The Printed Circuit Board Design System for Windows

Includes

Advanced PCB Editor

Advanced PCB Library Editor

Features

16 signal layers, 4 internal power planes, intelligent copper pours, gridless manual routing, autorouting, auto component placement and Gerber and NC drill output.

Now with EDA/Client Server technology; providing a fully customizable user environment, macros, a built in text editor, MS Mail and full support for complementary EDA Servers.

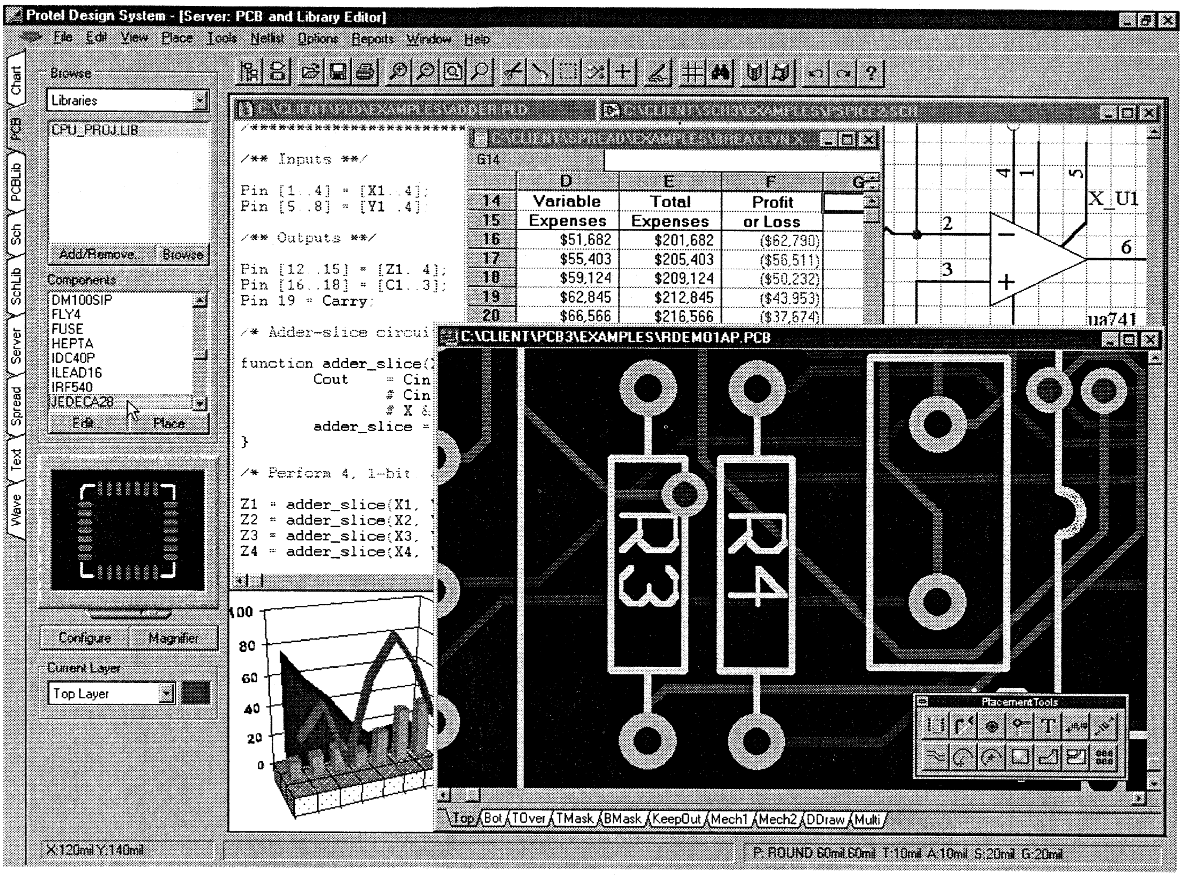
Protel Technology

FIRST IN WINDOWS EDA

Printed by Star Printery Pty Ltd

Contents

Introduction



This section provides an overview of the Protel design environment, including many of the tools, features, key concepts and terminology used in Advanced PCB. The *Advanced PCB User Guide* is intended to provide the information you need to get up and running with the system and to learn to use the basic features required to layout and route a printed circuit board, perform design rule checks, generate PCB fabrication files and print out design documentation.

System Overview

The Protel Design System has been created for today's preferred PC environment. This system combines the natural advantages of Microsoft® Windows™ with a number of sophisticated tools to yield a sophisticated printed circuit board design system, with powerful links to other Electronic Design Automation Tools. Advanced PCB now runs as a server in the Protel EDA/Client™ Server environment. Advanced PCB includes two editors: the PCB Editor and the PCB Library Editor.

EDA/Client

EDA/Client has been developed to fulfill the ongoing demands of the electronics engineering industry. These demands include a need for a standard user environment, high integration of EDA tools with ease of expansion, and total support for networked distribution of EDA resources. A customizable environment, where the user can create and modify menus, toolbars and shortcut keys as well as create and run macros is now expected by users of EDA tools.

All this, without being restricted to a single EDA vendor, is what the EDA/Client Server environment offers. Client provides the standard user environment, and the platform on which you can run any number of EDA Servers. These servers can include a schematic capture tool, a PCB design tool, simulation tools, PLD/FPGA design tools, customized documentation tools, in fact any tool which conforms to the open architecture of EDA/Client. These tools can all be running concurrently within EDA/Client.

PCB Editor

The PCB Editor is the primary document editor in Advanced PCB. This editor allows you to create, edit and verify the PCB design. From the PCB Editor you can also generate the output files required to manufacture the printed circuit board.

PCB Library Editor

The PCB Library Editor is the second document editor in Advanced PCB. It is used to create, edit and manage libraries of component footprints. The PCB Library Editor shares many common features with the PCB Editor, plus specialized tools and features for library management tasks.

Advanced PCB Features

Advanced PCB is a complete PCB layout environment with many attractive features for productive design work. You can use the system for stand-alone manual board layout. Or, when combined with a schematic capture package, Advanced PCB becomes the backbone of a fully-automated, integrated, end-to-end design system.

Protel's EDA/Client and PCB design server work within the standard Windows user interface. If you are experienced with other Windows applications, you already know how to start and exit Protel's PCB design system, navigate menus and dialog boxes and use Explorer or the File Manager to locate and organize your documents.

In short, the system looks and runs like other Windows applications. However, you should be aware that this PCB design system differs from other applications in a number of fundamental ways due to the special requirements of PCB layout.

PCB layout differs from other drawing-oriented tasks in its requirement for extreme precision. As a result, Protel's PCB design system is more of a "placing" environment than a freehand "drawing" environment.

Another key difference is connectivity – the system's ability to recognize connections between track segments, tracks and component pads, and so on. For example, the system allows you to move a component without breaking its track-to-pad connections. You will be using connectivity on several levels as you design with the system.

PCB layouts are generated and displayed as a set of layers which correspond to the individual "phototools" used to fabricate the board, such as the top and bottom signal layers or the silkscreen overlay layer. Some operations, such as manual track placement, are layer dependent – you must first select the layer, then place the track.

Whether your design is a simple single-sided PCB, or a multi-layer board with multiple internal planes, you will be able to layout every item exactly as it will be fabricated in Advanced PCB.

32-bit PCB Design Database

Advanced PCB uses a 32-bit design database and can generate through-hole and SMD designs of up to sixteen signal layers, plus four mid-layer power planes. Four mechanical drawing layers allow you to generate fabrication and assembly drawings for your design. Boards can be as big as 100 inches (254 cm) by 100 inches. Placement accuracy on the 0.001 mils grid system is ±0.0005 mils.

The switchable metric/imperial grid system allows you to work accurately in both measurement systems and can be toggled on-the-fly as you design.

Advanced PCB Links To Schematic Capture

Full support is included for netlist-based design entry. Loading a schematic netlist allows you to take full advantage of Advanced PCB's on-line connectivity monitoring, comprehensive design rules, auto component placement, manual and autorouting, engineering change order and design rule checking facilities. Special links are provided to Protel's Advanced Schematic server, including bi-directional cross-probing between schematic sheet and board files as well as forward and back-annotation.

Design Rules

Today's electronic designs impose more than simple mechanical and current/voltage requirements on the PCB layout. They can also require that you apply specific conditions to individual nets, components, or regions of the board, as well as considering such issues as crosstalk, reflections and net lengths.

To specify these requirements Advanced PCB incorporates a large set of design rules. These include clearances, object geometry, parallelism, impedance control, routing priority and routing topology. Each rule can be applied to the board, to objects, nets, net classes, from-tos, from-to classes, components, component classes, layers, or user definable regions.

On-Line and Batch Design Rule Checking

The on-line DRC flags design rule violations as you route. The batch DRC allows comprehensive verification of the board layout to user-specified physical and logical design rules.

Automatic Component Placement

The Advanced PCB design system includes a high-performance global auto component placement server. The Global Placer uses an AI-based methodology called simulated annealing. It analyzes the entire design as it places, considering the connection length, the connection density on the board and the alignment of the components, all in accordance with the design rules.

Rip-Up and Retry Maze AutoRouter

Advanced PCB includes a high-performance rip-up and retry maze autorouter. This autorouter is designed to route through hole and SMD designs of "medium" density. It includes a via reducing smoothing pass and a miter pass.

Unbreakable Connectivity

A key feature of Advanced PCB is the way logical and physical (or electrical) connections between the elements in your design are recognized and managed. At all times Advanced PCB monitors the state of the connectivity, adding and removing connection lines as you place and delete tracks.

Sophisticated Gridless Manual Routing

With the ever increasing variety of component packaging technologies it is difficult for today's designer to manually route in a grid-based design environment. To keep pace with the changing demands of manual routing, tracks can be routed gridless in Advanced PCB.

Combine the electrical grid (which snaps objects together) with the avoid obstacle mode and the seven track placement modes (with look-ahead), and you can predictively route tracks to any object without creating a violation.

Flexible Selection

Groups of items can be selected by layer, by physical connectivity or by designating an area of the board. Individual items can be added to or removed from the selection. Advanced PCB also includes a Query Wizard, allowing you to create complex selections of different primitives, using standard query operators such as not equal to, less than, and so on.

Selections can be manipulated using standard Windows Edit menu items like Cut, Copy, Paste or Clear; moved; flipped on either axis or rotated in .001 degree increments.

Powerful Global Editing Options

Attributes can be edited by double-clicking directly on the item to open a dialog box. In Advanced PCB changes made to one object can be globally applied across an entire design using specific conditions to define the targets. For example, when editing tracks you can change the track width, track layer or both the width and layer. These changes can be globally applied to all tracks of the same width and/or layer; tracks which are not the same width and/or layer; all selected tracks; all non-selected tracks. Similar global options are provided for other design objects.

Linear and Circular Array Placement Options

Array placement (Advanced PCB option) allows selections to be placed in circular arrays as well as straight lines. Circular repeats are defined by radius and angular increment. Each repeated item can be rotated around its own axis.

Undo and Redo

Multi-level Undo and Redo processes work for all physical changes to the board layout. The designer can make multiple changes, backtrack using Undo, then reinstate each "Undo" change with the Redo process.

Complete Component and Library Management

Multiple libraries can be opened simultaneously. Open libraries in the PCB Library Editor while working on the board design in the PCB Editor. Over 300 component patterns, including through-hole and SMD footprints are included in the standard PCB design system library. Simultaneous multi-user library access is supported for network installations.

Advanced PCB also includes a powerful component building Wizard. This Wizard will ask a few questions and then build the component footprint for you, from a simple two pin resistor through to a Pin Grid Array with hundreds of pins.

Intelligent Polygon Planes

Solid or lattice polygon planes can be placed on any layer with optional automatic connection to a specified net. The copper "pours" automatically, wrapping around all placed objects, obeying any relevant design rules. Polygon shapes can be defined using line or arc perimeters and vertices can be moved, added or deleted after the polygon is generated. Polygons can be re-poured around new obstacles and you can redefine the polygon parameters each time the polygon is re-poured.

Split Internal Power Planes

Internal power planes can be "split" to be shared between multiple power rails. Split power planes are fully supported by the Design Rule Checker.

Thermal Relief Control

Where pins connect to polygons or power plane layers they can have thermal relief or direct connections. Both the conductor path width and air-gap can be user-defined, with a choice of 2 or 4 entry points.

Pad Stacks and Pad Removal

Advanced PCB multi-layer pads can be assigned independent size and shape attributes for the Top (component side) layer, Mid layers (1–14) and Bottom (solder) side layer. Unconnected multi-layer pads on Mid layers can be automatically removed when printing or plotting artwork.

Blind and Buried Vias

Vias can either pass through the whole board or connect any two layers. Blind and buried vias can be placed by the autorouter or manually specified. Vias use layer colors to indicate which layers are connecting. Blind and buried vias can also be used between any two layers, supporting build-up fabrication technology.

Fractional Arcs

Advanced PCB has an arc placement resolution of .001 degree. Arcs can be placed on any layer with full connectivity checking on signal layer arcs.

Component Rotation

Full rotation of components and their pads, down to .001 degrees. The same angular resolution is available for the rotation of any selection.

Multiple Fonts

Three display fonts (default, Serif and San serif) support vector plotting and photoplotting.

Automatic Photoplot Generation

Fully-automatic Gerber® plot file generation. Fully-automatic aperture file generation. On-line aperture editing. Composite photoplots of multiple layers. Automatically panelized plot files to specified film size and border requirements. Advanced PCB imports and displays generated Gerber files. Advanced PCB also batch loads Gerber files with each plot assigned to the appropriate PCB layer. Embedded aperture support for Gerber 274X format.

Windows Support for Printing and Pen Plotting

Dot matrix and laser printing, pen plotting and PostScript® output are all controlled from common Print options. Any device supported by Windows is available for output. Plots or prints can either be panelized or generated as a composite of multiple layers, with auto-centering on the sheet.

Automatic NC Drill File Generation

NC drill output is generated automatically without the need for user-defined tool files. A report file is generated that lists each tool required, in both metric and imperial units, and the travel distance for each tool. A fast sorting algorithm processes the NC drill output file for efficient drilling.

Editable Drill Drawings

Drill drawings are fully user-editable with optional markers for each hole location, including: coded symbol, alphabetical codes (A, B, C etc.) or the assigned hole size.

Windows Display Options

Protel's PCB design system allows full use of all 24 bit color graphics cards and monitors supported under Windows. On standard graphics adapters such as VGA, dithering can be used to simulate colors beyond the standard 16. Zoom levels support the full 32-bit system resolution (accurate to ± 0.0005 mils).

Multiple File Formats

Advanced PCB can import PCB design files directly from Protel Autotrax, PADS-PCB and PADS2000 (.ASC), PCAD (PDIF 5/6 format) or Tango Series II. The system saves design files in two different formats: an efficient binary format and a text format which can be directly edited.

Import and Export DXF Format Files

Import DXF files (AutoCAD®) and export PCB designs to DXF format files. Multi-layer DXF files are supported.

Export to Hyperlynx Board Simulation Tool

Export PCB designs to the Hyperlynx file format and analyze the layout with the Hyperlynx analysis tool, BoardSim for Windows. This is a Post-layout Signal-Integrity Simulator for Digital PCBs. BoardSim predicts transmission-line effects (like overshoot and ringing) based on the actual board layout.

BoardSim includes: direct data import from Advanced PCB, automatic electromagnetic modeling of complex board traces, a PCB-layout viewer for operation independent of Advanced PCB, a digital oscilloscope window for displaying signal waveforms, a graphical editor for changing the board stackup, comprehensive libraries of device models and is 32-bit Windows & Windows NT compatible.

Support for IPC-D-350

Export PCB designs to the IPC-D-350 format. This format is specifically tailored for fabrication, test and assembly equipment.

ECO System

An Engineering Change Order (ECO) system tracks physical changes made to the board during layout. This system is compatible with the PADS .ECO file format.

Forward and Back Annotation

Advanced PCB updates the design file every time the netlist is re-loaded. This forward annotation allows schematic-level changes to be automatically applied to a partially completed PCB layout. Components in completed layouts can be positionally re-annotated (re-labeled) and the updated designator assignments can be back annotated to the schematic in Protel's Advanced Schematic.

Reports

Advanced PCB will generate the following reports; Bill of Materials (BOM); Back Annotation files; NC Drill and Pick and Place reports for board fabrication and assembly; a Netlist Status report; Engineering Change Order (ECO) reports and other design reports.

Design System Documentation

The documentation for the Protel Design System and Advanced PCB is organized in the following manner:

Advanced PCB User Guide

This *User Guide* has been designed to guide the new user through the many features of Advanced PCB and to simplify the retrieval of specific information once you have a working knowledge of the package.

EDA/Client and Advanced PCB are similar in operation to other Windows applications. Once you have mastered a few Windows basics you will be ready to learn the Protel PCB design system.

This guide provides a general introduction to the Advanced PCB package including some of the key fundamentals of the PCB editor: design objects, placing and routing, design verification, generating fabrication files and printing. It also covers the fundamentals of component footprint libraries and library management using the PCB Library Editor. The emphasis of this guide is on the key concepts needed in order to use the PCB editor and PCB library editor effectively.

Some step-by-step procedures are provided to illustrate key operations. Process descriptions are included in the on-line help system.

This guide includes special terminology that is unique to circuit board design in the Advanced PCB system. For example, words like *via, clearance* and *thermal relief* each have a specific meaning within the Advanced PCB environment. Definitions for these words will be found in the *Glossary*, at the end of this guide. Your Windows documentation includes definitions for special Windows terminology. A comprehensive index follows the glossary, making it easy to search for specific information by topic or key word.

Using this guide

The following conventions are used to identify information needed to perform Protel's PCB design system tasks:

Windows always refers Microsoft Windows version 3.1, Windows 95, Windows NT 1.0 or later versions.

DOS refers to MS-DOS® or PC-DOS™ version 5.0 or later.

This manual generally follows the conventions used in the *Microsoft Windows Users Guide;*

italic	indicates anything to be typed. Always enter the italicized information exactly as it appears. Italics are also used to flag chapter names and other manuals, and to add emphasis to an important point.
CAPITALS	These are used to indicate directory or filename.
SMALL CAPS	These are used to indicate key names, such as ENTER OR ESC.
Initial Caps	These indicate menu items (e.g. File-Open), dialog box names (e.g. Document Options) or option names in a dialog box (Snap To Center). Process launching sequences are hyphenated, File-Open means choose the File menu and select the Open menu item.
SHIFT+ALT	The + sign means: hold down the SHIFT key then click the ALT key.
F1, F2	The comma (,) means press and release the F1 key then press and release the F2 key.
➡	The arrow symbol used to highlight warnings or other special advisory information.

On-line Help

The Help menu provides instant access to on-line information about this version of Advanced PCB.

On-line Manuals

All Protel products now include their manuals as on-line documentation. The Adobe® Acrobat™ Reader is supplied with Advanced PCB, allowing manuals to be read on your PC.

Installation

Assumptions made by this guide

Three assumptions are made about the user in this User Guide:

That the user is familiar with the principles, terminology and symbology of electronic circuit design. Wherever possible, Protel EDA tools and documentation uses standard electronic engineering principles and terminology;

The guide also assumes that you are familiar with Windows icons, menus, windows and using the mouse to make selections. It also assumes a basic understanding about how Windows manages applications (programs and utilities) and documents (data files) to perform routine tasks such as starting applications, opening documents and saving your work. If you are new to Windows, please start with your *Microsoft Windows Users Guide*;

That the user has a basic understanding of Microsoft DOS and its use of directories, file naming conventions, and so on.

System Requirements

Minimum

- Microsoft Windows 3.1 running on an IBM PC or compatible
- 486 processor with numeric co-processor
- 16 MB of RAM
- SVGA display, 16 color (800x600 resolution)
- 16 MB of hard disk space for minimum installation

Recommended

- Pentium processor
- 16 MB of RAM
- SVGA display, 256 color (1024x768 resolution or higher)
- 35 MB of hard disk space for complete installation (27 MB for Advanced PCB components)

What is Supplied With Your Protel Product?

Your Protel package includes the following materials:

- *User Guide* (this book)
- Letter with the required access codes (this may be delivered separately).
- Protel Software Registration Card (unless purchased as an upgrade).
- Install diskettes or CD.

If any of these items are missing from your package, contact your dealer immediately to arrange a replacement.

Installing the Software

Install the Protel Software by selecting the File-Run... menu item in the Windows 3.1 Program Manager or by selecting the Run option in the Start menu in Windows 95. In the Run dialog box, enter the following;

<drive_name>:\setup

Where <drive_name> is the drive you are installing from. This is typically A or B when installing from floppies, or D or E when installing from CD ROM.

Follow the installation instructions from there.

Enabling the Software

After installing your Protel software, it is important to enter the Access Key Codes to enable all the features of the package. Protel Software can be installed and run without entering any Access Key Codes. However, without codes the software is running in demo mode, so design files *cannot* be saved.

Unlocking the Software

Un-locking the various features of the software is done by entering the appropriate Access Key Codes.

Select the Help-About menu item, then press the Set Access Codes button to pop up the Security Locks dialog box. All the features currently available will be listed in the Locks section of the dialog box. If no Access Key Code has been entered for a feature, its door will be locked.

To un-lock a feature, select it and press the Un-Lock button. The Lock dialog box for that feature will pop up. Enter the appropriate Access Key Code for that feature and press the Test button. If the Access Key Code has been entered correctly the Access Rights will change, indicating successful un-locking. Click OK to close the Locks dialog box. When a feature has been successfully un-locked its door icon will appear open.

Continue to select and un-lock each feature for which you have Access Key Codes.

Register Your software

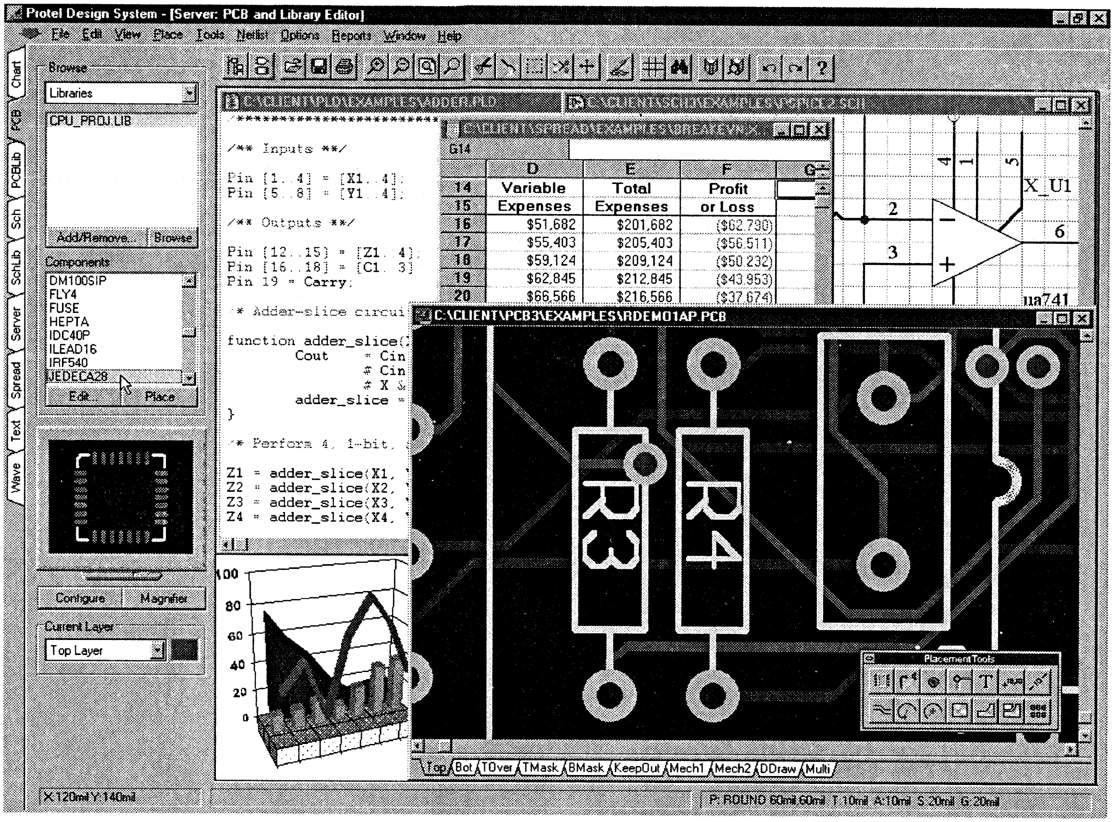
Sign and return the enclosed License Registration Card. By returning this card, you acknowledge that you have accepted the terms of the license. You should also notify Protel (or your local dealer or distributor) if your address changes. Maintaining your registration ensures on-going access to technical support, upgrade notices and other important product information.

➡ Your Protel Design System license number is displayed in the Lock dialog box, where you entered the Access Key Code. Quote this number when making any product inquiries.

Review the README Document

Please review the README document for up-to-date information regarding the current version of Advanced PCB. You have the option to review the README document while installing the software.

A Quick Tour of EDA/Client



The Client / Server Environment

Client / server architecture is an attempt to more sensibly partition the work performed by the various applications on a PC. Traditionally, users source their suite of EDA tools from various vendors. The schematic capture and PCB design tools might come from one vendor, the simulation tools from another, the PLD / FPGA tools from a third and the PCB autorouter tool from a fourth. This means designers must not only be competent in using numerous packages, but be able to switch fluidly back and forth between them, coping with the idiosyncrasies of each design environment. As EDA design tools become more powerful they are also becoming more complex to learn and to operate. This compounds the problem of multiple user environments.

Rather than this vertical approach, where each EDA vendor develops their own proprietary user interface for their tools, an alternative approach is for EDA vendors to partition their tools into the user interface (client) portion, and the "engine" or "services" (server) portion. This structure is known as client / server architecture.

Essentially, the client provides the user environment; the Windows, the menus, keyboard shortcut keys and toolbars, as well as supporting any specialized panels that a server may include.

The server then performs the tasks, for example creating a netlist, simulating a circuit or autorouting a board.

This architecture has many advantages, these include;

- The user only needs to learn one (client) environment.
- From within this one environment numerous EDA Servers can be run.
- Servers can be run remotely from the client. This could be across the network, or even across the Internet.
- Users can easily bring together their own specialized tool set, with servers from different vendors, into a highly integrated environment.
- As servers are accessible across the network, a company can purchase and position servers on their network to suit their needs.
- Servers can be upgraded independently of the client.
- Vendors of EDA products can more easily develop servers as they do not have to develop the user environment.

What is EDA/Client?

EDA/Client is an application, or user environment, developed for the EDA (Electronic Design Automation) industry. Within EDA/Client you able to run your own suite of EDA Servers. These could include a schematic capture server, a netlist server, an FPGA design server and a digital simulation server. Another user might run a PCB design server and an autorouter server on their Client, perhaps also providing a remote site autorouting service with their autorouter server.

Client provides the user resources, that is the menus, keyboard shortcuts, toolbars, status bar and project management panel (Project Manager). These are all user definable. You can easily modify or develop your own menus, keyboard shortcuts, toolbars and macros. You can also readily re-configure the EDA environment, positioning and hiding resources to suit your needs.

Within Client, you can have any number of documents open, each supported by a different server.

As in any application that provides a Multiple Document Interface, a simple click of the mouse takes you from one document to another. Having mastered and tailored this single environment, you are then able to move easily between the various design documents. These could be a schematic and a PCB, or a schematic and an FPGA design. Your focus can remain on the design task being performed in each document, rather than attempting to recall which keyboard shortcut keys to use in this application and where that command lives in another application.

What is an EDA/Client Server?

An EDA Server provides the “services” in the Protel EDA/Client environment. To provide these services, the server will support a set of processes. A process carries out the actual task, such as redrawing the screen, or generating a Bill Of Materials.

A server may include an extensive set of processes, such as a schematic server which needs to be able to place components, wires, buses, generate reports, and so on. Or it may include only a single process, such as a netlist server. There are three kinds of servers supported in EDA/Client.

Document Editor/Viewer Servers

This server kind will include one or more Document Editors/Viewers, allowing you to edit/view documents that this server can create. For example the Schematic Server has two Document Editors, the Schematic Sheet Editor and the Schematic Library Editor. The PCB Placer Server has one Document Viewer.

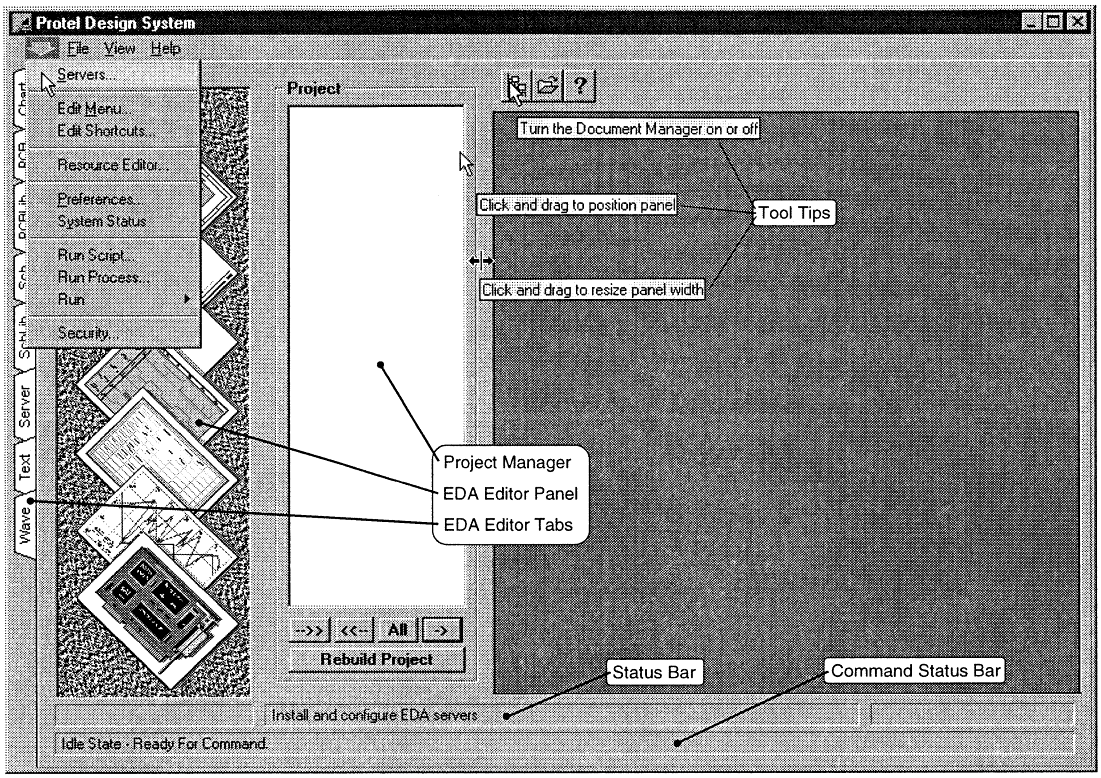
Wizard Servers

This server kind presents a multi-page dialog that guides you through a specific operation. For example, the PCB Component Wizard guides you through the steps of creating a PCB component footprint, performing the repetitious tasks of placing out pads, drawing the outlines, and so on. These servers typically include only one process, used to launch the Wizard.

Utility Servers

Utility servers do not need any user interface, they perform operations on documents that were created by another server. An example of a Utility Server is the Macro Server, which interprets macro scripts created in Text Expert. Another Utility Server is the Netlist Server, which compiles a netlist from schematic documents.

The EDA/Client Environment

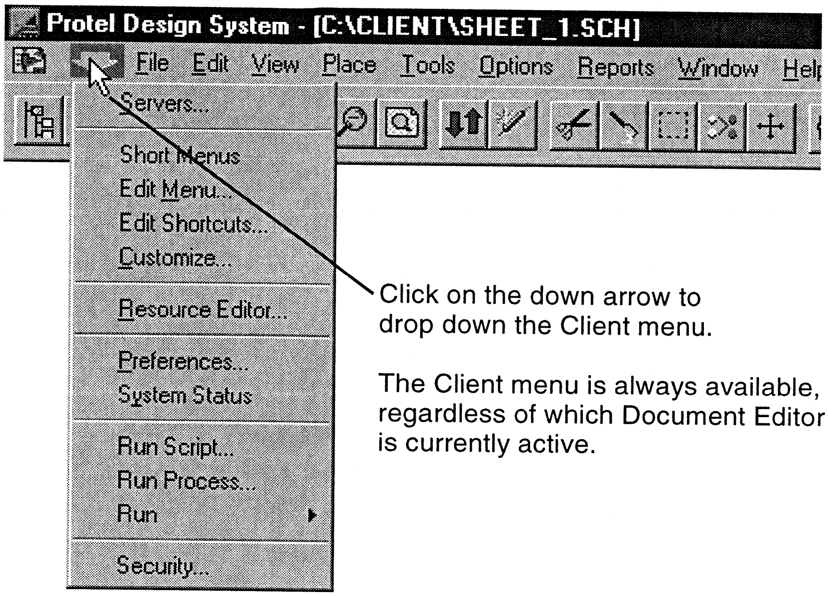


EDA/Client includes a number of features, such as the EDA Editor Tabs, which make it quick to learn and easy use. Following is a brief summary of these features.

Tool Tips

For a tip on what a toolbar button does, or how to move or size a panel, position the cursor and leave it stationary for about a second. A Tool Tip will appear with a brief description of what that button does, or how to re-size the panel, or how to edit the menus.

Client Menu



Click on the down arrow to drop down the Client menu.

The Client menu is always available, regardless of which Document Editor is currently active.

The Client menu is always available on the menu bar, regardless of which Document Editor is currently active. From the Client menu you control EDA/Client. It allows you to do things like install and remove servers, customize and edit resources, set the user preferences and run a macro or another Windows application.

EDA Editor Tabs

The EDA Editor Tabs initially appear down the left of the Client workspace. These allow you to readily identify which Document Editors are available and which is the active Document Editor.

Click on an EDA Editor Tab to make it the active Editor and present the document that is currently active in that Editor. If there are no documents of that type currently open in EDA/Client a new document will be created.

EDA Editor Panel

Client includes an Editor Panel, which Document Editors can then use to provide the user with access to features and information. An example is the Advanced PCB Document Editor Panel, which provides access to libraries and their components and also allows you to browse through the objects placed in the workspace.

Project Manager

The Project Manager, or Document Manager, displays all currently open documents, and any relationship between documents. A document can be made the active document by clicking on it in the Project Manager.

Client Status Bar

The Client Status Bar includes the Status Bar and The Command Status Bar. The Status Bar is dynamic, it provides information about the cursor position in the workspace and the current state of the process being executed.

The Command Status bar reports the current process being executed and gives a description of that process.

Resources

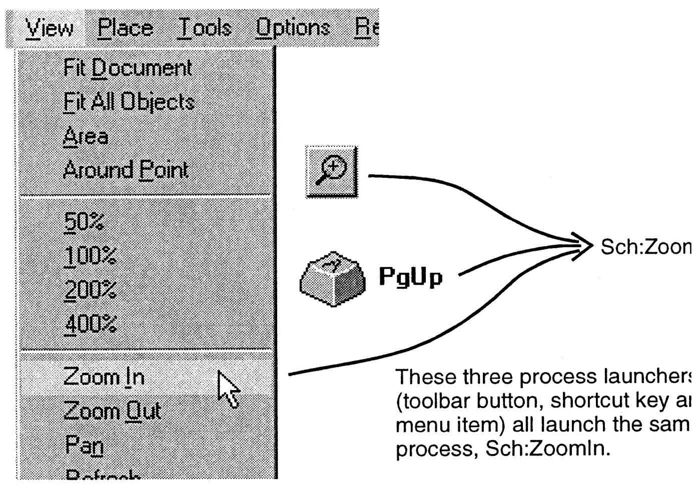
Within the EDA/Client Server environment you can perform operations such as opening and closing documents, editing these documents and generating output based on those documents. This is done via the menus, toolbars and keyboard shortcuts. Menus, toolbars and keyboard shortcut lists are known as *resources* in the EDA/Client Server environment.

Processes and how they are Launched

When you perform any action in the EDA/Client environment, such as opening a file or placing a wire, you invoke a *process*. A process can be thought of as the software executing a sequence of jobs, for example refreshing the screen, zooming in, placing a net label, and so on. Each process is identified by its process identifier. An example is *Client:OpenDocument.*

Processes are provided by EDA/Client and by the installed Servers. Each process is invoked, or *launched,* by a *Process Launcher.*

When you select a menu item or click on a toolbar button, you launch a process. Menu items are process launchers. Toolbar buttons and shortcut keys are also process launchers. Any process launcher can be linked to any process identifier.



Customizing The EDA Workspace

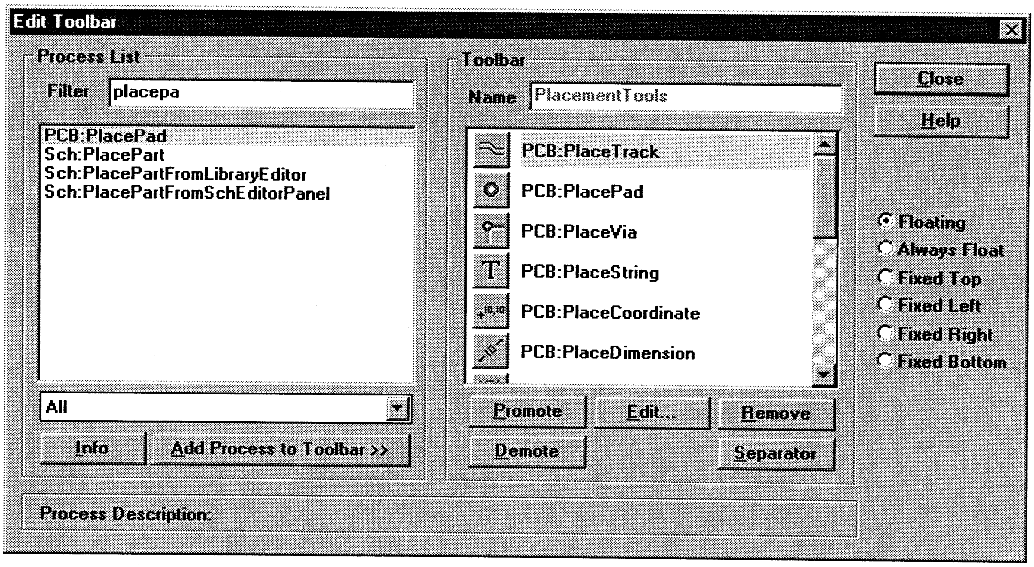
Resources

Resources for each Document Editor can be customized by selecting the Client Menu-Customize menu item. This will pop up the Customize Resources dialog box, allowing you to customize any of the resources currently available in that editor.

Menu and shortcut key resources can also be edited by selecting the appropriate menu item in the Client menu.

Toolbars

The easiest way to edit a toolbar is to double click anywhere in the toolbar. This will pop up the Edit Toolbar dialog box.



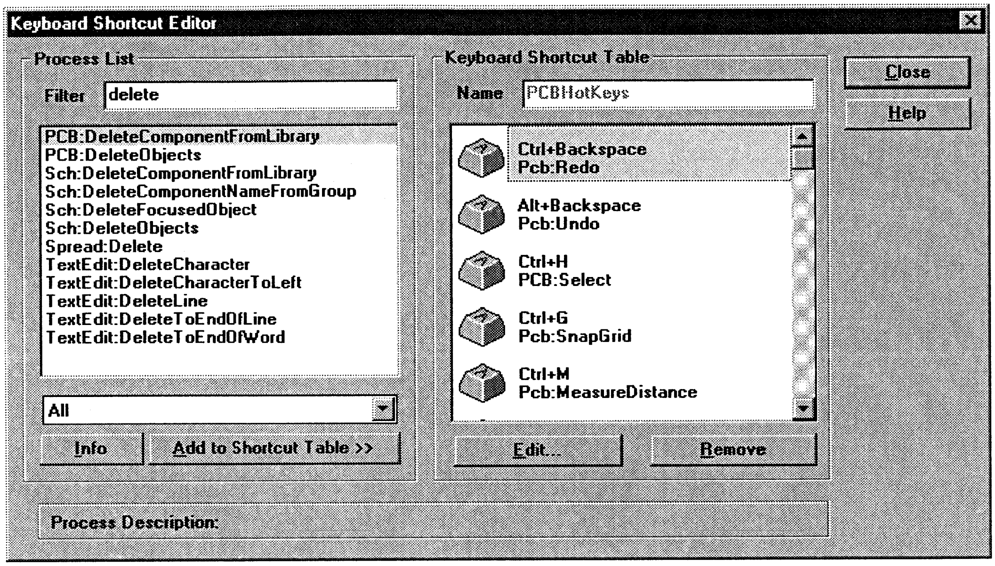
Here you can add, remove and re-position buttons. Select a button and press Edit to pop up the Edit Button dialog box. In this dialog box you can re-assign the button bitmap and the process this button will launch.

If the Position of the toolbar is not fixed, it can be repositioned in the workspace by simply dragging it so that it floats in the document workspace, or dragging it to any of the other edges of the workspace.

If a toolbar is hidden, its display status can be toggled in the Customize Resources dialog box (Client Menu-Customize menu item).

Keyboard Shortcut Keys

In the Client menu, select the Edit Shortcuts menu item. This will pop up the Edit Shortcut Keys dialog box.



Here you can create, edit and remove Shortcut keys. Select a Shortcut key and press Edit to pop up the Edit Shortcut Key dialog box. In this dialog box you can assign the shortcut key(s) and the process this key will launch.

Menus

The easiest way to edit a menu is to double click anywhere in the menu bar. This will pop up the Edit Menu dialog box. Here you can create and remove menu items and change the structure of the menus. The structure of the menu can be changed graphically (simply click and drag) or by selecting the menu item and pressing the appropriate direction button.

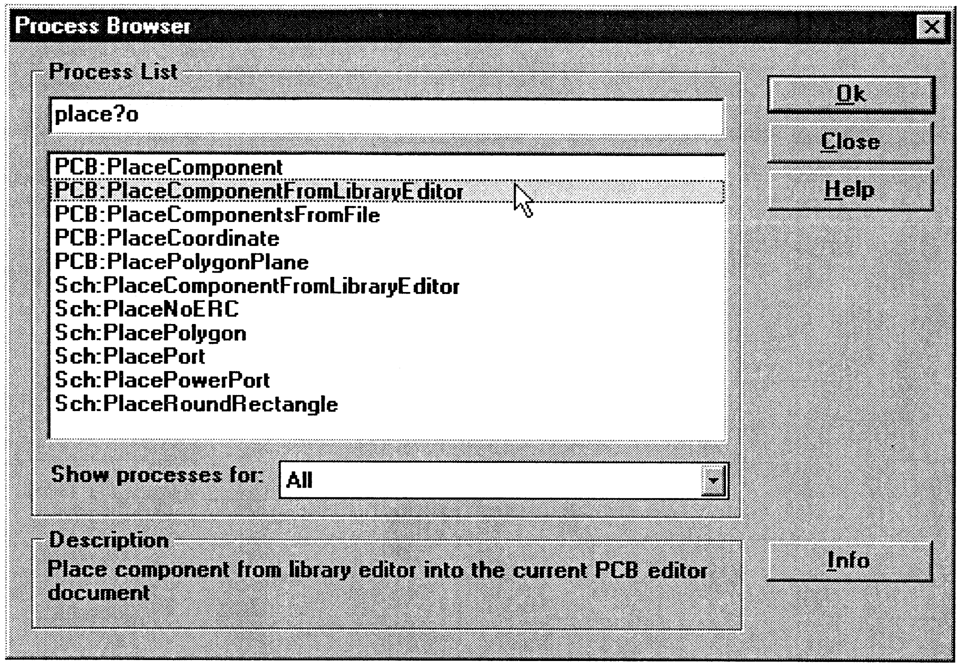
Assigning a Process to a Process Launcher

When you create a new process launcher, such as a toolbar button, you must assign a process for that button to launch. To assign a process you need to know what process you would like to be launched. EDA/Client includes features to assist in locating a particular process and learning what each process does.

➡ To create a new process launcher, edit the resource you wish to customize as previously described.

The Edit Toolbar and Keyboard Shortcut Key Editor dialog boxes will pop up with a list of all currently available processes on the left of the dialog box. There are two ways to filter this list so it displays a narrower range of processes to choose from. Below the list is a drop down list box which allows you to display the processes for a particular server. Use this if you know which server the process is provided by. Above the list of processes is a filter text box. To display only those processes which contain a certain string, type the string into the filter text box. The * (any characters) and ? (any single character) wildcards can also be used.

Click on a process to see a brief description at the bottom of the dialog box. Press the Info button to pop up a complete description of the selected process.



Selecting a process using the Process Browser

The Edit Menu, Edit Button and Edit Keyboard Shortcut dialog boxes have a Browse button, which pops up the Process Browser dialog box. This can also be used to locate a process in the same way as described in the previous paragraph.

➡ Use the CTL+P shortcut keys to pop up the Process Browser dialog box while working in any server.

All the dialog boxes used for editing resources have an Info button. Press this button to pop up a help window with a full description of the process.

Editor panel

The Editor Panel display status can be toggled in either the View menu, or the Preferences dialog box (Client Menu-Preferences).

The Editor Panel can positioned on either side of the workspace. To move it to the other side of the workspace position the cursor within the Panel and click and drag it to the desired location.

Project Manager

The Project Manager (Document Manager) display status can be toggled in either the View menu, or the Preferences dialog box (Client Menu-Preferences).

The Project Manager can positioned on either side of the workspace and its width can be re-sized.

To move it to the other side of the workspace, position the cursor within the frame around the Project Manager window and click and drag it to the desired location.

To alter the width, position the cursor along the right hand edge of the Project Manager. When the cursor changes to a vertical bar with left and right arrows, click and drag to re-size it.

Client Status Bar

The Client Status Bar can be positioned on either the bottom or top of the workspace by dragging it to the desired location. Its display status can be toggled in either the View menu, or the Preferences dialog box (Client Menu-Preferences).

Editor Tabs

The Editor Tabs can be positioned on any side of the workspace by dragging them to the desired location. Their display status can be toggled in the Preferences dialog box (Client Menu-Preferences).

Further Reading

Once you become comfortable working in EDA/Client you will want to create your own resources, such as a specialized toolbar. Refer to the chapters *Understanding Processes* and *Resource Management* to extend your knowledge of resources and processes.

Installing and Starting a Server

Servers are run from within EDA/Client. You must be running EDA/Client to be able to install and start a server. To install and start a server;

- Go to the Client menu and select the Servers menu item. The EDA Servers dialog box will pop up, with a list of all currently installed servers.
- To install a new server, press the Install button. The EDA/Client Server Install dialog box will pop up.
- The installation file for each server has the file extension INS. Locate and select the server you wish to install.
- Clicking OK will install the server. It will have a Status of Not Started. When a server is Not Started it is not occupying any memory. If you do not start it now it will automatically be started the first time you use it. When you close the EDA Servers dialog box an EDA Editor Tab will appear for each Document Editor the Server provides.

Opening a document

One of the powerful features of EDA/Client is that it allows you to have multiple documents open, documents which may have been created by different servers. Having this capability means you must have the server that created a document installed before a document of that type can be opened.

Opening a New Document

- Click on the desired Editor Tab to make that Editor active.
- If the Editor has no documents open, it will automatically open a new empty document for you.
- Otherwise, Select File-New to create a new empty document. The Select EDA Document Type dialog box will pop up. Select the desired document editor and an empty document will open.

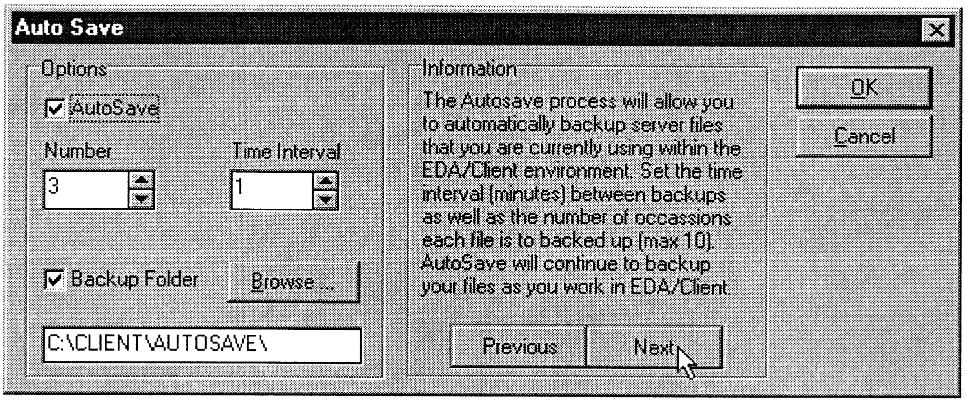
Opening an Existing Document

The desired Document Editor does not need to be active but it must be installed.

- Select File-Open. The Open Document dialog box will be presented.
- In the Document Types section of the dialog box select the desired Document Editor and file Type.
- Locate and select the file you wish to open and click OK.

- The document will be opened and presented in the appropriate Document Editor.
- To open a project, the procedure is the same except for checking the Project check box in the Open Document dialog box.

Automatically Saving Documents



EDA/Client includes an AutoSave Server, which will automatically create backup copies of all currently open documents.

The AutoSave Server can create multiple backups of each open document, with each document being numbered in a cyclic fashion. For example, if your schematic is called MySheet.sch and you have specified three backups, then these will be named MySheet.0ch, MySheet.1ch and MySheet.2ch. Once the AutoSave Server is due to save a forth time it will start the cycle again, calling the next file MySheet.0ch. This means that you must always check the time stamp on the backup files to identify the latest one.

If you enable the Backup Folder option then all the backup files are stored in the one directory, simplifying the management of backup files. If you do not enable this option then the backups for each document are stored in the same directory as each document..

➡ Select the Client menu-Auto Save menu item to set up the AutoSave feature.

➡ These backup files are independent of the backup file that is created each time you select File-Save. This option is enabled in the Client Menu-Preferences dialog.

Text Expert

Text Expert is a text editing server supplied with EDA/Client. Having a document editor for text editing in EDA/Client means there is no need to leave the EDA/Client environment to work with ASCII files. Netlists and reports can be viewed, macro scripts can be written. All general text editing can be performed in Text Expert.

Text Expert includes the normal text editing facilities such as cutting, copying and pasting, search and replace. It also includes a feature known as Syntax Highlighting. Syntax highlighting allows you to highlight different elements in the document based on the syntax, where different word types, symbols and identifiers are assigned unique colors. This feature is an excellent document editing aid, particularly when working with documents with a repetitive, structured nature, such as macro scripts.

To broaden the usefulness of the syntax highlighting feature, Text Expert allows the definition of multiple languages. Syntax highlighting can be uniquely configured for each of these languages.

Within each language, there is a set of six types of syntax identifiers available; reserved word, symbol, string, number, comment and identifier. The user can then define the set of valid words or characters for each of these identifiers and assign a unique color to each type of identifier.

Languages

Text Expert includes a number of pre-defined languages as well as the capability to create new languages. These languages are not the language of a country or culture like French or Mandarin, rather they are a language because each can have their own syntax highlighting definition.

Languages can be created, edited and deleted in the Languages dialog box (Options-Change Language menu item, or the Change Language button on the panel).

Each document can have a language associated with it. The language is selected in the Languages dialog box. A language is associated with a document type by the file extension. To associate a file extension with a language, select the language in the Languages dialog box and press the Edit button. The Edit Syntax dialog box will pop up. In the Associated Files text box enter the file extension. For multiple file extensions, separate each with a comma.

Syntax Highlighting

There are two distinct parts to Syntax Highlighting. The first is editing the syntax, the second is assigning the highlight colors to each type of syntax identifier.

To edit the syntax, select the Options-Change Language menu item. This pops up the Languages dialog box. Select the language you wish to edit the syntax for and press the Edit button. In the Edit Syntax dialog box you define the set of reserved words, how comments and strings are delimited, the valid set of symbols and any file extensions to be associated with this language.

Highlight colors are then assigned in the Text Editor Options dialog box (Options-Preferences menu item).

Document Options

Assigning of colors to each type of syntax identifier is done in the Text Editor Options dialog box. There are also a number of user editing preferences that can be enabled here.

Resetting Defaults

EDA/Client allows the user total freedom customizing the menus, toolbars and shortcut keys. At any stage you can restore the menus, toolbars and shortcut keys back to their original state. To do this, select the Client Menu-Servers menu item. In the EDA Servers dialog box select the server and press the Configure button. The Configure Server dialog box will list all the Document Editors provided by this server. Resetting the defaults for this server will reset the resources for all Document Editors provided by this server. Press the Default button to restore the server's resources back to their defaults.

Macros

EDA/Client is supplied with a Macro server. The macro server supports two macro scripting languages, Client Pascal and Client Basic.

Macros provide a powerful mechanism to enhance productivity when working in EDA/Client. The macro server supports all the processes available in the EDA/Client environment and allows passing of parameters to those processes. Macros can be written to work with any server running in EDA/Client.

Macros can be written to perform anything from a repetitive sequence of processes, through to complex wizards which pop up dialog boxes and respond to user choices. The macro server also supports OLE automation, a feature where operations can be performed in other Windows applications (which support OLE automation).

Client Basic and Client Pascal are interpreted rather than compiled, so macros can be run as soon as they are written. Like all processes in the EDA/Client environment, macros can be launched from any process launcher.

The macro server includes a comprehensive error flagging mechanism. When an error is encountered, the script file is opened in Text Expert, the line in error is displayed and highlighted, and a dialog box pops up with a description of the error condition.

General Topics

Setting Up The PCB Workspace

Opening, Saving and Closing Documents

Working in Advanced PCB

Design Objects

Components and Libraries

Library Editor

Defining the Board

Working With a Netlist

Design Rules

Component Placement

Routing Your Design

Design Verification

Generating Output

Reports

Linking to Advanced Schematic

Setting Up The PCB Workspace

Coordinate System

Coordinates displayed on the left end of the Status Bar indicate the position of the cursor relative to the *current* workspace origin. The coordinates indicate the cursor distance from the *current origin* in mils (thousandths of an inch) or millimeters, depending on the units selected. Advanced PCB allows you to set a new 0,0 coordinate anywhere in the workspace.

The *absolute origin* (the default position of the current origin) is the extreme lower left corner of the workspace.

Setting the Current Origin

Select Edit-Origin-Set to set the current origin at the current cursor position. Once it has been set the Status Bar will display X:0 mils Y:0 mils (or X :0 mm Y:0. mm if a metric snap grid is used) at the current cursor position.

To set the current origin back to the absolute origin (extreme lower left corner of the workspace) choose Edit-Origin-Reset.

Units

Printed circuit boards are manufactured to very close tolerances. Advanced PCB provides an absolute design resolution of ± 0.001 mils (.000001 inch or .00025 mm) – which will provide sufficient precision for any PCB design task. The workspace is 100 inches by 100 inches.

Toggling Units

Advanced PCB supports both imperial (mils) and metric (mm) measurement units and dimensions. The unit of measure changes when you select the View-Toggle Units menu item or press the Q shortcut key. When a metric snap grid is selected, Advanced PCB displays workspace coordinates and other dimensional information in millimeters (indicated mm on the Status Bar). This allows accurate dimensioning of boards in millimeters or creation of new library components with metric pin spacing. Measurement units can be switched at any time.

Grids

Advanced PCB includes three user-definable grid systems. The first is the *snap grid*, which controls the placement of objects in the workspace. The second is the *electrical* *grid*, which defines a "range of attraction" within which electrical objects attract to each other. The third is the *visible grids*, which provide a visual reference as you move around the workspace.

Snap Grid

The snap grid defines an array of points in the workspace which restrict cursor movement and the placement of primitives. When using the mouse to control the cursor, you will notice that the cursor moves freely between snap grid points. If an edit function is being performed, such as placing a component or selecting an object, a cross hair appears. This cross hair will "snap" to the current snap grid. When the cursor keys are used, the cursor always "snaps" to the grid.

➡ Hold SHIFT while pressing a cursor key to make the cursor jump 10 times the current setting of the snap grid.

This grid can be changed at any time in the Options Tab of the Document Options dialog box (Design-Options) or via the set snap grid button on the main toolbar (shortcut; G). Setting the snap grid to 100 mils will mean the cursor can only be on points, 0.0 inch 0.1 inch 0.2 inch, etc. The snap grid can have a value between 0.001–1000 mils (or 0.0025–25.0mm).

The snap grid setting defines where objects can be placed in the workspace. It is vital that the snap grid is set appropriately for good board design. It is typically set to either a multiple of the component pin pitch, or a fraction of it. For example, while placing components with a pin pitch of 100 mil, a snap grid of 50 or 100 mil could be used. To route one track between the pins of these components a snap grid of 25 mil could be used. Working with an appropriate snap grid will assist in orderly component placement and provide the maximum amount of routing channels.

Electrical Grid

To ease the placement of electrical objects, such as tracks and vias, Advanced PCB includes an *electrical grid*. This grid defines a range within which a moving electrical object (such as a track, pad or via) will attract, or snap to, another electrical object.

As you move an electrical object around the workspace and it falls within the electrical grid range of another electrical object, the object you are moving will snap to a hot spot on the fixed object. The electrical grid is configured in the Options Tab of the Document Options dialog box (Design-Options).

➡ The electrical grid overrides the snap grid. This allows you to easily connect to an off grid object.

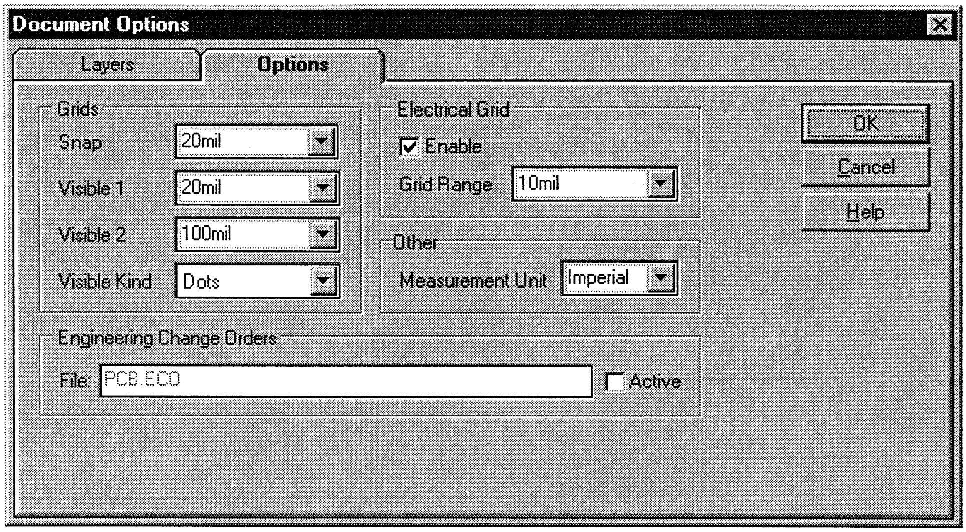
Visible Grids

Two visible grids are provided as a visual reference for placing and moving items. You can set the sizes of these grids independently. For example, you could select one visible grid to be fine and the other coarse, or even separate metric and imperial visible grids.

The visible grid displays a system of coordinate lines (or dots) in the workspace background. The display of the visible grids is constrained by the current zoom level, if you cannot see a visible grid you are either zoomed too far out or zoomed too far in.

Setting the Grids and Units

All grids are set in the Options Tab of the Document Options dialog box. This Tab also allows you to toggle the units and to enable the Engineering Change Order feature.



Set all the grids, toggle the units and enable the ECO feature in the Options Tab

Engineering Change Orders

As you make changes to the PCB, the changes are written to a special text file *filename*.ECO. Changes that are recorded in the ECO file include: adding a new node to a net, deleting a node, renaming a net, adding a component, deleting a component, changing a component footprint pattern, renaming a component, joining two or more nets into a single net or splitting a net into two or more new nets.

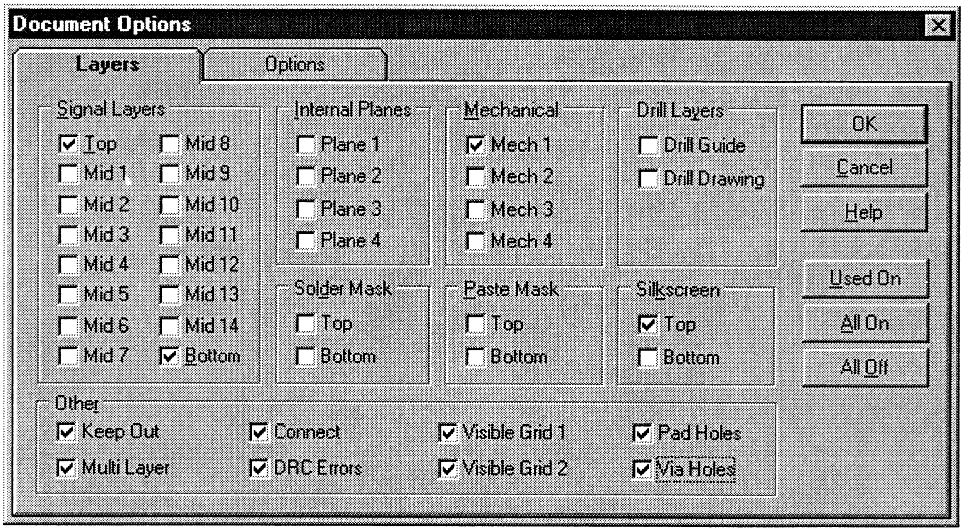
➡ Protel's .ECO file format is fully-compatible with PADS .ECO files.

Layers

Advanced PCB is a "layered" environment. You create your board design by placing objects on these layers. These layers are either "physical" layers, from which the fabrication information is created, or system layers, such as the Connect layer which displays the unrouted connections. Physical layers include the signal layers, internal plane layers, silkscreen, solder mask and paste mask layers. Each of the layers can be assigned a unique identifying color.

This concept of multi-layered design distinguishes Advanced PCB from many other drawing or design applications. Although all the layers in your design can be viewed simultaneously, you will need to select individual layers for some tasks, such as placing a primitive which belongs to a single layer, typically tracks, polygons, fills or free text strings.

➡ Protel's PCB system is a "layered" environment. Certain design operations, such as track placement, are layer dependent.



Enable the required layers in the Layers Tab of the Document Options dialog box.

Before you can access any of these layers, the layer must be turned on. Once a layer is turned on a Layer Tab for that layer will be displayed at the bottom of the workspace.

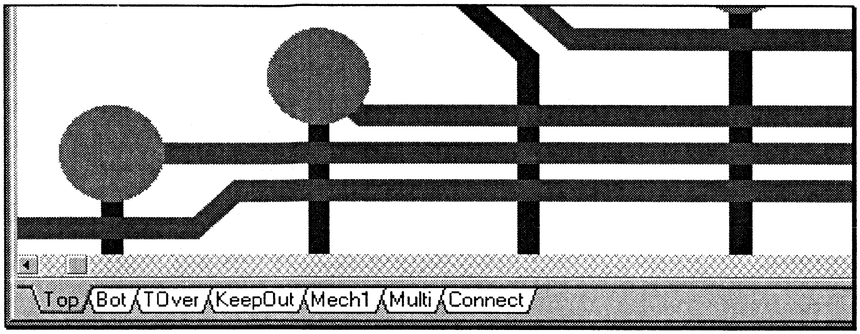
To activate layers:

1. Select the Layers Tab in the Document Options dialog box (Design-Options).

Note how the layers are grouped by layer type. For each of the layers there is a check box next to the layer name, which you can click (LEFT MOUSE) to turn the layer on or off. A tick in the check box indicates that this layer is active.

➡ Any layers that you have activated will be active the next time you open this design in Advanced PCB.

2. Click in the layer check boxes to activate the required layers.
3. Click OK to close the Preferences dialog box.



A Tab for each active layer will appear at the bottom of the document window.

The Current Layer

One workspace layer is "current" at any given time. At the bottom of the workspace there is a Tab for each active layer. The Tab for the current layer will be displayed on the top. Some items, such as tracks, text, fills or single-layer pads are placed on the current layer. Other items, such as components, multi-layer pads and vias can be placed without regard to the current layer. Selection (for moving, deleting, etc.) is layer-independent - you can perform these operations on any primitives without changing the current layer.

➡ Click on a layer tab to make that layer the current layer, or use the + and - keys on the numeric keypad to toggle through all the active layers. The * key on the numeric keypad can be used to toggle through active signal layers.

Signal Layers

There are 16 signal layers which can be used for track placement. Anything placed on these layers will be plotted as solid (copper) areas in PCB artwork. As well as tracks, other primitives (area fills, text, polygon planes, etc.) can also be placed on these layers.

Top

component side signal layer.

Mid Layers

inner signal layers (numbered Mid Layer 1–14).

Bottom

solder side signal layer.

Internal Planes

Four solid copper mid layers (numbered Internal Plane 1–4) are available. Nets can be assigned and automatically connected to these planes and individual component pins can be assigned to internal planes at any time. Special *thermal relief* pad shapes are an option when plotting internal plane artwork. These planes are displayed (and printed/plotted) in the negative for efficiency. In other words, placing any primitive on these layers will create a void in the copper.

Silkscreen Overlay layers

Top Overlay and Bottom Overlay (or "silkscreen") layers are typically used to display component outlines and component text (designator and comment fields that are part of the component description). Advanced PCB library components assign outlines and component text to the Top Overlay by default. If the component is placed or moved to the bottom layer, these items will be displayed on the Bottom Overlay. The designer can include other primitives, such as free text strings, on the Overlay layers.

Mechanical Layers

Four mechanical drawing layers are provided for fabrication and assembly details such as dimensions, alignment targets, annotation or other details. Mechanical layer items can be automatically added to other layers when printing or plotting artwork.

Mask Layers

Solder masks

Top and bottom masks are provided for photo or silkscreen solder masks. These automatically generated layers are used to create masks for wave soldering, usually covering everything except component pins and vias. You can control the expansions for these masks when printing/plotting by including a Solder Mask Expansion rule. Refer to the *Design Rules* chapter for more information on the Solder Mask Expansion rule. This chapter also includes tips on masking all the vias. These layers are plotted in the negative, for efficiency.

Paste masks

Top and bottom masks are provided for photo or silkscreen masks of solder paste locations for boards with surface mount devices (SMDs). You can control the expansions (or contractions) for these masks by defining a Paste Mask Expansion design rule. Refer to the *Design Rules* chapter for further information. These layers are automatically generated and are plotted in the negative, for efficiency.

Drill Layers

Drill Drawing

Coded plots of board hole locations, typically used to create a manufacturing drawing. Individual layer pair plots are provided when blind/buried vias are specified. Symbols are plotted at each hole location. Three symbol styles are available; coded symbol, alphabetical codes (A, B, C etc.) or the assigned size. A table of symbols, metric and imperial hole sizes and hole counts can be included in the plot. Refer to the *Generating Output* chapter for more information.

Drill Guide

Plots of all holes in the layout - sometimes called *pad masters*. Individual layer pair plots are provided when blind/buried vias are specified. These plots include all pads and vias with holes greater than zero (0) size. You can specify the size of the guide marker which locates the hole center in the Drill Plots Tab of the Setup Output Options dialog box (File-Setup Printer, Layers button).

Other Layers

Keep Out

This layer is used to define the regions where components and routes can validly be placed. For example, the board boundary can be defined by placing a rectangular perimeter of tracks and arcs, defining the region within which all components and routes must be placed. "No-go" areas for mechanical objects can be created inside this boundary by blocking off regions with tracks, arcs and fills. Keep outs apply to all copper layers. The basic rule is; components can not be placed over an object on the Keep Out layer and routes can not cross an object on the Keep Out layer.

Multi Layer

Objects placed on the multi layer will appear on all copper layers when output is generated. The multi layer is typically used for through hole pads and vias.

Connect

This option controls the display of the *connection lines*. Advanced PCB creates connection lines on the connection layer wherever it locates part of a net that is unrouted.

DRC Errors

This option controls the display of the DRC errors.

Visible Grids

Controls the display of the two visible grids. Grids can be displayed as either dots or lines (set in the Options Tab).

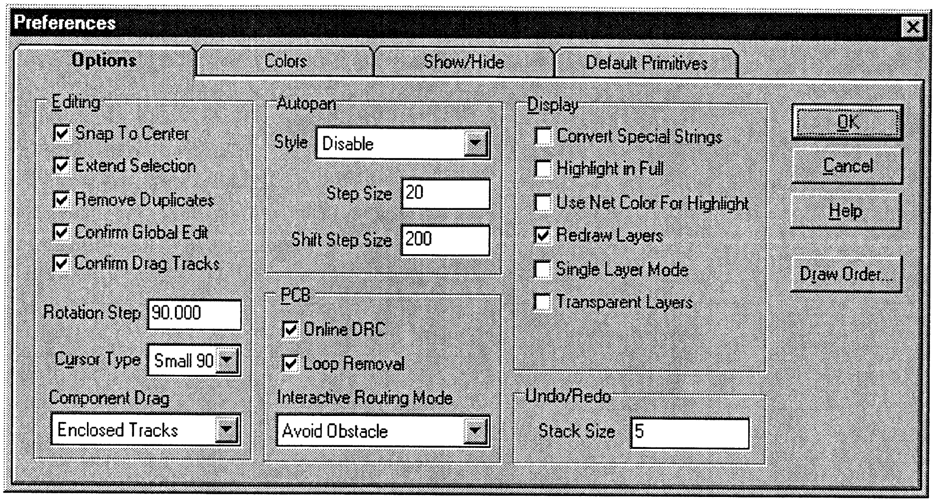
Pad and Via Holes

Controls the display of pad and via holes. To be able to distinguish pads from vias in draft mode, pad holes are outlined in the current Pad Holes color (set in the Colors Tab of the Preferences dialog box).

Workspace Preferences

User definable workspace preferences are set in the Preferences dialog box (select the Tools-Preferences menu item). The Preferences dialog box is divided into four Tabs.

Options Tab



Editing

Snap to Center

Snaps to the center when moving a free pad or via, snaps to the reference point of a component, snaps to the vertex when moving a track segment. The object will be "held" at the cursor location if this option is disabled.

Extend Selection

Selection is cumulative with this option enabled. With it disabled all currently selected objects are de-selected each time a new selection is made.

Remove Duplicates

With this option enabled a special pass is included when data is being prepared for output. This pass checks for and removes duplicate primitives from the output data.

Confirm Global Edit

Pops a dialog reporting the number of objects which will be altered by the global edit and allows you to cancel.

Confirm Drag Tracks

Pops a dialog asking if you wish to drag the tracks attached to the component being moved.

Rotation Step

When an object that can be rotated is floating on the cursor, press the spacebar to rotate it by this amount in an anti-clockwise direction. Hold the shift key whilst pressing the spacebar to rotate it in a clockwise direction.

Cursor Type

Set the cursor to small or large 90 degree cross, or small 45 degree cross.

Component Drag

This option determines how tracks are dealt with when moving a component. The Enclosed Tracks option will move tracks that pass under the component as well as connected tracks. The Connected Tracks option will only drag tracks which connect to the component.

Autopan

Style

If this option is enabled it is possible to auto pan when the cursor includes a cross hair. There are four Autopan modes;

Re-Center - re-centers the display around the location where the cursor touched the Window edge. It also holds the cursor position relative to its location on the board, bringing it back to the center of the display.

Fixed Size Jump - pans across in steps defined by the Step Size. Hold the SHIFT key to pan in steps defined by the Shift Step Size.

Shift Accelerate - Pans across in steps defined by the Step Size. Hold the SHIFT key to accelerate the panning up to the maximum step size, defined by the Shift Step Size.

Shift Decelerate - Pans across in steps defined by the Shift Step Size. Hold the SHIFT key to decelerate the panning down to the minimum step size, defined by the Step Size.

Step Size

Specifies the amount the display should shift each time the cursor touches the Window edge. The numerical value is in the current measurement units.

Shift Step Size

Specifies the amount the display should shift each time the cursor touches the Window edge when you are using one of the Shift auto pan options. The numerical value is in the current measurement units.

PCB

Online DRC

Checks to ensure that the object(s) currently being placed in the workspace do not violate the current design rules. The design rules are defined in the Design Rules dialog box (select the Design-Rules menu item).

Loop Removal

With this option enabled loops that are created during manual routing will be automatically removed.

Interactive Routing Mode

Ignore Obstacle - If you select this option you can place primitives anywhere in the workspace. If the Online DRC feature is enabled clearance violations are flagged immediately.

Avoid Obstacle - If you select this option you can only place primitives where they do not violate any clearance design rules. This feature is particularly useful when routing as it allows you to route hard up against existing objects, without fear of violating any clearance rules. Refer to the chapter *Routing Your Design* for more information about using this feature.

Display

Convert Special Strings

Special strings, such as .LAYER_NAME or .PRINT_DATE, are interpreted on screen as well as when output is generated. Refer to the *Strings* topic in the *Design Objects* chapter for information about using special strings.

Highlight in Full

Completely highlights the selected object in the current selection color. With this disabled the selected object is outlined in the current Selection color.

Use Net Color For Highlight

Highlights the selected net in the net color (assigned in the Change Net dialog box). Use with the Highlight in Full option for better results.

Redraw Layers

Forces a screen redraw as you toggle through layers, with the current layer being redrawn last.

➡ To redraw the current layer only use the ALT+END shortcut keys.

Single Layer Mode

Displays the current layer only. Provides a method of examining what will be output on each layer. If the current layer is a signal layer multi layer objects are also displayed. Use the "+" and "-" keys to toggle through the layers, press END to redraw the screen.

Transparent Layers

Gives layer colors a "transparent" nature by changing the color of an object which overlaps an object on another layer. Allows objects which would otherwise be hidden by an object on the current layer to be readily identified.

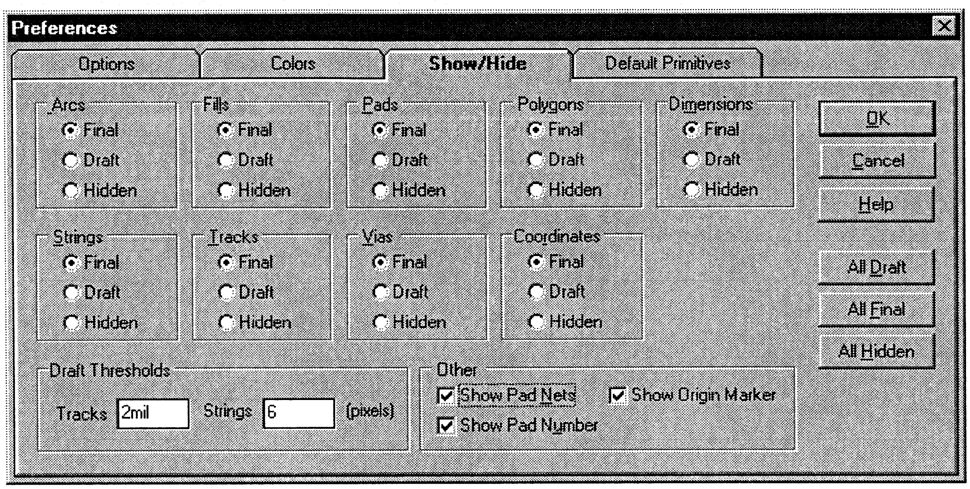
Undo/Redo

Set the stack size to specify how many of the previous operations can be undone. Set the stack size to zero to empty the Undo stack.

Draw Order

Advanced PCB allows you to control the order in which layers are redrawn. Press the Draw Order button to pop up the Layer Drawing Order dialog box. The order that the layers appear in the list is the order they will redrawn in. The layer at the top of the list is the layer which will appear on top of all other layers on the screen.

Show / Hide Tab



Display Mode

The display mode sets how design objects will be displayed on the screen. There are three possible modes; Final, where each object is displayed as solid; Draft, where each object is displayed as an outline (influenced by the draft threshold); and Hidden. Click to select the preferred display mode for each object type, or use the "All" buttons to switch all simultaneously.

Draft Thresholds

Tracks of this width or narrower will be displayed as a single line, tracks of greater width will be displayed as an outline (when tracks are displayed in Draft Mode).

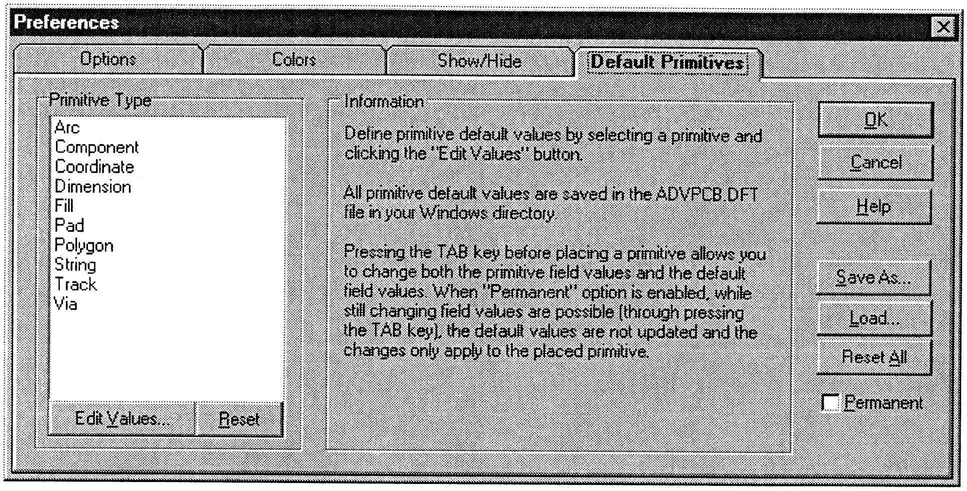
Strings which are this many pixels high or more (at the current zoom level) will be displayed as text, text which are fewer pixels high will be replaced by an outline box. Set these thresholds as required.

Other

Display the pad numbers, net names and the origin marker. The pad numbers and net names will not be visible at lower zoom levels.

Default Primitives

The default settings for each of the design objects available in Advanced PCB are configured in the Default Primitives Tab of the Preferences dialog box.

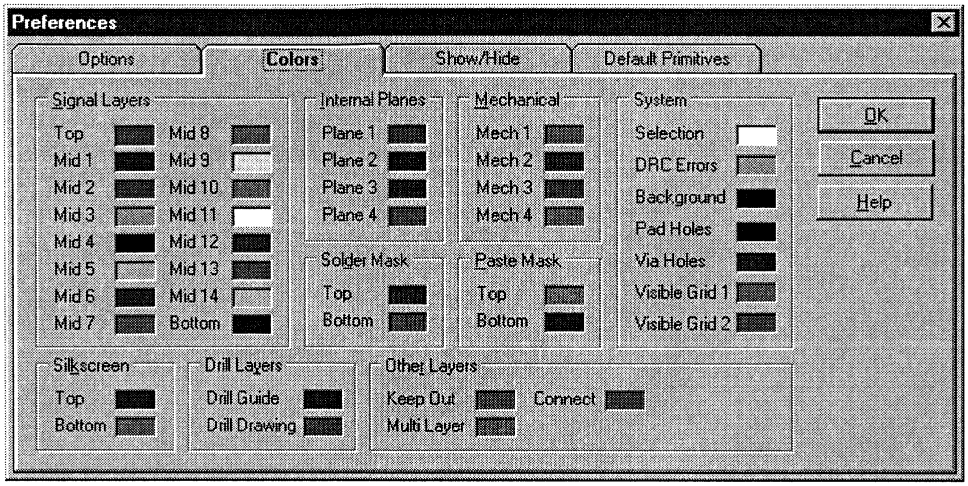


Refer to the *Design Objects* chapter for details about setting the attributes of each design object.

These defaults can be changed "on the fly" during object placement, by pressing the TAB key while the object is floating on the cursor. The changes made on the fly will not affect the defaults if the Permanent option is enabled in the Default Primitives Tab.

Colors

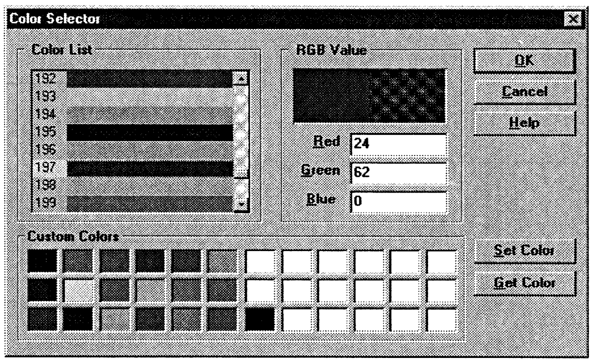
All the workspace colors are defined in the Colors Tab of the Preferences dialog box.



To set a layer color:

1. Click in the color box adjacent to the layer name to open the Color Selector dialog.

The color currently assigned to the layer is displayed in the RGB Value color panel. The Solid color (left pane) is the nearest available solid color assignment for the dithered color at right (dithered colors are colors derived by mixing two solid colors). Depending upon your graphics card and display, you can choose from up to 224 solid colors or a mixture of solid and dithered.



Colors can be chosen from the Color List or from the Custom Colors.

2. A single click will select a color from the Color List. To select one of the Custom Colors, click to select the color then press the Get Color button.

➡ Colors can be created by adjusting the Red, Green and Blue values. To save a created color (or one from the color list) to the Custom Colors palette, click to nominate a square in the palette and press the Set Color button.

3. Click OK to accept the color assignment.

Opening, Saving and Closing Documents

New Document

When you choose the File-New menu item the Select EDA Document Type dialog box pops up. Select PCB from the Available Types and click OK. If the Board Wizard Server is installed the Board Wizard pops up, allowing you to select from one of the many industry standard board templates, or one of your own custom templates. If the Board Wizard Server is not installed an empty PCB document window is opened. For information on installing a server refer to the chapter *A Quick Tour of EDA/Client*.

Opening Documents

Advanced PCB is Windows Multiple Document Interface (MDI) compliant. You can load any number of PCB documents, limited only by available memory.

Advanced PCB will load the following PCB file formats: Advanced PCB (text and binary), Protel Autotrax files (ASCII text files), PADS-PCB and PADS 2000 (.ASC), PCAD (PDIF 5/6) and Tango Series II PCB files.

Loading file formats other than Protel Binary (*.PCB) or Protel ASCII (*.PCB) is done by selecting the File-Import menu item. Translation is performed during the import sequence and non-Protel files are automatically converted to Advanced PCB binary format when saved. Note - PADS files require special handling during the load sequence. Refer to the *Import Options* chapter for further information.

The Advanced PCB binary format is a compact format that load and save faster than the ASCII format. The ASCII format allows you to directly manipulate PCB data.

When no open document (or *file*) is open, the menu bar displays three options: File, View and Help. Select the File-Open menu item to pop up the Open Document dialog box. Select PCB in the Editor drop down list and ensure that the file Type is set to PCB files (*.PCB).

➡ If PCB is not available in the Editor drop down list then the PCB server is not installed. For information on how to install a server refer to the chapter *A Quick Tour of EDA Client.*

Locate the file to be opened in the correct directory, select the file and press the Open button. The file will be opened and displayed in a PCB Editor window.

Saving Documents

There are four Save options on the file menu:

Save

Select the File-Save menu item to save the active PCB design with the same name. The document is always saved in the Advanced PCB binary format. (shortcut; F, S)

Save As

To save the active file with a new name or format select the File-Save As menu item (shortcut; F, A). The Save Document As dialog box will open. There are two file format options available in the Document Types pull down:

PCB Binary files (*.PCB)

This is the default setting - an efficient format that enables Advanced PCB to open or save files more quickly than the ASCII versions.

PCB ASCII (*.PCB)

Protel ASCII format creates larger files and is slower to save and load than the binary version, but allows you direct access to the design database for translation into other formats or other manipulation. If you open a Protel ASCII format file it will be saved in the default binary format when you select File-Save. To re-save as text you must select the File-Save As menu item.

➡ To transfer a file back to an older version of Advanced PCB refer to the *Export Options* chapter.

Save All

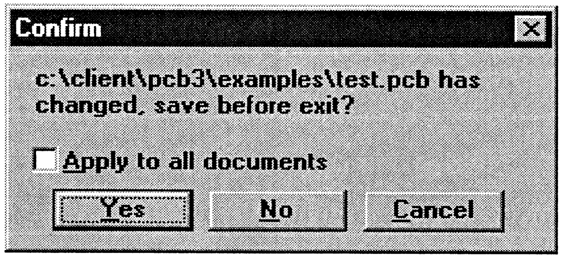
Save All (shortcut; F, L) can be used to save all documents in all currently opened windows. This includes documents of other types, such as text and schematic documents.

Save Project

Save Project (shortcut; F, V) can be used to save all documents which are part of a project. This option is typically used with Advanced Schematic to save multi-sheet projects.

Closing Documents

Selecting the File-Close menu item will close the active PCB document. If the document has changed since last saved, the Confirm dialog will pop up. To close all open documents, select the Windows-Close All menu item. Again, if any document has changed since last saved, the Confirm dialog will pop up.



The Confirm dialog box includes an option "Apply to all documents". Check this if you do not want to be prompted at each file which has changed since the last save. If this is enabled and the Yes button is pressed all files will be saved and then closed. If this is enabled and the No button is pressed all files will be closed without being saved.

Working in Advanced PCB

EDA/Client behaves like any application running in the Windows environment. It can be sized, or it can be maximized to occupy the entire screen. Client can have multiple documents open and you can easily move between them. Where EDA/Client differs from other applications is that different *types* of documents can be worked on in the one environment. These documents are created by different document editors. These document editors are provided by servers, all running in the EDA/Client environment. So rather than having to move across to another application to perform a different task on your design (say, from schematic capture to PCB design), the various tasks can be performed in the one environment. There are many advantages to this approach. Read the chapter *A Quick Tour of EDA/Client* for more information on the client/server architecture.

The Advanced PCB server includes two document editors, the PCB Editor and the PCB Library Editor. Working in either of these document editors is quite similar, you build your design from the set of objects provided, placing these objects in the workspace. The strategies used for placing objects, editing their attributes, positioning and deleting them in the workspace, and so on, are common to both document editors. In essence, the way you work in either editor is the same, what you do in each editor is different.

In the PCB Library Editor you create, edit and validate components and component libraries. In the PCB Editor you create, edit and validate Printed Circuit Boards.

Each of these editors makes use of the facilities provided by EDA/Client. Each has a menu bar, toolbars and a panel and each uses the status bar. Each editor has its own set of shortcut keys and each editor can run macros written in Client Basic or Client Pascal.

Organizing the Workspace

The application and document windows can be sized and positioned like any other Windows application. Refer to your *Windows User's Guide* for tips.

Through the View menu you can toggle the Project Manager, the Editor Panel, the Editor Tabs, the Toolbars and the Status Bars, on and off.

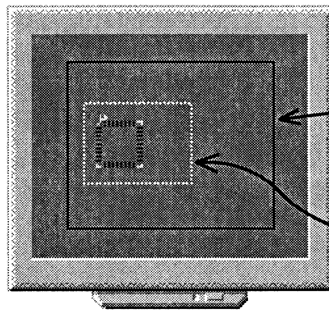
The Project Manager and PCB Editor Panel can be dragged to either side of the workspace, the Editor Tabs can be dragged to any edge of the workspace and the toolbar can also be positioned on any edge of the workspace, or float in the workspace. For more clues on organizing and editing workspace resources read the chapters *A Quick Tour of EDA/Client* and *Resource Management.*

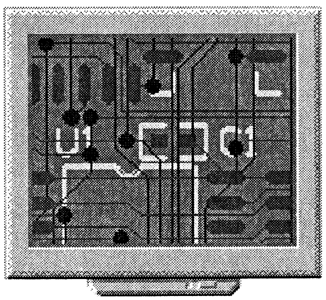
Changing Your View of the Workspace

Each sheet you open will appear in its own window. You look "through" this window to view your design. You can bring the design closer to you (zoom in), or move the design away (zoom out). The PCB Editor Panel and the View menu provide a number of ways of changing you view of the workspace.

The Panel MiniViewer

Use the MiniViewer in the PCB Editor Panel to change your view of the workspace. The dotted ZoomBox marks the current zoom level in the main document window (if the board has a boundary on the Keep Out layer or one of the mechanical layers). Click and drag on one of the ZoomBox corners to change the zoom level in the main document window. Click anywhere within the ZoomBox and drag it to change the region being viewed in the main document window.





Use the Magnifier to present a magnified view of the board in the MiniViewer. Click on the Magnifier button, then simply move the magnifying glass cursor over the board in the main window to examine a particular area in detail. The Configure button allows you to change the magnification level. Alternatively, press the SPACEBAR while using the magnifying glass cursor to toggle through the three magnification levels.

View-Fit Document

Selecting this menu item will change the view to display all objects in the workspace. (shortcut; V, F)

View-Fit Board

Selecting this menu item will change the view to display the entire board, based on the board keep out boundary. (shortcut; V, D)

View-Area

Select this menu item to re-define the display area. Click to define the first corner, then drag the dotted zoom window to define the new display area. (shortcut; V, A)

View-Around Point

Select this menu item to re-define the display area. Click to define the center point, then drag the dotted zoom window to define the new display area. (shortcut; V, P)

View-Zoom Options

The View-Zoom In menu item will bring the design closer to you, relative to the cursor position on the board (shortcuts; V, I or PAGEUP).

View-Zoom Out will move the design away from you (shortcuts; V, O or PAGEDOWN). This is also relative to the cursor, so position the cursor first.

➡ To zoom in and out in smaller increments hold the shift key while pressing PAGEUP or PAGEDOWN.

The View-Zoom Last menu item will return you to your last view of the screen (shortcut: V, L). Repeatedly pressing V, L allows you to toggle back and forth between views.

Moving Around the Workspace

Scrolling

In Advanced PCB scroll bars are provided to allow you to scroll around the workspace. These scroll bars have a sliding button, which you can click and drag to scroll up and down or left and right across the workspace. The position of the sliding buttons gives an indication of where in the workspace you are currently viewing. Click above or below the sliding button to scroll across the workspace in large steps, or click on the arrows at each end of the scroll bars to scroll in small steps.

Manual Panning

To pan across the workspace without using the scroll bars, select the View-Pan menu item (shortcut; V, N or HOME). This will re-center the screen around the current cursor position. The cursor will remain where it was, allowing you to continue panning.

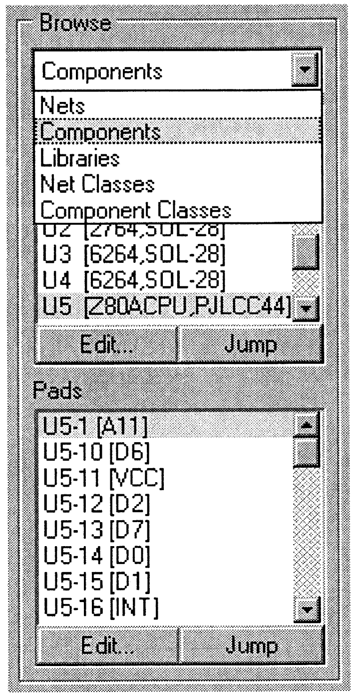
You can also pan across the workspace by pressing one of the arrow keys. The cursor will move one snap grid increment with each press. Holding the shift key as you press an arrow key moves the cursor in steps of 10 times the current snap grid.

Auto Panning

Auto panning is enabled whenever you have a cross hair cursor. You have a cross hair cursor whenever you perform an "edit" type operation such as placing, selecting, moving or deleting an object. This cursor can be moved either by moving the mouse or pressing the arrow keys on the keyboard. If the cursor is moved such that it hits the window frame, you will pan across the workspace. Advanced PCB has four Auto Pan modes. The mode is set in the Options Tab of the Preferences dialog box.

Browsing

The PCB Editor Panel can be used for browsing through the Current PCB. There are four Browse modes specifically for this task; Nets, Components, Net Classes and Component Classes.



- If you browse by Net or Component the small MiniViewer in the panel will display the selected net or component.
- Use the Zoom button to quickly locate and identify the selected net or component in the main document window.
- When you select an item in a list information about that item will appear on the Status Bar.

Jumping

The Edit-Jump menu (shortcut; J) allows you to conveniently locate a specific component, net, pad on a component, text string or board location without having to zoom, pan or scroll through multiple screens.

With all of these options, Advanced PCB will position the cursor on the target, only redrawing the screen if the search target is outside the current display area. When a redraw is needed, the target will be centered in the active window.

Jump options include:

Absolute Origin

Jump to the absolute origin. In Advanced PCB this is the lower-left corner of the workspace. (shortcuts; J, A or CTRL HOME)

Current Origin

Jump to the current (or relative) 0,0 origin (shortcuts; J, O or CTRL END). This origin is specified by selecting the Edit-Origin-Set menu item.

New Location

Jump to the specified location. Edit-Jump-New Location pops up the Jump To Location dialog box. The X and Y location text fields will contain the current cursor position. The cursor will jump to the specified location. (shortcut; J, L)

Component

Jump to the specified component. Edit-Jump-Component pops up the Component Designator dialog box. Type in the designator and click OK. If you do not know the designator, type ? and press ENTER or click LEFT MOUSE to scan the board for all placed components. Choose from the Components Placed dialog box and click OK. The cursor will jump to the reference point of the selected component. (shortcut; J, C)

Net

Jump to a pin on the specified net. Type the net name in the Net Name dialog box and click OK. If you do not know the net, type ? and press ENTER or click LEFT MOUSE to scan the board for all nets. Choose from the Nets Loaded dialog box and click OK. The cursor will jump to the nearest pin that belongs to the selected net (shortcut; J, N).

Pad

Jump to the specified pin on the specified component. Type the component designator and pin number in the Jump to Pin Number dialog box (e.g.; U1-6) and press enter. The cursor will jump to the center of the pin. (shortcut; J, P)

String

The cursor will jump to the named string. The system will perform three searches:

First – for a string that matches the specified string in both case, characters and length.

Then – for a string with same characters in it but perhaps having more characters.

Finally – for a string with same characters but ignoring case.

For example, typing "component" would find the string "component" first. If no match is found it would next find the string "components" and finally "CompONENT". When the string is found, the cursor will be relocated to the specified string. (shortcut; J, S)

Error Marker

Select this menu item to jump to the first DRC error marker. Repeating will jump to a second error marker, and so on. Removing the violation will clear the error from the "Jump to" list. Otherwise, repeating will continue to cycle through all errors in the current document window.

Selection

Select this menu item to jump to the first selected object. Repeating will jump to a second selected object, and so on. Repeating will continue to cycle through all selected objects in the current document window.

➡ Jumping around the workspace can be a very efficient way of working in Advanced PCB, as it allows you to reposition your view of the workspace without zooming out and in. To speed the process even more, all the Jump process launchers can be executed without the use of the mouse. Use the shortcut keys to pop the dialog box. For example, to jump to the location 1000, 1000, press the J, L shortcut keys. When the Jump To Location dialog box pops up the X-Location text box will be highlighted. Highlighted text is replaced by whatever you type in any Windows application. Simply type the new X-location in. To move to the Y-Location text box, press TAB. As with all Windows dialog boxes, TAB moves the cursor forward to the next field or button, SHIFT TAB moves you backward. Type the new Y-Location in. Press ENTER on the keyboard. The dialog box will close and the cursor will jump to the location 1000, 1000.

Editing

There are two approaches to editing an object in Advanced PCB, either by changing its attributes in its Change dialog box or graphically modifying the object. Certain operations (such as changing the color of a primitive) can only be performed by editing the attributes, others (such as re-sizing a fill) can be done through the dialog box or by graphically modifying the object.

To edit the look of an object in the workspace it is generally easier to do it graphically. To do this you must first bring the object into *focus*. To edit the object through its Change dialog box select the Edit-Change menu item and then click on the object.

Editing While Placing

You can edit the attributes of an object while it is being placed. While the object is floating on the cursor, press the TAB key. This pops up the object's dialog box. The advantages of editing during placement are;

- Changes made to attributes can become the defaults for that type of object. These changes are stored in the defaults file ADVPCB.DFT. Note - this method of setting defaults depends on the setting of the "Permanent" check box in the Preferences dialog box (Tools-Preferences). If this is on, these changes will not become the defaults and will only apply to the object floating on the cursor.
- Objects that have a numeric identifier, such as pad designators, will auto-increment.
- There is no need to edit the object after it has been placed, speeding the design process.

Changing Placed Objects

The Edit-Change menu item is used to modify a placed object. Each object has its own range of editable attributes. You can change one object or extend changes across your entire design using powerful global editing options.

To change any placed object select the Edit-Change menu item, move the cursor over the item and click LEFT MOUSE (shortcut: double-click LEFT MOUSE). If there is more than one object under the cursor a selection list will pop up.

➡ Refer to the *Design Objects* chapter for details of the editable attributes of each object and the *Global Editing* chapter for tips on Global editing.

Editing Graphically - Focus and Selection

One of the advantages of a graphical based editing environment is the ability to make changes directly to objects displayed on the screen. To be able to graphically edit an object or a group of objects you must first identify the objects. This is done through *focus* or *selection.*

In other Windows applications selection is a single concept, the process of choosing or designating objects as a prerequisite for modification. A typical example would be selecting one or more objects to be *copied* to the clipboard and then *pasted* to another location. Often selected objects can be modified directly. For example, selected objects can be moved or re-shaped in most graphical applications.

Unlike other Windows applications, Advanced PCB uses two independent methods for accomplishing selection oriented tasks. These methods, *selection* and *focus* are used repeatedly when creating or editing your PCB. Breaking selection into two these independent processes allows Advanced PCB to perform complex modifications of objects which would be either difficult or impossible using the simple selection method described above.

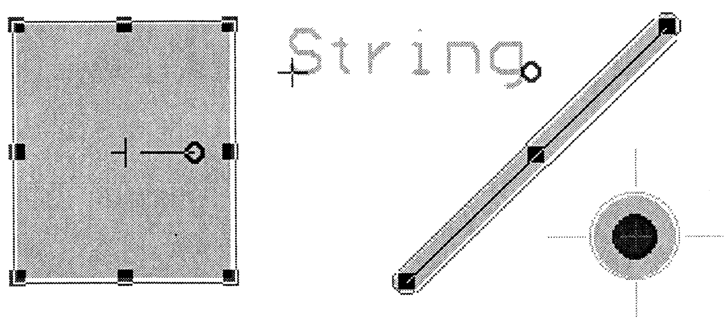
➡ Focus and selection provide two distinct and independent methods for changing objects in the workspace. These two methods distinguish Advanced PCB from other Windows applications where focus and selection are normally merged into a single operation.

Focus

When you position the cursor over a design object in Advanced PCB and click LEFT MOUSE this object then "has the focus" and the way it is displayed changes. This is similar to the way you can change the focus in Windows by clicking on an open window to make it active.

Only one object can be in focus at a time. You can tell which object is currently in focus because its graphical editing handles and/or a focus cross-hair are displayed. For example, if you click LEFT MOUSE over a fill a re-sizing handle will appear at each corner and along each side, and a rotation handle will appear near the center. If you click LEFT MOUSE over a via a focus cross-hair will appear. To move the focus to another object click on that object. Click in a clear area of the workspace to release the focus.

Graphical editing



Objects in focus, displaying their focus handles

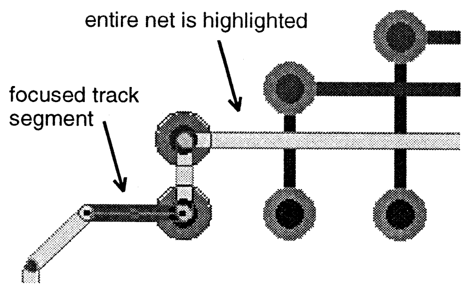
When an object is in focus, you can move the object or edit its graphical characteristics. For example, you can change the size or shape of a fill by dragging the square re-sizing handles. Click LEFT MOUSE on the circular rotation handle to rotate a fill or a string.

When you focus a track segment three editing handles appear, one at each end and one in the center. Click on one of the end handles to move that end or click on the center handle to "break" the original track segment into two segments.

Click anywhere on a focused object to move it. For strategies about moving or dragging objects refer to the *Moving and Dragging* topic later in this chapter.

Focusing a Track Segment that Belongs to a Net

When you click on a track segment it comes into focus. When you click on a track segment that belongs to a net, as well as the segment of track that you clicked on coming into focus, all other objects that belong to the same net will highlight. This makes it very easy to trace a net through your design. Remember, this feature only operates when the objects have a net name.



Summary

As illustrated in these examples, *focus* is prerequisite to a number of graphical editing functions that are performed on individual objects. Note that you cannot use the clipboard menu items: Edit-Copy, Cut, Paste or Clear with the focused object. These Clipboard features work only on a *selection*.

Selection

Selection provides a second, distinct method of manipulating objects. Unlike focus, selection can be used with both individual objects and with groups of objects.

➡ Selection supports the clipboard process launchers: Edit-Copy, Cut, Paste and Clear.

Unlike *focus*, described above, selection does not display an object's graphical editing handles or a cross-hair. Instead, the object is outlined in the *selection color* (Options-Preferences).

Once selected, objects can be moved, grouped, un-grouped, exported to another file, cut, copied, pasted into another window or location in the current window, or cleared. Special Edit-Move process launchers allow selections to be moved or rotated in a single operation. Selection also works with Advanced PCB's global editing feature, which can limit global changes to selected or un-selected objects.

➡ A key feature of Advanced PCB's complex selection model is the ability to click LEFT MOUSE without *de-selecting* objects that were previously added to the current selection. This allows you to perform a wide variety of operations without effecting the current selection.

Selection in Advanced PCB can operate in two modes. Selection can be cumulative (extendible), where objects remain selected until specifically de-selected. The other mode is non-cumulative. In the non-cumulative mode all currently selected objects are de-selected when one of the Edit-Select menu items is chosen. The extend selection mode is toggled in the Options Tab of the Preferences dialog box.

Selections are made in the following ways:

- Direct selection, using SHIFT+LEFT MOUSE to add (or remove) individual items to the current selection.
- The click-and-drag-a-window-around shortcut.
- Use the Edit-Select and Edit-DeSelect sub-menus to define a selection.
- By using the Selection field in Change dialog boxes. This option allows you to use Advanced PCB's global editing feature to apply selection status changes to other primitives in the current board window. Refer to the *Global Editing* chapter for tips on global editing.

➡ Care must be taken when manipulating selections to ensure that the current selection includes *only* the desired objects. Use the Edit-De-Select All process launcher (shortcut: X, A) to clear the current selection prior to making a new selection.

If something unexpected happens, remember that the Edit-Undo process launcher can be used to restore a previous condition.

Displaying Selections

Selections are outlined in the selection color specified in the Display Tab of the Preferences dialog box. There are a number of ways of affecting the visibility of a selection.

Draft Mode

If the primitive has its display mode set to Draft, it will be outlined in the selection color and displayed in Final mode when it is selected. This makes it very easy to identify. Set the display mode in the Show/Hide Tab of the Preferences dialog box.

Highlight In Full

To display the entire primitive in the selection color, enable the Highlight In Full option in the Display Tab of the preferences dialog box.

Use Net Color For Highlight

Nets can also be highlighted in their net color. Set the net color in the Change Net dialog box (Set the Browse mode in the panel to Nets, select the net and press the Edit button). This option works well in combination with the Highlight In Full option. Enable the Use Net Color For Highlight option in the Display Tab of the preferences dialog box.

Making Selections

Direct Selection of an Individual Object

Direct selection is the most flexible way to select an individual object. To select one object at a time:

1. Hold down SHIFT and click LEFT MOUSE with the cursor positioned over an object.

The item will be redrawn, outlined in the Selection color (Tools-Preferences). You can do this repeatedly, each time adding another item to the current selection.

If you hear a "beep" or nothing appears to be selected, try zooming in closer (press PGUP) and make sure that the cursor is directly over the item you wish to select. To select a component, position the cursor within the component outline. Components, especially complex components can take a moment to select.

To add another item to the current selection:

2. Hold down SHIFT and click LEFT MOUSE over another item.

To release individual items from the selection:

3. Hold down SHIFT and click on the selected item.

When released, the item will be redrawn in its original colors. Other selected items remain selected until they are either individually released (SHIFT+LEFT MOUSE) or until an Edit-DeSelect is executed.

Direct Selection of an Area

Direct selection can also be performed on an area. To select all objects within an area:

1. Position the mouse where there are no objects under the cursor.
2. Click and hold the LEFT MOUSE button.

The Status Bar will prompt "Select Second Corner".

3. Drag the mouse diagonally away. Define the area with the selection rectangle and release the LEFT MOUSE button.

Only objects that fall entirely with the rectangle will be selected. The selected objects will highlight in the current selection color.

4. Repeat the process to extend this selection or to make a new selection.

➡ The setting of the Extend Selection option in the Options Tab of the Preferences dialog box determines whether selection is cumulative or non-cumulative.

The Select and DeSelect Sub Menus

The Edit-Select menu allows you to select all items inside or outside of an area, all items on one layer, or all free primitives (all items other than components). You can also select by physical net or off grid component pads.

Use the Select and DeSelect options to define complex grouping which can then be moved, copied or deleted.

➡ Use the S shortcut key to pop up the Select sub menu and the X shortcut key to pop up the DeSelect sub menu.

Select and De-select menus include:

Inside Area

Allows you to define a rectangular selection area. Only those objects that lie completely inside the area are included. Free pads or vias are included in the selection if their center is inside the rectangle.

Outside Area

This option selects everything outside the selection rectangle. The rules for inclusion in the selection are the same as for Select-Inside Area. The procedure for defining the selection rectangle is the same as for Inside Area.

All

Selects everything placed in the document window. This includes all objects which have their display state set to hidden or are not visible because their layer is off.

Physical Net

This selection will include all primitives (tracks, vias, fills) that are in *physical contact* with the point where you click. It will not include any part of a net that is not physically connected, that is, connected by a connection line.

To use this feature:

1. Choose Edit-Select Physical Net (shortcuts; CTRL+H or S, N).

You will be prompted "Select Physical Net Starting Point".

2. Position the cursor over any primitive within the desired net and press ENTER or LEFT MOUSE.

The continuous physical net, extending from the selection point, will highlight in the selection color.

All On Layer

This selection will include all primitives on the current layer. Multi layer items (typically multi-layer pads and vias) are excluded from the selection.

Free Objects

This option selects all objects which are not part of a group (component, polygon, dimension or coordinate). This feature is useful for stripping a routed, or partially routed board back to its "placed" condition. Limit the selection by turning off layers which contains objects you do not want selected.

All Locked

This will select all primitives and components that have their Locked attribute set.

Off Grid Pads

Choose Edit-Select-Off Grid Pads to select all component pads that do not fall on the current snap grid. Use this prior to autorouting to check how many component pads are off grid. Use the partner process, Tools-Align Components-Move To Grid to bring the components onto the current snap grid (shortcut; A, G).

Using the Query Wizard to Create Complex Selections

Advanced PCB includes a Query, or selection Wizard which you can use to create complex sets of selections. This Wizard allows you to simultaneously select different types of primitives, based on a set of user definable selection criteria. For example, you could select all pads and vias with a hole size <= 0.5 mm, or all tracks whose width <> 8 mils.

Select the Edit-Selection Wizard menu item to launch the Wizard. If the Wizard does not launch you may have to install the Query Wizard Server. For tips on installing a server refer to the chapter *A Quick Tour of EDA/Client*.

Working with a Selection

The Clipboard

Advanced PCB uses a special proprietary clipboard format that supports PCB data such as connectivity and layer attributes of primitives. This internal Protel clipboard is not the same as the standard Windows clipboard that allows you to move selections, such a text, between various Windows applications. The Windows MetaFile (.WMF) graphics format is not supported in the Advanced PCB clipboard.

Use the clipboard in Advanced PCB the same as you would in any Windows application. The sequence is; select the objects to perform the operation on, cut or copy the selection to the clipboard, then paste the clipboard contents to the desired location.

Cutting

Edit-Cut clears the current selection from the workspace and copies it to the (internal) Advanced PCB clipboard. Edit-Paste can be used to place the selection back into any open Advanced PCB document window.

➡ The clipboard holds the last Cut or Copy selection contents only, each time you use Cut or Copy you overwrite the previous selection.

To cut the current selection from the active window:

1. Make sure that the current selection includes only those items you wish to cut.

Use the shortcut SHIFT+LEFT MOUSE to add items to the current selection or to de-select any currently selected items.

2. Choose Edit-Cut (shortcut; SHIFT+DEL).

You will be prompted “Select Reference Point” on the Status line. The point where you click will be the point you will be holding the selection by when you paste it back into the workspace.

3. Position the cursor at the desired reference point and click LEFT MOUSE or press ENTER.

The selection will be cleared from the display and copied to the clipboard. You may need to use View-Redraw (shortcut; END) to restore the display under the selected items.

Copying

Edit-Copy copies the current selection to the (internal) Advanced PCB clipboard. Edit-Paste can be used to place a copy of the selection back into any open Advanced PCB document window.

➡ The clipboard holds the last Cut or Copy selection contents only, each time you use Cut or Copy you overwrite the previous selection.

To copy the current selection from the active window:

1. Make sure that the current selection includes only those items you wish to copy.

You can use the shortcut SHIFT+LEFT MOUSE to add items to the current selection or to de-select any selected items.

2. Choose Edit-Copy (shortcut; CTRL+INS).

You will be prompted “Select Reference Point” on the Status line. The point where you click will be the point you will be holding the selection by when you paste it back into the workspace.

3. Position the cursor at the desired reference point and click LEFT MOUSE or press ENTER.

The selection will be copied to the clipboard.

Clearing

Edit-Clear deletes the current selection from the workspace without copying it to the clipboard.

To clear the current selection from the active window:

1. Make sure that the current selection includes only those items you wish to clear.

You can use the shortcut SHIFT+LEFT MOUSE to add items to the current selection or to de-select any selected items.

2. Choose Edit-Clear (shortcut; CTRL+DEL).

The selection will be cleared from the display. You may need to use View-Redraw (shortcut; press END) to restore the display under the selected items. You can use Edit-Undo (shortcut; ALT+BACKSPACE) to restore the cleared selection.

Pasting

Edit-Paste can be used to place the current clipboard contents into any open Advanced PCB document window.

To paste the current selection from the clipboard:

1. Choose Edit-Paste (shortcut; SHIFT+INS).

You will be prompted "Select Location to Place Selection" and a highlighted outline of the selection will be displayed. The position where the cursor holds the selection is determined by the Reference Point you selected when Cut or Copy was used to add the selection to the clipboard.

2. Position the selection in the workspace and click LEFT MOUSE or press ENTER.

You can repeat the Paste action to duplicate the selection.

If components are added by the paste action their designators will have the word "copy" added to avoid duplication of existing designators. If you wish to duplicate the designators, use the Edit-Paste Special option.

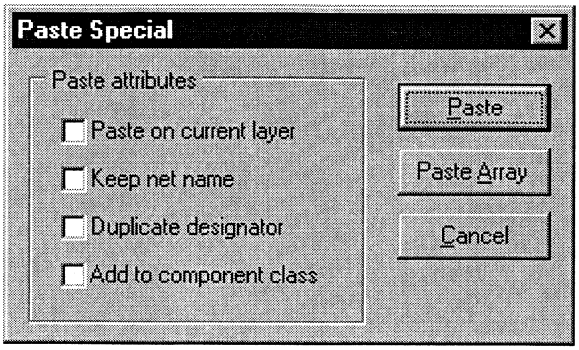
Paste Special

Paste Special allows you to control what happens to the attributes of the objects that are in the clipboard when they are pasted back into the workspace.

To control the attributes as you paste the current clipboard contents:

1. Choose Edit-Paste Special.

The Paste Special dialog will appear. Paste Special includes the following options;



Paste on current layer

If this option is disabled all single layer objects such as tracks, fills, arcs and single layer pads keep their existing layer assignments. If this option is enabled then all single layer objects are pasted onto the *current layer.*

Keep net name

If this option is enabled then all objects which have a net name will keep the assigned net name. If this option is disabled the net attribute is set to "No Net".

Duplicate designator

This option supports the creation of a PCB panel, where you wish to copy and paste the entire design. Typically the Keep net name option would be disabled if this option is enabled.

Add to component class

Pasted component(s) are added to the same class as the component(s) they were copied from.

2. Click OK after setting the options in the dialog.
3. Position the selection in the workspace and click LEFT MOUSE or press ENTER.

Creating a Panel

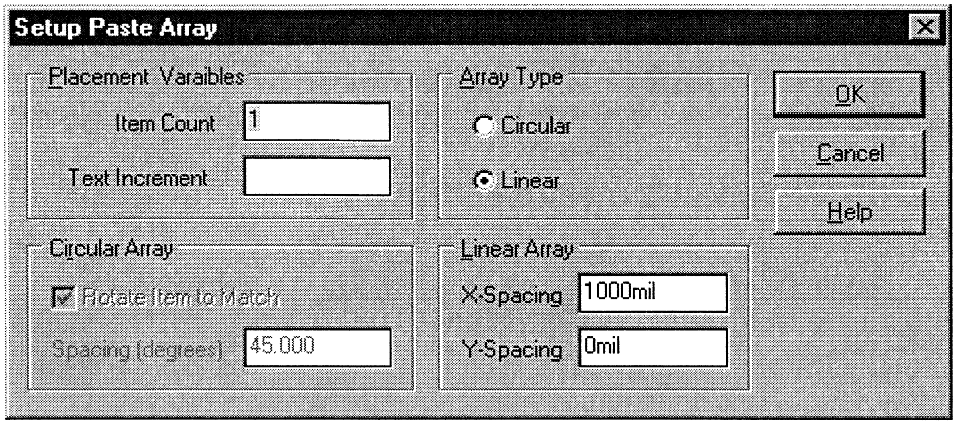
Advanced PCB can be used to create a panel of PCBs. This panel could be multiple copies of the same PCB or be made up of different PCBs. To create a panel, first copy the PCB to the clipboard. Then use the Paste Special feature to copy the PCB the required number of times. If you have difficulty positioning accurately as you paste, use the Jump Location process (shortcut; J, L) while the paste is floating on the cursor. This allows you to position the paste without using the mouse. Remember, you can move through fields in a dialog box by pressing the TAB key (SHIFT+TAB to go back through fields) and press ENTER instead of clicking OK.

Paste Array

When you use Edit-Cut (or Copy) you are placing a copy of the current selection in the clipboard. Edit-Paste Array provides a powerful way to place multiple copies of the clipboard contents back into the workspace.

To paste an array of the current clipboard contents:

1. Choose Edit-Paste Special. Set the Paste Special dialog as required and press the Paste Array button. The Paste Array dialog will appear.



Placement Variables

Item Count

The number of repeat placements to be performed. For example, typing 4 will place 4 of the current clipboard contents.

Text Increment

This option is used for designators on pads and components. Setting this to 1 (default) will increment the designators in series, for example U1, U2, U3 etc.

Both alpha and numeric increments other than 1 are also supported. By setting the designator of a pad prior to copying it to the clipboard and setting the Text Increment field, the following types of pad designator sequences can be placed: Numeric (1, 3, 5); Alphabetic (A, B, C); Combination of alpha and numeric (A1, A2, or 1A, 1B, or A1, B1 or 1A, 2A, etc).

To increment numerically set the Text Increment field to the amount you wish to increment by. To increment alphabetically set the Text Increment field to the letter in the alphabet that represents the number of letters you wish to skip. For example, if the initial pad had a designator of 1A and the Text Increment field was set to C (the third letter of the alphabet), the pads would have the designators 1A, 1D (three letters after A), 1G (three letters after D), and so on.

Array Type

Circular

Repeat placements are made in a circular array using the rotation and spacing values specified under Circular Arrays.

Linear

Repeated items are placed in a linear array, using the spacing values specified under Linear Array.

Circular Array

Rotate Item to Match

Array items will be rotated by the same angular amount as their Spacing.

Spacing

Specifies the angular spacing between each pasted item. Advanced PCB has an angular resolution of 0.001 degrees.

Linear Array

These values specify the X and Y distance between each item as it is placed.

2. Enter the desired values and click OK to place the array.

   When placing the array you are prompted either:

   "Select Starting Point For Array" if the Array Type is Linear, or:

   "Select Center Of Circular Array" if the Array Type is Circular.

3. Position the cursor and press ENTER or LEFT MOUSE.

   If you are placing a circular array you will then be prompted "Select Starting Point For Array". This will set the radius and start point of the array.

4. Position the cursor and press ENTER or LEFT MOUSE.

The clipboard contents will be pasted multiple times to create the desired array.

Moving and Dragging

To *move* an object is to reposition it without regard to objects in contact with it. For example, if you move a component, all the tracks connected to its pads will not move. If you *drag* the component, all the tracks will remain connected to the pads as you reposition it. Move operations can be performed on a selection or on individual objects.

➡ The concept of dragging is only applicable when the objects are part of a net.

Move Shortcut

To select and move any object or selection:

1. Position the cursor over the object to be moved.
2. Click and hold LEFT MOUSE.

The Status Bar will report what is being moved.

3. Move the object to the new location.

Drag Shortcut

To drag an object that is part of a net:

1. Position the cursor over the object to be dragged.
2. Click LEFT MOUSE to focus the object.

If the object is part of a net the entire net will highlight. If no other objects highlight then this object does not have the same net attribute as the joining objects and can not be dragged.

3. Click LEFT MOUSE on the object again.

If you click on one of the sizing handles of the focused object, you will drag that handle rather than the entire object. For more information on the behavior of a focused object refer to the *Focus* topic in the *Editing* section of this chapter.

4. Drag the object to the new location.

Dragging a Component

To drag a component:

1. Select the Edit-Move-Drag menu item (shortcut; M, D).
2. Click on the component you wish to drag and move it to the new location.

The behavior of the tracks that connect to and pass under the component are influenced by the Component Drag option in the Preferences dialog box (Options-Preferences). Move-Drag can also be used to drag any primitive object.

Selection Moves

Once selected, an individual item or a complex selection containing many items can be moved as a single entity. Select Edit-Move or press the M shortcut key to pop up the Move sub-menu. This sub-menu includes;

Move Selection

This option allows you to select a new location for the selection, which will move as a block (shortcut; M, L). When you invoke this process you will be prompted for a reference point.

Flip Selection

This option flips the selection around the vertical axis. Pressing the X or Y keys during a Move Selection can also be used to flip a selection.

Rotate Selection

Selecting this option will pop the Rotation Angle dialog box. Enter the rotation angle, which has an angular resolution of .001 degrees. You will then be prompted to select a reference point, about which to pivot the rotation.

Selections can also be rotated during a Move Selection by pressing the SPACEBAR. The spacebar rotation angle is set in the Options Tab of the Preferences dialog box.

For Gerber plot final artwork, the target photoplotter may not allow rotated primitives. Rotated rectangles (such as rectangular component pads) will be painted using a round aperture during Gerber generation in Advanced PCB.

Moving Individual Items

The other Edit-Move process launchers work with items that have not been previously selected. Because these Move process launchers manipulate specified items, it is easy to control the selection in a dense layout, where many items overlap.

Moving or deleting one or more items can leave a "hole" in the display under the moved/deleted primitives. This is because Advanced PCB does not continuously redraw the screen during moves or deletions, as this would significantly slow system performance. Click the Redraw button on the Tool bar or press END to refresh the screen.

➡ Rotatable objects, such as components, pads and text strings, can be rotated in anti-clockwise steps during moves by pressing SPACEBAR, or in clockwise steps by pressing SHIFT+SPACEBAR. The degree of rotation is pre-set in the Options Tab of the Preferences dialog box.

Break Track

Break Track converts a single track segment into two connected segments. To break a track:

1. Choose the Edit-Move-Break Track menu item (shortcut; M, B).

You will be prompted to "Choose a track".

2. Position the cursor over the track segment and press ENTER or click LEFT MOUSE.

The track will be displayed in draft mode.

3. Move the cursor to drag the break point to a new location.

4. Press ENTER or click LEFT MOUSE again to complete the move.

You will be prompted "Choose a track" again. During the drag, you can abort the move by clicking RIGHT MOUSE or pressing ESC once. Note that the "Select track" prompt is still displayed.

5. Select another track or press ESC (or click RIGHT MOUSE) a second time to quit from the break track mode.

Polygon Vertices

The boundary of a polygon plane can be reshaped by moving the vertices. To move the polygon vertices;

1. Select the Edit-Move-Polygon Vertices menu item.

The Status Bar will prompt "Choose a Polygon".

2. Click on the polygon to be edited.

The internal polygon tracks will disappear, leaving only the boundary tracks.

3. Click on the vertex (editing handle) to be moved. The vertex can then be repositioned.

4. Continue to reposition the vertices.

5. Click RIGHT MOUSE or press ESC when finished.

You will be asked if you wish to re-pour all modified polygons. Click No if you want to modify other polygons and would prefer to save re-pouring them until they have all been modified. Click Yes to immediately re-pour the modified polygon.

Refer to the *Polygons* topic in the *Design Objects* chapter for further information about working with polygons.

Deleting

Select the Edit-Delete menu item to remove objects from the PCB workspace. Delete differs from the Cut or Clear processes described previously. With Cut or Clear you identified the objects first (selected them), then picked the action (Cut or Clear). To Delete you pick the action first (Delete), then click on the object.

➡ If more than one object is under the cursor when you click to delete, a pop up menu will appear allowing you to choose exactly which object to delete.

Delete is also independent of selection. For example, when deleting individual tracks, tracks that are part of the current selection will be left undisturbed.

All deletions can be restored by using Edit-Undo (or ALT+BACKSPACE). If you have deleted a series of items, they will be restored one-at-a-time starting with the last deleted item. Edit-Redo uses the same first-in/last-out logic. Redo reverses the Undo operations, one-at-a-time.

➡ Press the DELETE key to immediately delete the focused object. Press the CRTL+DELETE keys to immediately delete the current selection.

Editing Tips

Re-entrant Editing

Advanced PCB allows you to launch a process whilst currently executing a process. This facility is known as re-entrant editing. This is a powerful feature, allowing you to perform an operation without having to quit from the operation you are currently carrying out.

Re-entrant editing allows you to work more flexibly and intuitively. For example, you start placing a track, then realized that another track segment must be deleted. There is no need to drop out of Place Track mode. Simply press the Delete shortcut keys (E, D), delete the required track segment and click RIGHT MOUSE or ESC to terminate the Delete process. You are now back in the Place Track process, ready to place the new segment.

➡ Accessing another process while executing a process is only possible via the shortcut keys.

Any number of processes can be completed within another process. The number of times another process can be launched before the current process is complete depends on the demands each of these incomplete processes is placing on the software. For graphical type processes approximately ten processes can be nested. A dialog box will pop up if the limit has been reached.

Canceling a Screen Redraw

Whenever you change the size and/or position of your view of the screen, the contents of the workspace will be redrawn to reflect the change. You can terminate the redrawing process by pressing the SPACEBAR anytime while the redraw is in progress. This saves time whenever you wish to immediately scroll or zoom again, without waiting for the redraw to complete. Use the solid (not transparent) layer display option to make redraws faster (Options Tab of the Preferences dialog box). Working with the Display mode options set to Draft for all primitives will also slightly improve redraw speed (Show/Hide Tab of the Preferences dialog box).

➡ To redraw the current layer only use the ALT+END shortcut keys.

Mouse Shortcuts

As you read through this guide, you will notice several mouse and keyboard shortcuts that are used to speed-up or simplify frequently performed operations. For example, pressing P, P allows you to place a pad without having to go to the Place menu and choose the Pad menu item. Using the left mouse button for ENTER and the right mouse button for ESC will allow you to perform many operations without using the keyboard. The opposite can also be done, press ENTER or ESC on the keyboard rather than clicking OK or CANCEL in a dialog box. Sometimes keyboard actions provide the only practical way of performing an operation when you do not wish to move the mouse in the workspace, such as setting a new grid while placing an object, or changing the zoom level while moving a selection.

➡ If you double-click on any placed item, the Change dialog box for that item will be opened, allowing you to edit its attributes.

➡ To move an item simply click and hold LEFT MOUSE, hold on the object and drag the mouse to the new position.

➡ To delete an object from the design, press the DELETE key and click on the object you wish to delete.

Standard Windows shortcuts, such as pressing ALT+F4 to close the application window or CTRL+TAB to toggle through document windows are supported. Other shortcut keys are specific to Advanced PCB. You can also create your own custom shortcut keys. Refer to the chapter *A Quick Tour of EDA/Client* for clues on creating your own shortcut keys.

Windows allows you to assign operations to specific key combinations by using the Recorder feature. See your *Microsoft Windows Users Guide* for details.

Keyboard Shortcuts

There are two ways of creating shortcuts invoked through the keyboard. The first is through the Keyboard Shortcut Editor (Client menu-Edit Shortcuts). These are known as Keyboard Shortcuts. They launch a process directly. For example, pressing CTRL+G will pop up the Snap Grid dialog box, allowing you to change the snap grid.

Processes can also be launched through the keyboard via the menu keyboard shortcuts. Underlined menu items which lead to a sub-menu will pop up that sub-menu, underlined menu items which do not pop up a sub-menu will launch the process tied to that menu item. For example, press P to pop up the Place menu, then press V to present the current via on the cursor, ready for placing. Press T to pop up the Tools menu, followed by R to pop up the Auto Route sub-menu.

➡ If the same keyboard key has been assigned as a keyboard shortcut and as a menu shortcut the keyboard shortcut will take precedence.

Menu Shortcuts include:

A	Tools-Align menu
B	View-Toolbars sub-menu
D	Design menu
E	Edit menu
F	File menu
G	Snap Grid pop-up menu
H	Help menu
J	Edit-Jump sub-menu
M	Edit-Move sub-menu

O	Options pop-up menu
P	Place menu
R	Reports menu
S	Edit-Select sub-menu
T	Tools menu
U	Tools-Unroute sub-menu
V	View menu
W	Window menu
X	Edit-DeSelect menu
Z	Zoom pop-up menu

Keyboard Shortcuts include:

L	Layers Tab of Document Option dialog
Q	Toggle Units
CTRL+G	Snap Grid dialog box
CTRL+H	Edit-Select-Physical Net
CTRL+P	Run Process
CTRL+Z	Cross Probe
PGUP	View-Zoom In
PGDN	View-Zoom Out
CTRL+PGUP/PGDN	Zoom maximum / minimum
SHIFT+PGUP/PGDN	Zoom at 0.1 step rate
HOME	View-Pan
END	View-Refresh
CTRL+HOME	Jump Absolute Origin
CTRL+END	Jump Current origin
CTRL+INS	Edit-Copy
CTRL+DEL	Edit-Clear
SHIFT+INS	Edit-Paste
SHIFT+DEL	Edit-Cut
ALT+BACKSPACE	Undo
CTRL+BACKSPACE	Redo
SHIFT+F4	Cascade Windows
SHIFT+F5	Tile Windows
*	Toggle active signal layers
+ or -	Next / previous active layer
F1	Help Index
UP, DOWN	Move one snap grid point, vertically
SHIFT+UP, DOWN	Move 10 snap grid points, vertically

LEFT, RIGHT	Move one snap grid point, horizontally
SHIFT+LEFT, RIGHT	Move 10 snap grid points, horizontally

Special Mode-Dependent Keys

TAB	Opens a dialog box for the object currently being placed. Allows you to edit attributes when placing any object. Use the tab key to "edit on the fly".
SPACEBAR	Toggles between Start and End track placement modes; rotate item anti-clockwise during move (set step value in Preferences dialog box); abort screen redraw; change the MiniViewer magnification level.
SHIFT	Control acceleration during Autopan (set mode in Preferences dialog).
SHIFT+SPACEBAR	Toggle track placement modes; rotate item clockwise during move.

Locating Components

Often you will know what component you wish to edit or move but do not have the view set to see it. For example, you might want to place a particular component where you are currently working but do want to scroll or zoom to find it. Select the Edit-Move-Move Component menu item (shortcut; M, C). Click somewhere in the workspace where there is no component under the cursor and the Component Designator dialog box will pop up. If you know the designator type it in and click OK. If the component is off screen the view will start to scroll so move the mouse to bring the cursor and component into view.

When the Component Designator dialog box pops up you can also leave the ? and click OK. This will pop up the Components Placed dialog box, allowing you to select any component that is in the workspace.

Undo and Redo

Advanced PCB includes a full multi-level Undo and Redo facility. Each procedure is stored in a stack-like arrangement. When Undo is chosen, the last operation is undone. Choosing Undo again will undo the next-to-last operation, and so on.

The Redo facility will reverse a previous Undo. Every time you undo something, the operation is stored in memory, in a stack-like arrangement. If you then select Redo the last undo operation that you did will be reversed, then the next-to-last, and so on.

Check in the Edit menu to confirm exactly what action will take place when you select Undo or Redo. While Advanced PCB has been optimized to ensure that Undo and Redo operations are not memory intensive, you can clear the Undo stack by temporarily setting the Stack Size to zero in the Preferences dialog box.

Design Objects

The Advanced PCB environment, whether creating and editing PCBs, or working in the library editor, consists of two basic features: *objects* that are placed in the workspace to build up the design, and *processes,* which are used by the system or the user to create, modify, save and report on the objects.

There are two types of objects in Advanced PCB, *Primitive Objects* and *Group Objects*. Primitive objects are the most basic elements in Advanced PCB and include; tracks, pads, vias, fills, arcs and strings. Anything that is made up of primitives and identified as a design object is a group object. Examples of group objects include; components, dimensions, coordinates and polygons.

Primitive Objects

Tracks

Tracks can be placed on any layer, with any width from 0.001 to 10000 mils wide. Tracks can be placed by pressing the Track button on the PlacementTools toolbar, by selecting the Place-Track menu item, or by the autorouter. Tracks are also used to generate polygon planes.

Track placement in Advanced PCB works differently from draw/paint applications where you "click-and-drag" to "stroke" a line. In Advanced PCB a single track can be made up of many track segments. A track is laid by placing each of these segments, by precisely "marking" the track's starting, corners and end point. This provides the accurate placement control required for PCB layout.

Tracks that carry either signals or supply power can be placed on:

- The Top (component side) signal layer.
- Any of the fourteen Mid-signal layers (number Mid Layer 1, etc.).
- The Bottom (solder side) signal layer.

"Non-electrical" tracks can also be placed on:

- The two Silkscreen overlays (normally used to draw component package outlines).
- Any of the four Internal Plane layers (creating voids in these solid copper planes).
- The Keep Out layer to define the board perimeter for autorouting and auto component placement.
- Any of the four Mechanical layers for mechanical details.
- Solder and Paste Mask layers for any special openings required in the masks.

Tracks are placed, edited, moved or deleted using the same methods, irrespective of the layer.

- ➡ When tracks (or any other primitives) are placed or moved they are always located on the current snap grid. As you move an object you will notice that it snaps from grid point to grid point.

Default Track

The default attributes for all design objects are set in the Defaults Tab of the Preferences dialog box.

- ➡ The attributes of the design object currently being placed can be set by pressing the Tab Key as soon as you select the place menu item or press the button on the PlacementTools toolbar. If the "Permanent" option is not set in the Defaults Tab of the Preferences dialog then changes made during placement will become the new defaults.

Placing Tracks

A track is a layer dependent object. To place track segments on the current layer:

1. Choose Place-Track (shortcut; P, T or click the Track button on the PlacementTools toolbar).

The prompt "Choose start location" is displayed on the Status Bar.

2. If you have not already done so, select the desired layer before you commence placing the track. You can toggle to the desired active signal layer by pressing * or toggle through all active layers using the + and – keys.
3. Click LEFT MOUSE (or press ENTER) once to define a start point for the track.

- ➡ Press TAB to change the track attributes during placement.

The Status Bar will display the net assigned to this track, the current segment length and the total track length in brackets.

4. Drag the track segment in any direction. Click LEFT MOUSE (or press ENTER) to end this first segment of the track.
5. Move the cursor to continue with a new track segment, which is extended from the existing track. Click LEFT MOUSE or press ENTER again to define this segment.

- ➡ If you make a mistake press BACKSPACE to remove the last track segment.

6. Click RIGHT MOUSE to end a series of connected track segments.

Note that "Choose start location" is still displayed on the Status Bar. This allows you to end one series of connected tracks and then begin a new series of track segments elsewhere in the workspace without having to choose Place-Track again.

7. To exit track placement, press ESC or RIGHT MOUSE a second time.

Track Placement Mode

Advanced PCB provides seven *track placement modes.* The mode specifies how corners will be created when placing tracks.

The track placement modes include:

Any Angle

Allows track to be placed at any angle.

90 Degree Horizontal Start and 90 Degree Horizontal End

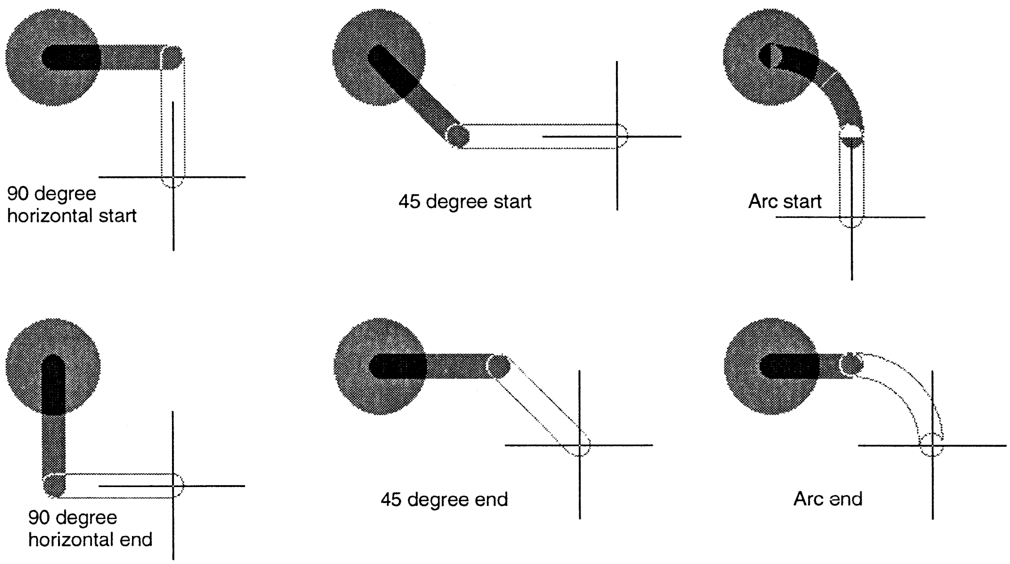
Constrains track placement to a horizontal and vertical orientation.

45 Degree Start and 45 Degree End

Constrains track placement to a 45 degree line and a horizontal/vertical line.

Arc Start and Arc End

Constrains track placement to an arc and a horizontal/vertical line.



Six of the seven possible track placement modes.

➡ Press the SPACEBAR to toggle between the Start and End placement modes while placing a track. Hold the SHIFT key and press the SPACEBAR to toggle through the four different types of track placement modes.

➡ The direction the arc renders is determined by the X/Y relationship. If the distance from the last vertex to the cursor is smaller in the X direction, the arc will render toward the X axis. If the distance is smaller in the Y direction, the arc will render toward the Y axis. Move the cursor to change the direction the arc renders.

Placing Tracks to Route a Connection

When you place a track that starts on an object with a net name, such as a pad, you are *routing* that net. The track you are placing will adopt the net name of the pad and the design rules that apply to that net will be observed. Placing tracks to route a net is supported by a number of features, such as the track placement modes, that simplify this task. For a complete discussion of these refer to the chapter *Routing Your Design.*

Changing Tracks

Tracks can be changed both individually and globally. Select the Edit-Change menu item to edit an existing track. Click on a track to pop up the Change Track dialog box. Editable track attributes include:

Width

Track width can be set in mils (.001 in) or mm. Range is 0.001 to 10000 mils.

Layer

Tracks can be assigned to any layer. Generally, tracks are placed on the Top, Bottom and Mid layers to carry signals or currents. Tracks are placed on other layers for non-electrical purposes, for example, on the Overlay (silkscreen) layers to indicate component outlines. Tracks are also used to create voids in layers which are normally plotted in reverse; power planes, solder and paste masks.

Net

The Net attribute of a primitive is used by the Design Rule Checker to ascertain if a primitive is legally placed. The net name is automatically assigned if the primitive is placed by the autorouter, during manual routing or if it is placed at the same location as a primitive that already has a net name.

Locked

As with all primitives, tracks can be locked in the workspace. Lock a track whose position is critical or which must not be affected by the autorouter.

Selection

Tracks can be selected or de-selected.

Coordinates

Track start/end x and y coordinates can be edited. These attributes are not globally editable.

➡ To edit the track attributes during placement press the TAB key to pop up the Change Track dialog box.

Pads

Pads can be either multi-layer or placed on any individual layer. For example, surface mount components and edge connectors have single layer pads on the Top and/or Bottom layers. Pads shapes include circular, rectangular, rounded rectangular or octagonal with X and Y size definable from 1 to 10000 mils. Hole size can range from 0 (SMD) to 1000 mils. Pads can be identified with a designator up to 4 characters long.

On a multi-layer pad the Top layer, Mid layers and Bottom layer pad shape and size can be independently assigned, to define a pad stack. Pad can be used individually as free pads or they can be incorporated with other primitives into components.

Default Pad

The default attributes for all design objects are set in the Defaults Tab of the Preferences dialog box.

➡ The attributes of the design object currently being placed can be set by pressing the Tab Key as soon as you select the place menu item or press the button on the PlacementTools toolbar. If the "Permanent" option is not set in the Defaults Tab of the Preferences dialog then changes made during placement will become the new defaults.

Placing Pads

Free pads (pads that are not grouped in a library component) can be placed anywhere in your design. Through-hole pads (and vias) are multi-layer objects which occupy each signal layer of the PCB and can place without regard to the current layer setting. Single layer pads can be placed on any layer.

To place a pad:

1. Choose Place-Pad (shortcut; P, P or click the Pad button on the PlacementTools toolbar).

Press TAB to change the default pad attributes during placement. When placing single layer pads you can change layers at any time by pressing * (to toggle active signal layers); + or – (to toggle up and down through all active layers). When placing a pad, the cursor cross hair is at the pad center.

Pad Designator

Pads can be labeled with a designator (usually representing a component pin number) of up to four alphanumeric characters. Spaces are not allowed but the designator can be left blank if desired.

Pad designators will auto-increment by 1 during placement if the initial pad has a numeric designator. To set the designator prior to placing the first pad, press the TAB key while the pad is floating on the cursor.

To achieve alpha or numeric increments other than 1, use the Paste Array feature. By setting the designator of the pad prior to copying it to the clipboard and setting the Text Increment field in the Paste Array dialog box, the following types of pad designator sequences can be placed;

numeric (1, 3, 5)

alphabetic (A, B, C)

combination of alpha and numeric (A1, A2, or 1A, 1B, or A1, B1 or 1A, 2A, etc.).

To increment numerically set the Text Increment field to the amount you wish to increment by. To increment alphabetically set the Text Increment field to the letter in the alphabet that represents the number of letters you wish to skip. For example, if the initial pad had a designator of 1A and the Text Increment field was set to C (the third letter of the alphabet), the pads would have the designators 1A, 1D (three letters after A), 1G (three letters after D), and so on.

Changing Pads

Both free pads and component pads can be individually and globally edited. To edit a pad select the Edit-Change menu item. The Status Bar will prompt "Change Any Object". Click on a pad to pop up the Change Pad dialog box. This dialog box includes three Tabs - Attributes, Pad Stack and Advanced.

Attributes Tab

This tab contains the most commonly edited attributes of a pad.

Use Pad Stack

Enable the Use Pad Stack option if you require a different pad shape on different layers. Once this is enabled the features on the Pad Stack Tab will be available.

X Size

Sets the size of the pad in the horizontal (X) direction. Range is 1–10000 mils.

Y Size

Sets the size of the pad in the vertical (Y) direction. Range is 1–10000 mils.

➡ The X size and Y size can be changed independently to define asymmetric pad shapes.

Shape

Pad shapes include Rounded, Rectangular, and Octagonal. Shapes can be manipulated by changing the X and Y size settings. A component can have any number of differently shaped pads.

Designator

Label for this pad, usually the component pin number. Free pads can include a designator or this value can be left empty.

Hole Size

This attribute specifies the diameter of the hole to be drilled in the pad during fabrication, in mils (.001 inch) or mm. For SMD pads or edge connectors this should be set to zero. The hole size can be set from 0 to 1000 mils and can be larger than the pad, for defining (copper free) mechanical holes.

➡ The hole size can be entered in units other than the current snap grid units. They will automatically be converted to the current units.

Layer

Pads can be assigned to any single artwork layer or to the Multi layer for through-hole components. SMD pads are typically assigned to either the Top layer or Bottom layer. To choose a new layer click the Layer button to scroll the selection bar through the available layers.

Rotation

Advanced PCB supports full pad rotation, with an angular resolution of 0.001 degrees.

X-Location

The X location of this pad, relative to the Current origin.

Y-Location

The Y location of this pad, relative to the Current origin.

Locked

As with all primitives, pads can be locked in the workspace. Lock a pad whose position is critical.

Selection

Pads can be selected or de-selected.

Pad Stack Tab

Multi-layer pads can have size and shape attributes individually assigned for the Top layer, Mid 1–14 layers and Bottom layer, creating a pad stack. Enable the Use Pad Stack option on the Attributes Tab if you wish to create a pad stack.

Advanced Tab

Net

The Net attribute of a primitive is used by the Design Rule Checker to ascertain if a primitive is legally placed. Used component pads will automatically be assigned a net name if a netlist is loaded.

Electrical Type

By default all pads have their status set to Load. The Source and Terminator settings are used when a net requires one of the Daisy chain topologies. Refer to *Net Topology* topic in the *Working With a Netlist* chapter and the Routing Topology Rule in the *Design Rules* chapter for further information on applying a topology to a net.

Plated Hole

Pads can be either plated or non-plated. If both types of pads exist in a design the non-plated holes will use different tools from the plated holes in the NC drill files. Refer to the *NC Drill* topic in the *Reports* chapter.

Vias

When tracks from two layers need to be connected, vias are placed to carry a signal from one layer to the other. Vias are like round pads, which are drilled and usually through-plated when the board is fabricated.

Vias are either multi-layer, blind or buried, and can be any diameter from 2 to 10000 mils wide. Vias can be placed using the Via button on the PlacementTools toolbar, by selecting the Place-Via menu item, by the auto via feature when placing tracks, or by the autorouter. The hole size can be set from 0 to 1000 mils.

Via Type

Advanced PCB vias can be multi-layer, blind or buried. A multi-layer via passes from the Top layer to the Bottom layer and allows connections to all other signal layers. A blind via connects from the surface of the board to an internal layer, a buried via connects from one internal layer to another internal layer.

Blind and Buried Vias

Before using blind or buried vias it is important to establish the level of support provided by the manufacturer. Most manufacturers support blind and buried vias between what are termed *layer pairs*. Using this technology, a multi-layer board is fabricated as a set of thin double sided boards which are then "sandwiched" together. This allows blind and buried vias to connect between the surfaces of these thin double sided boards, which become the layer pairs. In these circumstances blind and buried vias can only be used between the following layer pairs.

Top layer - to - Mid layer 1 (blind)

Mid layer 2 - to - Mid layer 3 (buried)

Mid layer 4 - to - Mid layer 5 (buried)

Mid layer 6 - to - Mid layer 7 (buried)

Mid layer 8 - to - Mid layer 9 (buried)

Mid layer 10 - to - Mid layer 11 (buried)

Mid layer 12 - to - Mid layer 13 (buried)

Mid layer 14 - to - Bottom layer (blind)

Advanced PCB also supports the use of blind and buried vias between any two layers. These must only be used if the PCB manufacturer supports the "build-up" fabrication technology required to manufacture these vias.

➡ Check that your board manufacturer supports the desired via technology before using blind or buried vias.

Default Via

The default attributes for all design objects are set in the Defaults Tab of the Preferences dialog box.

➡ The attributes of the design object currently being placed can be set by pressing the Tab Key as soon as you select the place menu item or press the button on the PlacementTools toolbar. If the "Permanent" option is not set in the Defaults Tab of the Preferences dialog then changes made during placement will become the new defaults.

Placing Vias

Vias can be placed by the Autorouter, by the Auto Via feature or manually.

Autorouter Vias

Define the autorouter via parameters by adding Routing Via Rules. Refer to the *Design Rules* chapter for more information on design rules.

Auto Via Feature

If the * key is used to toggle to another signal layer when a track is being placed a via is added automatically.

Manually Placed Vias

To manually place a via select the Place-Via menu item. The current default via will appear on the cursor. Click to place a via in the workspace.

Changing Vias

To edit a via, select the Edit-Change menu item and click on the via. The Change Via dialog box will pop up. This dialog box includes two Tabs - Attributes and Advanced.

Attributes Tab

Diameter

Vias can be any size in the range of 2 to 10000 mils.

Hole Size

Sets the hole diameter to be drilled at each via. The range is 0 to 1000 mils.

Layer Pair

Select the required layer pair. Only select the Blind & Buried (Any) option if you require blind and buried vias between *any* two layers. The start and end layers for Blind & Buried (Any) vias are specified in the Advanced Tab.

➡ Blind and buried vias should only be used in consultation with your board manufacturer.

X, Y Location

Via coordinates can be edited. These changes cannot be applied globally to other placed vias.

Locked

As with all primitives, vias can be locked in the workspace. Lock a via whose position is critical or which must not be affected by the autorouter.

Selection

Vias can be selected or de-selected.

Advanced Tab

Net

The Net attribute of a primitive is used by the Design Rule Checker to ascertain if a primitive is legally placed. The net name is automatically assigned if the primitive is placed by the autorouter, during manual routing or if it is placed at the same location as a primitive that already has a net name.

Start Layer

The Start Layer is the upper layer that the via starts on. Set the Layer Pair option in the Attributes Tab to Blind & Buried (Any) to use this feature.

End Layer

The End Layer is the lower layer that the via ends on. Set the Layer Pair option in the Attributes Tab to Blind & Buried (Any) to use this feature.

Fills

Fills (or *area fills*) are rectangles which can be placed on any layer. When placed on a signal layer they become areas of solid copper and can be used to provide shielding or to carry large currents. Fills of varying size can be combined to cover irregularly shaped areas. They can be combined with track or arc segments and be recognized as electrically connected when running the design rule check (DRC) feature.

Fills can also be placed on non-electrical layers. For example, place a fill on the Keep Out layer to designate a "no-go" area for both autorouting and auto component placement. Place a fill on a Power plane, Solder Mask, or Paste Mask layer to create a void on that layer.

Default Fill

The default attributes for all design objects are set in the Defaults Tab of the Preferences dialog box.

➡ The attributes of the design object currently being placed can be set by pressing the Tab Key as soon as you select the place menu item or press the button on the PlacementTools toolbar. If the "Permanent" option is not set in the Defaults Tab of the Preferences dialog then changes made during placement will become the new defaults.

Placing Fills

1. To Place a fill, select the Place-Fill menu item.

The Status Bar will prompt "Select First Corner".

2. Position the cursor and click LEFT MOUSE.

Drag the mouse to position the diagonally opposite corner.

3. Click LEFT MOUSE to define the second corner.

Continue placing fills or click RIGHT MOUSE to stop.

➡ A fill will "adopt" a net name if the first corner is placed on an object which has a net name.

Changing Fills

To edit a fill, select the Edit-Change menu item and click on the fill. The Change Fill dialog box will pop up.

Layer

The layer that this fill is currently placed on. Fills can be assigned to any layer in the workspace.

Net

The Net attribute of a primitive is used by the Design Rule Checker to ascertain if a primitive is legally placed. The net name is automatically assigned if the primitive is placed by the autorouter, during manual routing or if it is placed at the same location as a primitive that already has a net name.

Rotation

Advanced PCB supports full rotation of fills, with an angular resolution of 0.001 degrees.

➡ Note that Gerber generation does not support rotated fills. Rotated fills will be painted with a round aperture during Gerber generation.

Coordinates

Fill corner coordinates can be edited.

Locked

As with all primitives, fills can be locked in the workspace. Lock a fill whose position is critical.

Selection

Fills can be selected or de-selected.

Arcs

Arcs are essentially circular track segments. They can be placed on any layer with a radius between 0.001 to 16000 mil and width from 0.001 to 10000 mils wide. The angular resolution is 0.001 degrees. Arcs can be placed using the Arc button on the PlacementTools toolbar, Place-Arc process launchers or as part of a track using the Place Track process. Arcs are also used when generating polygon fills and by the autorouter.

Arcs have a variety of uses in PCB layout. For example, they can be used to indicate component shapes on the Overlay layers or on a mechanical layer to indicate the board outline, holes, etc. Arcs can be open, or closed to create a circle.

Arcs can also be placed on signal layers as part of a track. These arcs can be generated on-the-fly while placing tracks if the Track Placement Mode is set to Arc Start or Arc End. Press SHIFT+SPACEBAR during track placement to toggle through the four types of placement modes. Once you have selected the Arc mode, press the SPACEBAR to toggle between the Arc Start and Arc End modes.

Default Arc

The default attributes for all design objects are set in the Defaults Tab of the Preferences dialog box.

➡ The attributes of the design object currently being placed can be set by pressing the Tab Key as soon as you select the place menu item or press the button on the PlacementTools toolbar. If the "Permanent" option is not set in the Defaults Tab of the Preferences dialog then changes made during placement will become the new defaults.

Place Arc (Center)

To place an arc on the current layer using the arc center as the starting point:

1. Select Place-Arc (Center) (shortcut; P, A) or the Arc tool button.

The prompt "Select Arc Center" is displayed on the Status Bar. You can change layers at any time during this operation by pressing * (to toggle active signal layers); + or – (to toggle up and down through all active layers).

2. Position the cursor to set the center of the arc and click LEFT MOUSE.

As you move the mouse a highlighted arc will be displayed.

3. Position the cursor to set the radius and click LEFT MOUSE.
4. Position the cursor to define the start point of the arc and click LEFT MOUSE.
5. Position the cursor to define the end point of the arc and click LEFT MOUSE.

➡ To render the arc in the other direction, press the SPACEBAR before defining the end point.

If you are drawing a 360 degree arc, click to define the Start and End points without moving the cursor.

6. Start a new arc or press ESC or RIGHT MOUSE to stop placing arcs.

Place Arc (Edge)

To place an arc on the current layer using the arc edge as the starting point:

1. Select Place-Arc (Edge) (shortcut; P, E).
2. Position the cursor to set the start point of the arc and click LEFT MOUSE once.
3. Position the cursor to set the end point of the arc and click LEFT MOUSE again.

➡ To render the arc in the other direction, press the SPACEBAR before defining the end point.

Changing arcs

Arc attributes which can be edited include:

Width

Arc line width can be any value between 0.001–10000 mils.

Layer

Arcs can be assigned to any layer of the PCB. To change the layer assignment, click in the Layer box and choose a new layer.

Net

The Net attribute of a primitive is used by the Design Rule Checker to ascertain if a primitive is legally placed. The net name is automatically assigned if the primitive is placed by the autorouter, during manual routing or if it is placed at the same location as a primitive that already has a net name.

X and Y Center

Defines the arc center point.

Radius

Defines the arc radius.

Start and End Angle

Defines the angle from the zero degree point (3 o'clock on a watch dial) to each of the two end points on the arc.

Locked

As with all primitives, arcs can be locked in the workspace. Lock an arc whose position is critical.

Selection

Arcs can be globally selected or de-selected.

Strings

Text strings can be placed on any layer with any height from 0.010 to 10000 mils. Strings can placed with the Strings button (T) or by the selecting the Place-String menu item.

Place-String is used to place free text strings (up to 254 characters, including spaces) on any layer of your PCB. Text is generated using one of three special fonts. The Default style is a simple vector font which supports pen plotting and vector photoplotting. The Sans Serif and Serif fonts are more complex – and will slow down printing or plotting on vector devices. The fonts supplied with the system are built into the software and cannot be changed.

Free Text strings can be moved or edited like other primitives. Component text can be moved independently of the component (Edit-Move). If the component is moved, component text will move relative to the component.

Component text, (designators and comments) have the same font options and height and width range as free text and can be moved and edited in the same way. Free text can be placed on any layer. Component text is automatically assigned to the Top or Bottom Overlay layer when the component is placed, but can be moved to any layer.

All fonts can be plotted and photo plotted and will appear the same as on the screen. All fonts have the full IBM extended ASCII character set that supports English and other European languages.

➡ Strings in converted Protel Autotrax files will be converted to the Default font type. Note that the Advanced PCB Default font has wider characters than Autotrax strings.

Advanced PCB includes "Special Strings". These are strings which are interpreted when output is generated. Special strings are discussed below.

Default String

The default attributes for all design objects are set in the Defaults Tab of the Preferences dialog box.

➡ The attributes of the design object currently being placed can be set by pressing the Tab Key as soon as you select the place menu item or press the button on the PlacementTools toolbar. If the "Permanent" option is not set in the Defaults Tab of the Preferences dialog then changes made during placement will become the new defaults.

Placing Strings

To place a free string,

1. Select the Place-String menu item. The current default string will appear floating on the cursor.
2. Press the Tab key to pop up the Change String dialog box.
3. Type the string into the Text field or select one of the special strings from the drop down list.
4. Set the Height, Width and Font as required.
5. Click OK. The string will appear floating on the cursor. Click to place the string.

During placement you can change layers by pressing * (to toggle active signal layers); + or – (to toggle up and down through all active layers). The string can be mirrored along the X or Y axes by pressing the X or Y key and rotated by pressing the SPACEBAR.

Changing Strings

Free text strings and component text (designators and comments) can be changed both individually and globally. Editable text string attributes include:

Text

The text content of the string. Free text strings can be up to 255 characters long and any alphanumeric character, including spaces. Default string font, height and width are set in the Current Tab of the Preferences dialog box.

Height

Text size can be set in mils (.001 in) or mm. Range is 0.01 to 10000 mils. A minimum text height of 36 mils will be allow strings to be legibly photo plotted.

Width

The text stroke width can be set in mils (.001 in) or mm. Range is 0.001 to 255 mils.

Font

Three fonts are available: Default, Sans Serif and Serif. The default font is tailored for vector plotting.

Layer

Text strings can be assigned to any layer.

Rotation

Advanced PCB supports full rotation of strings, with an angular resolution of 0.001 degrees.

X, Y Location

String coordinates can be edited. These changes cannot be applied globally to other placed strings.

Mirror

Mirror flips the string for plotting in the correct orientation on bottom-side artwork layers. The string can also be mirrored along the X or Y axes by pressing the X or Y key when it is floating on the cursor.

Locked

As with all primitives, strings can be locked in the workspace. Lock a string whose position is critical.

Selection

Strings can be selected or de-selected.

Special Strings

Special strings allow you to place generic, non specific text which is interpreted when printing, plotting or generating Gerber files. For example, the string .PRINT_DATE will be replaced by the current date when output is generated. The available special strings are;

.PRINT_DATE

.PRINT_TIME

.PRINT_SCALE

.LAYER_NAME

.PCB_FILE_NAME

.PCB_FILE_NAME_NO_PATH

.PLOT_FILE_NAME

.ARC_COUNT

.COMPONENT_COUNT

.FILL_COUNT

.HOLE_COUNT

.NET_COUNT
.PAD_COUNT
.STRING_COUNT
.TRACK_COUNT
.VIA_COUNT
.DESIGNATOR
.COMMENT
.LEGEND
.NET_NAMES_ON_LAYER

The .DESIGNATOR and .COMMENT special strings are added to the component in the library. Use these if you need to control the location of these attributes on a component. They can be placed on any layer. The standard designator and comment can be hidden if desired.

Place the .LEGEND string on the Drill Drawing layer. It will be replaced by a drill table when the output is generated.

➡ To interpret special strings on screen, enable the Convert Special Strings option in the Display Tab of the Preferences dialog box. Note that not all special strings can be interpreted on screen.

Group Objects

A group object is any set of primitives which has been defined to behave as an object. These may be user defined, such as components and polygons, or system defined, such as coordinates and dimensions. A group can be manipulated as one object - they can be placed, selected, copied, changed, moved and deleted.

Polygons

Polygons are special areas of copper formed when you use the Place-Polygon Plane process. Polygon planes (or copper pours) are similar to area fills, except that they can fill irregularly shaped areas of a board and can connect to a specified net as they are poured.

Although polygons consist of tracks and arcs, polygons can be manipulated as a unit. Polygon boundaries can be re-shaped and polygons can be re-poured around new obstacles after placement, and any of their attributes, such as grid and track size, can be changed. By adjusting the grid and track size, a polygon plane can be either solid (copper) areas or a cross-hatched "lattice".

When placed in occupied board space polygon planes "pour" copper around any tracks, pads, vias, fills or text while maintaining the clearances specified in the design rules. If you are working with a netlist-based layout the plane can automatically connect to any component pads on the specified net that are within the polygon plane.

Polygons can be poured on any layer. Use a polygon to create a multi-sided shape on any layer. If a polygon is placed on a non-signal layer it will not be poured around existing objects as these objects are not assigned to a net and therefore do not "belong" to anything.

Placing a Polygon Plane

To place a polygon plane:

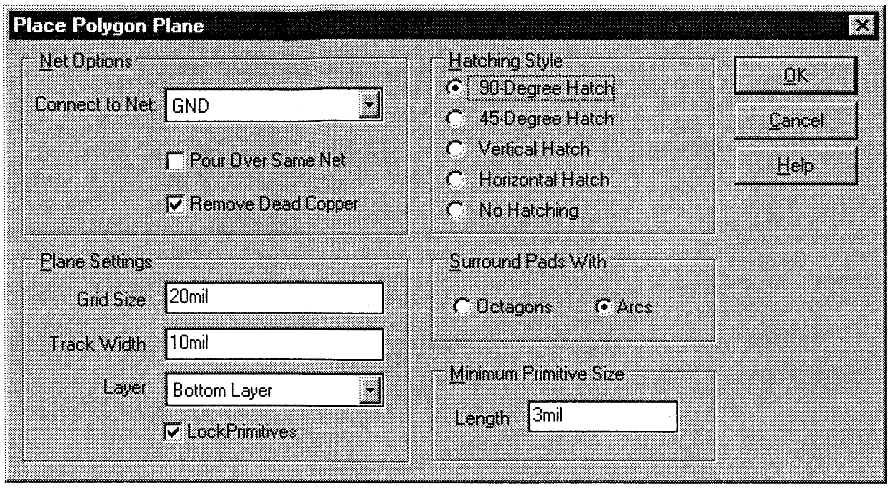
1. Choose Place-Polygon plane (shortcut; P, G).

The Place Polygon Plane dialog box will pop up. Set the attributes as required. Each attribute is defined below.

3. Click to define the starting point of the polygon.
4. Continue to click at each vertex of the polygon until the boundary of the polygon plane is defined. Press SHIFT+SPACEBAR to change track placement modes. Use the SPACEBAR to change between the Start and End placement modes.

The polygon will pour when it is closed. If you do not actually close the polygon, when you press ESC or RIGHT MOUSE the polygon will automatically be closed, from the last vertex to the initial vertex.

Options in the Place Polygon Plane dialog box include;



Net Options

Connect To Net

If a netlist has been loaded, one of the nets in the netlist can be selected in the Connect To Net drop down. If the polygon is being connected to a net, the other two Net Options can be applied.

Pour Over Same Net

If the Pour Over Same Net option is enabled any existing tracks within the polygon which are part of the net being connected to will be covered by the polygon.

Remove Dead Copper

Dead copper is copper placed by the Place Polygon Plane process which can not be connected to the selected net. Regions of dead copper are created when existing tracks, pads and vias prevent the plane pouring as one continuous area. These can be removed if desired. If this option is enabled and the polygon does not enclose a pin on the selected net, the entire polygon is removed as it is all dead copper.

Plane Settings

Grid Size

This is the grid on which the tracks within the polygon are placed. Ideally this grid is a fraction of the component pin pitch, to allow the most effective placement of the polygon tracks.

Track Width

This is the width of the tracks which are placed to form the polygon. If the track width is smaller than the grid size the polygon will be hatched. If the track size is equal to or greater than the grid size the polygon will be solid. For a solid plane set the track width slightly larger than the grid size.

Layer

This is the layer the polygon is to be placed on. Polygons can be placed on both copper and non-copper layers.

Hatching Style

90 Degree Hatch

Fill the polygon with tracks running both horizontally and vertically.

45 Degree Hatch

Fill the polygon with tracks running at 45 degrees (in both directions).

Vertical Hatch

Fill the polygon with tracks running vertically.

Horizontal Hatch

Fill the polygon with tracks running horizontally.

No Hatching

Do not place any tracks inside the polygon. Use this option if you want to place the polygon, but do not want it to slow system performance. It can be re-poured later with the desired hatching.

Surround Pads With

Pads can be surrounded with either Arcs or Octagons. Octagons give smaller Gerber files and faster photoplotting.

Minimum Primitive Size

Length

The length field allows you to limit the minimum size of primitives used in the polygon. When polygons are poured they can contain many short pieces of tracks and arcs, placed to create smooth shapes around the existing objects on the board. By limiting the length of primitives used you will get faster pour times, screen redraws and output generation. This will be at the expense of the smoothness of the polygon edges.

How the Polygon Connects to Pads

To control how a polygon connects to pads when the Connect To Net option is used, include a Polygon Connect Style design rule. This rule allows you to select between a direct connection and a thermal relief connection. It also allows you to set the conductor width and connection angle if you select relief connection. Refer to the *Design Rules* chapter for more information on the Polygon Connect Style design rule.

Re-pouring Polygons

To re-pour a polygon select the Edit-Change menu item. Click on the polygon you wish to re-pour and the Place Polygon Plane dialog box will pop up (shortcut; double click on the polygon). Change the attributes as desired and click OK. You will be asked if you wish to re-pour all modified polygons. Click No if you want to modify other polygons and would prefer to save re-pouring them until they have all been modified. Click Yes to immediately re-pour the modified polygon. The polygon will re-pour with the new settings.

Changing the Shape of the Polygon Boundary

The boundary of a polygon plane can be reshaped by moving the vertices. To move the polygon vertices;

1. Select the Edit-Move-Polygon Vertices menu item.

The Status Bar will prompt “Choose a Polygon”.

2. Click on the polygon to be edited.

The internal polygon tracks will disappear, leaving only the boundary tracks.

3. Click on the vertex (editing handle) to be moved. The vertex can then be repositioned.
4. Continue to reposition the vertices.
5. Click RIGHT MOUSE or press ESC when finished.

You will be asked if you wish to re-pour all modified polygons. Click No if you want to modify other polygons and would prefer to save re-pouring them until they have all been modified. Click Yes to immediately re-pour the modified polygon.

Dimensions

Dimensions are special entities consisting of text and track segments. They are automatically generated when you indicate the starting and ending points after choosing the Place-Dimension menu item.

One of the first tasks many designers will want to perform when starting a design is to define the dimensional details for the board. Advanced PCB provides a convenient auto-dimensioning feature to make the process both highly-accurate and easy. Imperial or metric units will be calculated, depending upon the current Snap grid setting.

Default Dimension

The default attributes for all design objects are set in the Defaults Tab of the Preferences dialog box.

➡ The attributes of the design object currently being placed can be set by pressing the Tab Key as soon as you select the place menu item or press the button on the PlacementTools toolbar. If the "Permanent" option is not set in the Defaults Tab of the Preferences dialog then changes made during placement will become the new defaults.

Placing Dimensions

1. Select the Place-Dimension menu item.
2. Click to define the start point.

The Status Bar will prompt "Select Measure End Point".

3. Click to define the end point.

The dimensioning information will then be placed.

Changing Dimensions

Dimension attributes such as string height, font etc. can be changed during placement (press the Tab key), or after the dimension has been placed. Select the Edit-Change menu item and click on the dimension to pop up the Change Dimension dialog box. Editable dimension attributes include:

Height

Height of the dimension end lines.

Line Width

Width of all the lines in the dimension.

Unit Style

Units displayed are the current snap grid units. The units can be hidden, displayed (e.g. 1200 mil) or displayed in brackets (e.g. 210 (mm)).

Text Height

Text size can be set in mils (.001 in) or mm. Range is 0.01 to 10000 mils. A minimum text height of 36 mils will be allow strings to be legibly photoplotted.

Text Width

The text stroke width can be set in mils (.001 in) or mm. Range is 0.001 to 255 mils.

Font

Three fonts are available: Default, Sans Serif and Serif. The default font is tailored for vector plotting.

Layer

Dimensions can be placed on any layer.

Start X, Y

Start X, Y location.

End X, Y

End X, Y location.

Locked

Dimensions can be locked in the workspace. Lock a dimension that must not be moved or adjusted.

Selection

Dimensions can be globally selected or de-selected.

Moving a Dimension

Advanced PCB allows dimensions to be moved, or "adjusted", after they have been placed. To move or adjust a dimension, click once anywhere on the dimension to bring it into focus (the square focus handles will appear). A second click anywhere on the dimension moves the dimension, a second click on a focus handle allows you to adjust the dimension. If you re-size a dimension the distance will be automatically updated.

Coordinates

Use a coordinate marker to indicate the coordinates of a specific point in the workspace. A coordinate marker includes a point marker (cross made of two tracks) and the X, Y coordinates of the position. They can be placed on any layer.

Default Coordinate

The default attributes for all design objects are set in the Defaults Tab of the Preferences dialog box.

➡ The attributes of the design object currently being placed can be set by pressing the Tab Key as soon as you select the place menu item or press the button on the PlacementTools toolbar. If the "Permanent" option is not set in the Defaults Tab of the Preferences dialog then changes made during placement will become the new defaults.

Placing Coordinates

1. Select the Place-Coordinate menu item.

The coordinate will appear floating on the cursor.

2. Click at the location which requires a coordinate marker.
3. Continue to place coordinates or click RIGHT MOUSE to stop.

Changing Coordinates

Coordinate attributes such as string height, font etc. can be changed during placement (press the Tab key) or after the dimension has been placed. Select the Edit-Change menu item and click on the coordinate to pop up the Change Coordinate dialog box. Editable coordinate attributes include:

Size

Size of the coordinate marker lines.

Line Width

Width of the coordinate marker lines.

Unit Style

Units displayed are the current snap grid units. The units can be hidden, displayed (e.g. 1220, 3400 mil) or displayed in brackets (e.g. 1220, 3400 (mm)).

Text Height

Text size can be set in mils (.001 in) or mm. Range is 0.01 to 10000 mils. A minimum text height of 36 mils will be allow strings to be legibly photoplotted.

Text Width

The text stroke width can be set in mils (.001 in) or mm. Range is 0.001 to 255 mils.

Font

Three fonts are available: Default, Sans Serif and Serif. The default font is tailored for vector plotting.

Layer

Coordinates can be placed on any layer.

X, Y Location

X, Y location of the coordinate marker.

Locked

Coordinates can be locked in the workspace. Lock a Coordinates that must not be moved.

Selection

Coordinates can be globally selected or de-selected.

Moving Coordinates

To move a coordinate marker and its coordinate values, simply click and hold on the marker or the value and drag to a new location.

Components and Libraries

A comprehensive library of over 300 predefined through hole and SMD component footprints is included with Advanced PCB. PCB footprints are created and modified in the PCB Library Editor, the second document editor provided with Advanced PCB. Refer to the chapter *PCB Library Editor* for information on using this document editor to create component footprints and libraries.

In the Advanced PCB Environment, a "footprint" is what exists in a PCB library. When this footprint is placed in the workspace, it is assigned a designator (and optional comment). It then is referred to as a component.

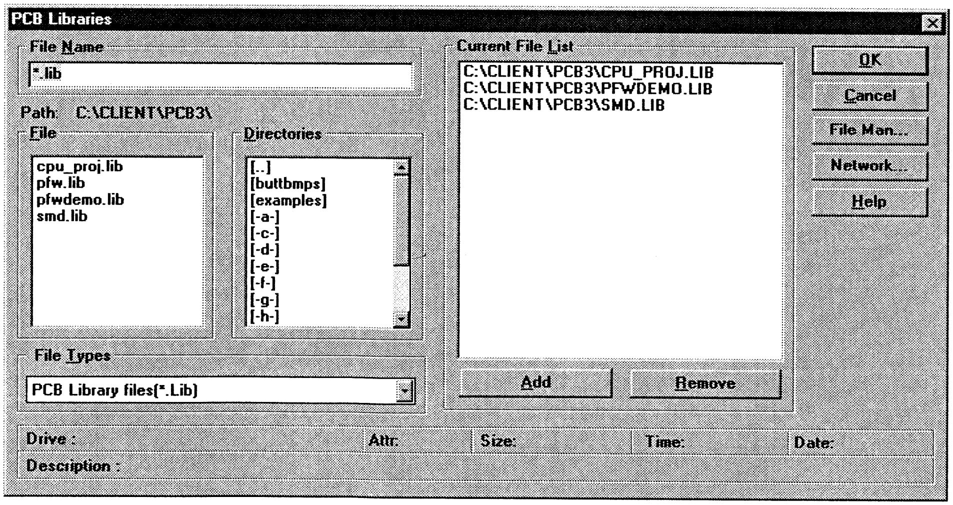
Accessing Component Footprints

To access the component footprints in libraries, the libraries must first be added to the *current library list* in the PCB Editor. Libraries are added and removed by selecting Add/Remove Library in the Design menu or pressing the Add/Remove button in the PCB Editor Panel (when the Browse mode is set to Libraries). This pops up the PCB Libraries dialog box, where new libraries can be added and open ones removed from the Current File List.

Once a library has been added, footprints from that library can be placed in the workspace. The only limit to the number of libraries that can be added is the memory available in your computer.

Adding and Removing Libraries

Selecting Add/Remove Library pops up the Change Library File List dialog box.



This dialog box includes the following features and options.

File Name

Enter the name of the library to be Added to the library list.

File

This window lists files in the current directory that match the current file mask used in the File Name window.

File Types

Choose from a pre-defined mask for loading library files. You can use this feature to specify the file type, by extension. Advanced PCB does not restrict the use of extensions to identify library types.

Directories

Double click in this window to change the current path and directory as you search for the desired library.

Current File List

This window lists all currently loaded libraries and is updated as libraries are added or removed from the list.

To add a library to the current list:

1. Move the selection bar through the files listed in the Files window to highlight the desired library;
2. Click the Add button. The library will be added to the Current File List window.

To remove a library from the current list:

1. Move the selection bar through the Current File List to highlight the desired library;
2. Click Remove;
3. Click OK to close the Library List window and re-set the updated library list.

Finding and Placing Components

To browse the components in a library, set the Browse mode in the PCB Editor Panel to Libraries and click on the desired library in the list. When you click on a component in the Components list it will be displayed in the MiniViewer.

Placing in the PCB Editor

To place a component footprint;

1. Set the Panel Browse mode to Libraries.
2. Select the desired library from the list.
3. Select the component from the list in the Panel and press the Place button (shortcut: double click LEFT MOUSE).

The Component will appear floating on the cursor.

4. Press the Tab key to edit the designator and comment prior to placing the component, then click OK to return to the component floating on the cursor.

Press PAGEUP to zoom in. Press the SPACEBAR to rotate and the L shortcut key to flip the component to the bottom of the board. Use the Jump shortcut keys to jump to an exact location.

5. Click LEFT MOUSE to place the component.

Placing from the PCB Library Editor

Components can also be placed from the PCB Library Editor. The Library Editor Panel has a Place button. Select the component from the list and press Place. The component will be placed in the last active PCB window. Placement then follows the same sequence as placing from within the PCB Editor.

All About Components

Once a footprint is placed in the workspace, it becomes a component. It has a designator, such as R3 and a comment, say 10K. It also behaves as a single object rather than a collection of tracks, pads, arcs etc. and can be moved, flipped and rotated as such. To edit a component, select the Edit-Change menu item or double click inside the component outline. The Change Component dialog box will pop up.

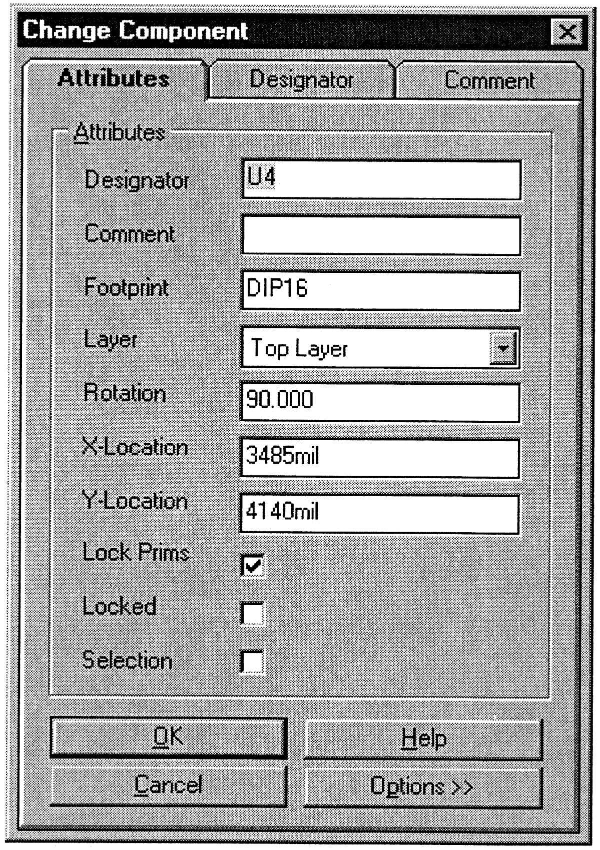
The Change Component dialog box is divided into three Tabs. Component attributes which can be edited include:

Attributes Tab

This Tab contains the commonly used component attributes.

Designator/Comment

The value of the component designator and comment can be changed here. Use the designator and comment Tabs to change the text attributes such as the font, text height and text width.



Footprint

The current footprint of the component can be changed to any other available footprint in any open component library. If you type the name of a different footprint in the Footprint field, when you exit the dialog box Advanced PCB will search the current open libraries to try and locate the new footprint.

Layer

Components can be assigned to either the top or bottom layer of the PCB. To change the layer assignment, click in the Layer box and choose Top layer or Bottom layer.

Changing the layer status

swaps the component to the opposite layer. For example, when moving a Top layer component to the Bottom layer, primitives on the Top Overlay layer will be automatically reassigned to the Bottom Overlay layer. The orientation of the component will be flipped along the x axis and the component overlay text will read from the bottom. Single layer pads are also swapped between the Top layer and Bottom layer. You can extend this to do global swaps of components from one layer to another.

Rotation

Components can be rotated to any angle, with an angular resolution of 0.001 degrees.

Lock Prims

Normally it is desirable to manipulate a component as one object. In this state the primitives that make up a component are "locked" together. However, if required you can unlock the primitives and modify or manipulate them. Component primitives should then be re-locked. Component pad attributes can be modified without un-locking the primitives.

Locked

The Locked attribute determines whether a component is fixed in the workspace or is free to move. If the Locked attribute is set the auto placer will not move the component. If you attempt to manually move the component the warning message "Object is locked, continue?" will pop up, allowing you to move the component without unlocking it. The locked attribute remains set after this move.

Selection

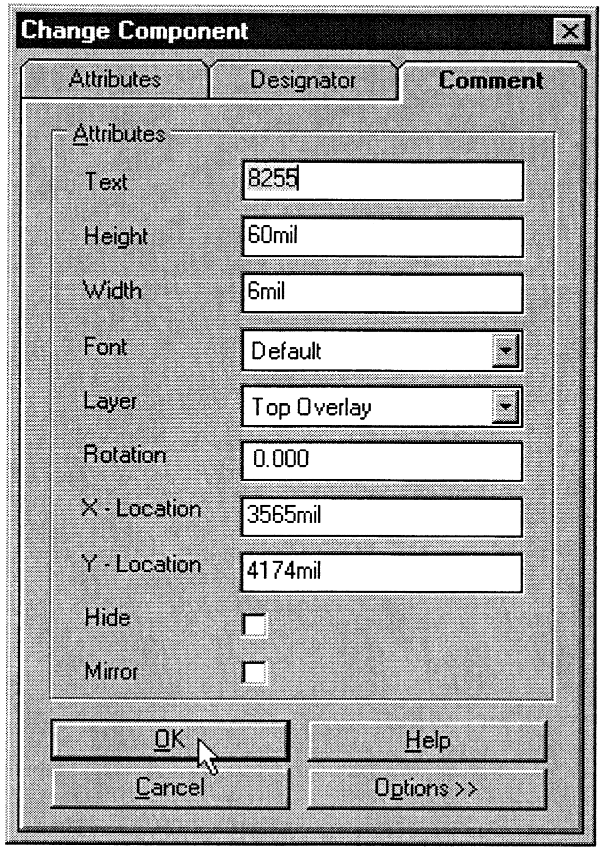
Components can be selected or de-selected. Use this attribute to help qualify components for a global edit operation.

Designator and Comment Tabs

These two Tabs are identical, containing the text attributes for the designator and comment.

Text

The component designator or comment. The designator attribute is not globally editable as each component must have a unique designator. The Comment attribute is globally editable. Designator and comment strings can be a maximum of 255 characters in length.



Height

Text size can be set in mils (.001 in) or mm. Range is 0.01 to 10000 mils. The character width used to display / print the text is automatically proportioned to the height. A minimum text height of 36 mils will allow strings to be legibly photoplotted.

Width

The text stroke width can be set in mils (.001 in) or mm. Range is 0.001 to 255 mils.

Font

Three fonts are available. Click the Font button to choose the Default font, Sans Serif font or Serif font.

Layer

Component text can be assigned to any layer. Click the Layer button to scroll the selection bar through the layer options. The selected layer will be displayed in the Layer box.

Rotation

Component text can be moved and rotated independently of the component. Click and hold on the component text and drag the mouse to reposition the text. Press the SPACEBAR to rotate the text while it is floating on the cursor.

X, Y Location

Location of the text in the workspace, relative to the current (relative) origin.

Hide

Component text can be displayed or hidden. Hidden text will not be printed.

Mirror

Component text can be mirrored independently of the component.

➡ To flip a component during placement, press the L key while the component is floating on the cursor. This will mirror the footprint, convert top layer pads to bottom layer pads and mirror the overlay onto the bottom overlay layer. Do not use the X or Y keys, as these will flip the component, but not change its layer.

Changing a Component Footprint

To change an existing component from one footprint to another, double click inside the component outline to pop up the Change Component dialog box. Type the new footprint name into the Footprint field of the Attributes Tab.

The current component footprint can be changed to any other available footprint in any open library. When you type a different name in the Footprint field and click OK to exit the dialog box Advanced PCB will search the current open libraries and attempt to locate the new footprint.

Component footprints can be changed freely. However, if there are netlist connections to the pads the new footprint must have the same used pin numbers available as the previous one. If it does not the warning message "cannot match pads with new footprint" will be displayed and the substitution will be aborted. For example, changing a DIP16 to an SMD16A is a legitimate change as the pin numbers match. Changing a DIP16 to a TO-3 would generate a warning and the change would be aborted. If the change is successful the connection lines will also be updated to remain connected to the appropriate pads.

Modifying an Individual Component on the Board

Generally, if a component footprint requires modification the footprint is edited in the library and then the PCB is updated. This will update all instances of that footprint in all currently open designs.

Advanced PCB also allows for the modification of components on the board. To modify a component on the board, double click inside the component outline to pop up the Change Component dialog box. Select the Attributes Tab. Un-check the Lock Prims check box and click OK to close the dialog box. The component primitives can now be modified or manipulated. The component primitives should then be re-locked. Component pad attributes can be modified without un-locking the primitives.

Un-Grouping a Component

If necessary, a placed component can be converted back into the original set of primitive parts. Select the Tools-Convert-UnGroup Component menu item. When you launch this process you will be prompted "Select Component". The prompt "Confirm convert Component To Primitives" will be displayed. If you click YES (or press y) the component designator and comment (if any) will be removed from the component and it will become a set of primitives. This is a one way process, it is not possible to re-group an un-grouped component.

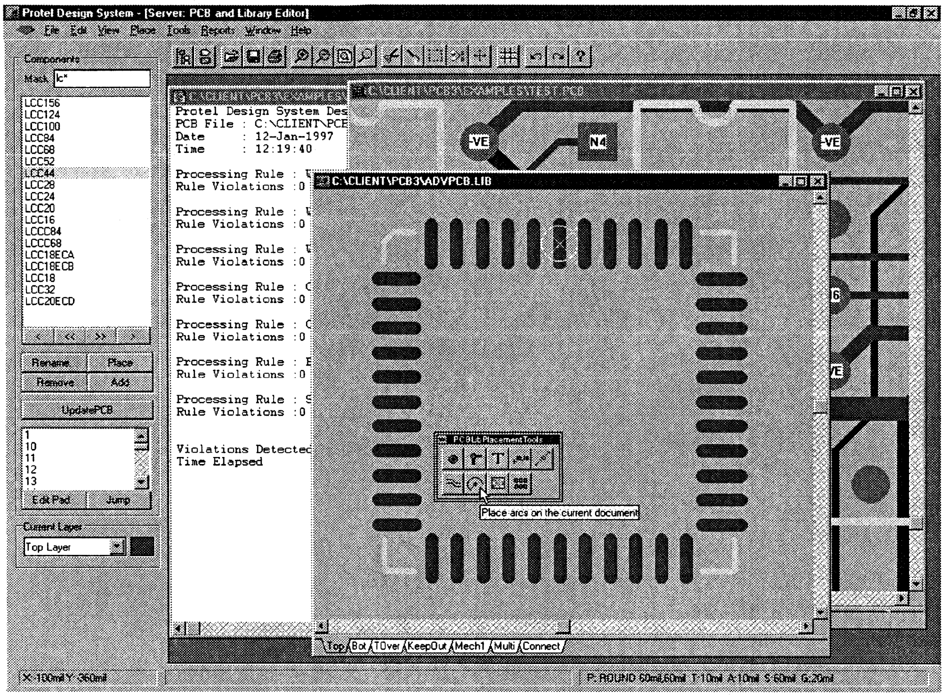
➡ Un-group has no effect on the component footprint stored in the library – only on the individual instance of the component placed in the document window.

Creating a Project Library

Advanced PCB can create a library of all components currently placed in the workspace. This allows the creation of Advanced PCB libraries from imported PADS, PCAD or Tango files, as well as from Advanced PCB designs.

To use this feature select the Design-Make Project Library menu item. Enter the library name in the Save Project Library dialog box and click OK. All components in the workspace will be added to the library.

Library Editor



The Library Editor is the second Document Editor provided by the Advanced PCB Server. Where the PCB Editor is used to design the printed circuit board, the Library Editor is used to create and modify the components used on those PCBs. It is also used to manage the PCB libraries.

The Library Editor includes a complete set of processes for creating, editing and placing library footprints. Custom libraries can be created and any number of component libraries can be opened at the same time, limited only by available memory. There is no limit to the number of component footprints that each library can hold.

Components generally include one or more pads (corresponding to component pins and numbered accordingly) plus track and/or arc segments on the overlay (silkscreen) layer to define the component body.

Opening a Library

Libraries are opened in the Library Editor in the same way documents are opened in all servers in EDA/Client, by selecting the File-Open menu item. Set the Editor to PCBLib in the Open Document dialog box, then locate and open the library. The only limit on the number of libraries you can have open is the memory available in your computer. Each open library will appear in a separate document window.

Creating a Library

To create a new library, select the File-New menu item. The Select EDA Document Type dialog box will pop up, select PCBLib from the list. A library will be opened with the name PCBLIB_1.LIB. As a library is a set of components, it can not exist without at least one component. So when you create a new library, an empty component sheet PCBCOMPONENT_1 will be presented. To save this library with a name of your choice, select the File-Save As menu item. To rename the component with a name of your choice, select the Tools-Rename Component menu item.

Creating a Component Footprint with the Component Wizard

Advanced PCB includes a powerful component building Wizard. This Wizard will ask a few questions and then build the component footprint for you, from a simple two pin resistor through to a Pin Grid Array with hundreds of pins.

To launch the Component Wizard press the Add button on the Library Editor panel or select the New Component menu item in the Tools menu. If the Wizard does not launch then you will need to install the CompMake Wizard server. For tips on installing a server refer to chapter *A Quick Tour of EDA/Client.*

Manually Creating a Component Footprint

Footprints are created in the PCB Library Editor using the same set of design objects available in the PCB Editor. Anything can be saved as a PCB footprint, including corner markers, phototool targets, mechanical definitions, and so on.

The typical sequence for manually creating a component footprint is;

1. New component - Open the desired library in the Library Editor. Select the Tools-New Component menu item. The Component Wizard will automatically start, press cancel to manually create a component. You will be presented with an empty component footprint workspace, called PCBComponent_1. Select Tools-Rename Component to change this to the required name. Component names can be up to 255 characters.

2. Place the pads - place the pads according to the component requirements. When a pad is floating on the cursor select Edit-Jump-Reference (shortcut; J, R) to position the cursor at the workspace 0, 0 coordinate. Prior to placing the first pad, press the Tab key to define all the pad attributes.

➡ Always build surface mount footprints on the top layer. Use the L shortcut key to flip them to the bottom layer during placement.

3. Component outline - Use the track tool to create the component outline on the Top Overlay layer. Use the SPACEBAR to change between the Start and End placement modes. Press SHIFT+SPACEBAR to change track placement modes.

4. Save and place - save the library. Components can then be placed in the PCB Editor directly from the library open in the Library Editor, using the Place button on the Panel. Refer to the chapter *Components and Libraries* for information on placing components in the PCB Editor.

➡ Always build the component around the workspace 0,0 reference point. The Reference is the point you will be "holding" the component by when you place it in the PCB Editor. Use one of the Reference options in the Edit menu to move the Reference if it needs to be changed.

➡ Designator and comment special strings - the special strings .DESIGNATOR and .COMMENT can be added to the component in the Library Editor if you require control over their layer, location and text attributes prior to placing the component. These will be in addition to the standard designator and comment which can be hidden if desired.

Updating a Footprint

After editing a footprint in the PCB Library Editor, use the Update PCB button in the PCB Library Editor Panel to update all instances of this footprint in all open designs.

Copying a Footprint

Components can be copied within a library or between libraries via the clipboard. Refer to the *Editing* topic in the chapter *Working in Advanced PCB* for tips on using the clipboard.

Defining the Board

The first step in designing your PCB is to define the board. The board definition could include a mechanical outline, keep out boundaries, dimension detail, photo tool targets and other company and fabrication specific information.

➡ If the board has been defined in a mechanical package, such as AutoCAD, you can transfer it into Advanced PCB using the Import DXF feature. Refer to the *Import Options* chapter for more information.

Everything you design in the Advanced PCB workspace is done with the set of design objects found in the Place menu. The board keep out boundary is created with tracks and arcs on the Keep Out layer, the mechanical boundary is created with tracks and arcs on one of the mechanical layers, dimensions are added with the dimension tool, and so on.

It is good practice to design the board in the lower left region of the workspace. One inch in, one inch up from the *absolute origin* is often used as a position for the lower left corner of the board. The *current origin* can then be set as required.

The Board Wizard

Advanced PCB includes a board Wizard, which allows you to select from a large number of industry-standard board templates. The templates include; a title block, alignment markers, reference rulers, dimensions, and standard edge connectors. The Wizard will fill the title block and let you to specify the number of routing layers and the track/pad technology.

You can also add your own company templates to the Board Wizard. For your templates to appear in the Wizard you will need;

- A *MyTemplates*.BDL file. This is an ASCII file which lists the boards of this type. Refer to one of the existing BDL files for an example of the structure.
- A 64x32 pixel bitmap to appear in the Wizard.
- A standard PCB file for each of the templates of this type.

Store the BDL, bitmap and PCB template files in the C:\Client\Wizards\Pcbboard\ directory.

If the PCBMaker Wizard Server is installed it will automatically be launched when you select File-New to create a new PCB document. If it is not automatically launched refer to the chapter *A Quick Tour of EDA/Client* for information on installing a Server.

Tips on Using Keep Outs

The placement and routing outer limits are defined by creating a boundary on the Keep Out layer. Typically this boundary is set slightly in from the physical edge of the board, ensuring that tracks and components do not get too close to the edge of the board. Create this boundary by placing tracks and arcs on the Keep Out layer. All components and tracks should then be placed within this boundary. This boundary is used by the Design Rule Checker, the auto placer and the autorouter.

You can also define any other "no go" regions where components and/or tracks are to be excluded. This can include zones for mounting hardware and regions required for board profiling. These zones are also created by placing design objects such as tracks, arcs and fills on the Keep Out layer. The basic rule to using the Keep Out layer is - routing on signal layers will not cross over design objects on the Keep Out layer.

➡ Keep out regions defined on the Keep Out Layer apply to all signal layers.

Mechanical Definition

The detail required for the mechanical definition will depend on company and manufacturer requirements. Generally, manufacturers require board corner markers, a reference hole location and external dimensions as a minimum. Contact your PCB manufacturer for details.

Use the mechanical layers to create the mechanical definition. The contents of any of the four mechanical layers can be added to any other layer during output generation. Refer to the *Generating Output* chapter for details.

Working With a Netlist

Electronic Design Automation tools provide the platform on which an electronic design can move from the conceptual stage in the designer's mind, right through to producing the files required to manufacture the printed circuit board. By capturing the design electronically, the power and efficiency of a computer can be harnessed to transfer the design rapidly and accurately through the various design phases.

Through these phases the design is held in a number of forms. It begins as a schematic, a collection of components which are "wired" together. It ends as a printed circuit board, manufactured from a set of files produced from the PCB design tool.

To transfer the design between the schematic capture software and the PCB layout software a *Netlist* is used. Like both the schematic and the PCB, the netlist contains information about the components used in the design; their designation, value and physical package. It also contains the connectivity created in the design, stored in the form of *nets*. A net is a list of component pins that are electrically connected. As well as component details and connective information, a netlist can also contain other design information including simulation data and PCB design parameters.

The purpose of Advanced PCB is to translate this component and connective information in the netlist into a "physical" layout with "physical" connectivity. This includes; placing the components to meet any mechanical constraints and translating each logical netlist connection into a physical connection, fulfilling the electrical parameters required for that connection.

Component placement is a fundamental part of the PCB design process with many issues to consider such as; routing, mechanical packaging, board assembly, thermal performance, and so on. Refer to the *Component Placement* chapter for information on placement strategies.

About Netlists

Netlists come in many different formats, but are usually generated as ASCII text files which carry at least two types of information:

1. Descriptions of the components in the design.
2. A list of all pin-to-pin connections in the design.

Some netlist formats combine both sets of data in a single description, Others, including Protel, separate the two data into separate sections.

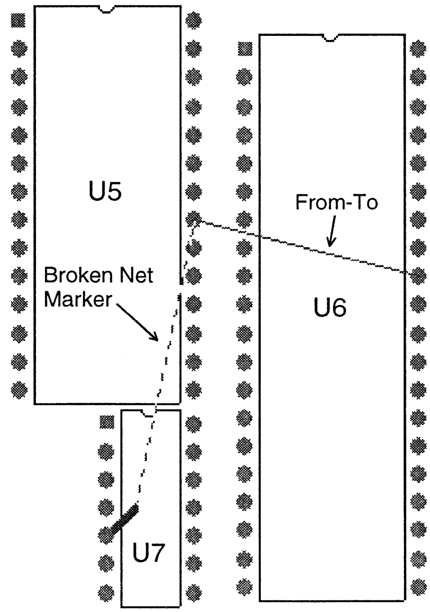
As straightforward text files, netlists are readily translated into other formats using a simple, user-written program. Netlists can also be created (or modified) manually using a simple text editor or word processor.

How the Netlist Connectivity is Displayed

When you load the netlist Advanced PCB displays the pin-to-pin connections in each net as a series of thin lines. The line that connects each pin in the net to another pin in the net is called a *From-To*, going *From* one pin in the net *To* another pin. The From-Tos are collectively referred to as the *Ratsnest.*

The pattern or arrangement of the From-Tos in a net is called the *net topology*. If a net has not been assigned a user-defined topology then Advanced PCB arranges the From-Tos to give the shortest possible connection distances for the entire net, based on the current arrangement of the components.

Once you commence routing From one pin To another pin the thin line changes from a solid line to a dashed line. This dashed line is called a *Broken Net Marker* and it indicates that this net is broken, or incomplete.



A net with one connection unrouted and the other partially routed.

Loading the Netlist

There are two reasons you will need to load a netlist. The first is when a netlist is loaded to commence the board design, the second is when a netlist is re-loaded to bring forward design changes from the schematic. This second process is known as *forward annotation.*

To load a netlist select the Design-Netlist menu item. The Load/Forward Annotate Netlist dialog box will pop up. Press the Browse button, locate and select the netlist and click OK in the Load Netlist dialog box.

Advanced PCB will then analyze both the netlist and any PCB design data present in the workspace. For each difference detected between the netlist and the existing design data Advanced PCB will create a *Netlist Macro*. This Macro tells Advanced PCB what action must be performed to update the design data to match the netlist.

If you are loading a netlist for the first time, Netlist Macros will be created for the entire netlist. If you are forward annotating your design, Netlist Macros are created for each design change.

Examine the Macros, and when you are satisfied that they will carry out the actions you require, press the Execute button.

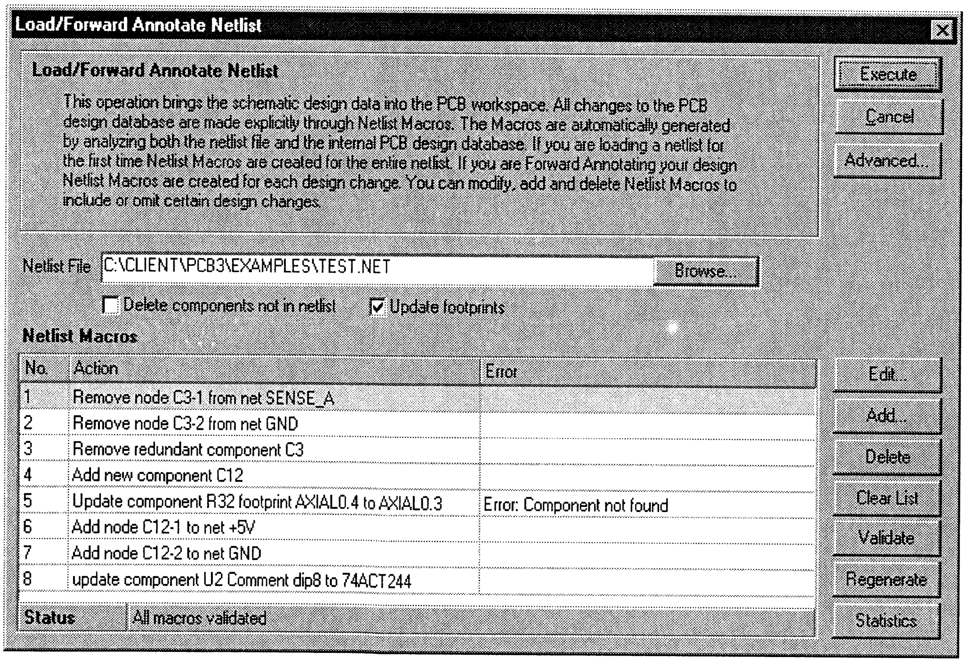
➡ It is good practice to resolve any Macro errors prior to Executing the Macros. Refer to the topic *Resolving Netlist Macro Errors* later in this chapter for tips.

➡ Ensure the required component libraries have been added to the current library list prior to loading a netlist (Design-Add/Remove Library).

Working with Netlist Macros

After Advanced PCB has analyzed the netlist and any PCB design data present in the workspace and created the Netlist Macros, they are listed in the Load/Forward Annotate Netlist dialog box, in the order that they will be executed. The Macro Commands include;

- Remove Node
- Remove Net
- Remove Component
- Add Component
- Add Net
- Change Net Name
- Change Component Footprint
- Change Component Designator
- Add Node
- Change Component Comment



Using Netlist Macros offers a number of distinct advantages. As well as allowing you to examine exactly what will happen when you load a netlist, you can also Add, Edit and Delete Macros, giving you total control over the changes to the PCB.

Adding, Editing and Deleting Netlist Macros

Use these options when you wish to omit or include certain design changes. You can also use them when you wish to modify the PCB netlist, without returning to the schematic. Points to be aware of when editing Netlist Macros include;

- Be consistent with the case of all text, such as Designators and Nets.
- A netlist node is specified as the *ComponentDesignator-PinNumber*. An example is J3-2. As per the PCB Library Editor requirements, the pin number can have a maximum of four alpha/numeric characters with no spaces.
- The footprint must match the name of a footprint in one of the libraries in the current library list.
- Refresh the screen after executing the Netlist Macros.

Validating Netlist Macros

After editing Macros you should always Validate them. This is particularly important if you have manually created Macros. Pressing this button instructs Advanced PCB to examine each Macro, check if it can be executed, and report any error condition.

Regenerating Netlist Macros

When you press Regenerate Advanced PCB will clear all existing Macros, then re-analyze the netlist and the PCB design data and create new Macros.

Resolving Netlist Macro Errors

Prior to executing the Netlist Macros it is good practice to resolve any errors or warnings. Following is a description of each error/warning. The description includes which Macro can report that error and what has caused the error.

Net not found

A Netlist Macro is attempting to: add or remove a node; remove a net; or change a net name when that net can not be found in the PCB netlist.

Component not found

A Netlist Macro is attempting to: add or remove a node when the component designator is incorrectly specified in the Macro or the component can not be found in the PCB netlist; remove a component; or change a footprint, designator or comment when the component can not be found in the PCB netlist.

Node Not found

A Netlist Macro is attempting to: add or remove a node from a component which does not have that pin; or remove a node which does not exist in the specified net.

Net already exists

A Netlist Macro is attempting to: add a net name when a net with that name already exists in the PCB netlist.

Component already exists

A Netlist Macro is attempting to: add a component when a component with that designator already exists in the PCB netlist.

New footprint not matching old footprint

A Netlist Macro is attempting to: change a component footprint when the *used* pins on the old footprint do not match the *used* pins on the new footprint. This can occur if the new component has fewer pins than the old, or if the pin numbering in the netlist (which comes from the schematic component pin numbers) is different to the pin numbering on the PCB component.

Footprint not found in Library

A Netlist Macro is attempting to: add a new component or change a component footprint when the specified footprint could not be found in any of the libraries in the current library list and no alternative library reference could be found in the Cross Reference file (ADVPCB.XRF).

Alternative footprint used instead (warning)

A Netlist Macro is attempting to: add a new component or change a component footprint when the specified footprint could not be found in any of the libraries in the current library list. An alternative library reference was found in the Cross Reference file (ADVPCB.XRF) and this component will be loaded from one of the libraries in the current library list. Always confirm that the alternative footprint is appropriate before executing a Macro with this warning.

➡ When a Netlist Macro attempts to load or change a component footprint which can not be located in any of the libraries, it then uses the component Comment to look-up the Cross Reference file (ADVPCB.XRF). The Cross Reference file lists components by their type against any appropriate footprint(s) for that component. For example, if the component U1 was a 74LS00, but you had forgotten to include the footprint, when the Macro to add this component was validated it would look-up 74LS00 in the XRF file. 74LS00 has DIP14 as a footprint, which would be loaded from one of the libraries in the current library list.

Summary

Most problems with loading a schematic netlist generally fall into two categories.

1. Component footprints - missing components occur when: a footprint is missing from the component information in the netlist; you have forgotten to add the required PCB libraries to the current library list (Design-Add/Remove Library); or the footprint in the netlist does not match any Advanced PCB library component.

2. New footprint not matching old footprint - the cause is usually that the pin numbering on the schematic component differs from the pin numbering on the PCB footprint.

Schematic libraries contain specific components and devices. The Advanced PCB component library contains generic footprints which can belong to various specific components – each having different pin assignments.

For example, a transistor shape can represent various combinations of "E," "B" and "C," – each of which must be assigned to the correct pin number in Advanced PCB. Diodes are a similar case, with pins often named "A" and "K" in the schematic.

You will need to either; modify the PCB footprint pin numbers to match the Schematic pin numbers, or change the schematic component pin numbers to match the PCB footprint.

Net Topology

When your netlist has loaded, the pin-to-pin connections are displayed for each net. The arrangement, or pattern of the pin-to-pin connections is called the net topology. By default Advanced PCB arranges the pin-to-pin connections of each net to give the shortest overall connection length (this topology is called Shortest). A different topology can then be applied to a net.

The topology of a net can be re-defined for a variety of reasons. High speed designs require that signal reflections must be minimized. To achieve this the high speed nets are arranged with a daisy chain topology, where all the pins are connected one after the other, with the source pin at one end and a terminator pin at the other end of the chain. Another requirement of your design may be that all ground pins in the ground net connect back to a common point. A star topology could be applied to the ground net to ensure this.

Specifying Net Topology

There are two techniques to apply a particular topology to a net in Advanced PCB. The first is at the net level and the second is at the individual pin-to-pin connection, or From-To level. If your design contains nets that require a certain topology, apply a Routing Topology design rule to those particular nets. Refer to the *Design Rules* chapter for further information about the Routing Topology design rule. If you wish to manually specify part or all of the topology of a particular net, then define From-Tos for that part of the net you wish to control the topology of.

From-Tos

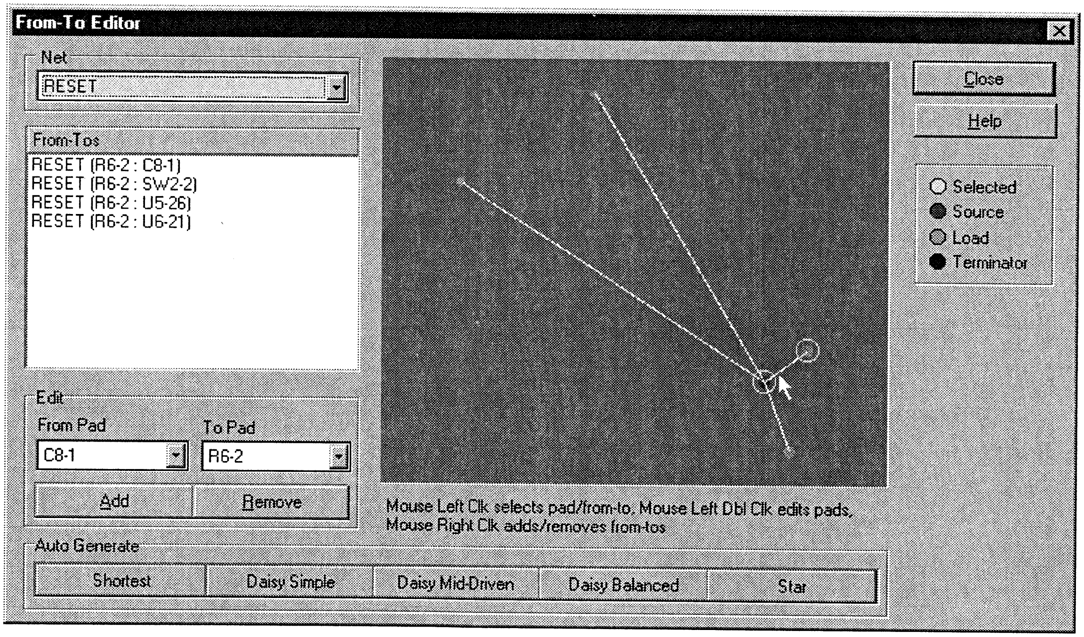
To give you total control of the arrangement, or pattern of the pin-to-pin connections in a net, Advanced PCB allows you to define your own set of From-Tos. A From-To instructs Advanced PCB, "I want to connect *From* this pin *To* that pin".

You can define one From-To for a net, a few From-Tos for a critical part of the net, or specify the entire topology of the net by defining From-Tos for all the pin-to-pin connections. If you create From-Tos for only part of a net Advanced PCB will set the remaining pin-to-pin connections to the shortest topology.

As well as using From-Tos to create a specific net topology, From-Tos can also be used as the scope for a design rule. The scope of a design rule designates exactly what the rule is to apply to. Using a From-To as the scope allows you to apply a rule to an individual pin-to-pin connection. This gives you total control over how rules apply to a net. You could specify that a net be routed at 25 mils, except for one From-To which you want to have routed at 40 mils. Refer to the *Design Rules* chapter for more information about design rules and their scope.

Creating From-Tos

To specify From-Tos for a net select the Design-From-To Editor menu item. The From-To Editor will be displayed.



Create and remove From-Tos in the From-To Editor.

Select the Net you wish to specify From-Tos for. All the pins in this net are displayed in the graphical window to the right. Any existing From-Tos are displayed as a thin line connecting the two pins in the graphical window, and they are also listed below the net name. Below the graphical window there are tips on how to quickly add and remove From-Tos.

Auto-Generated From-Tos

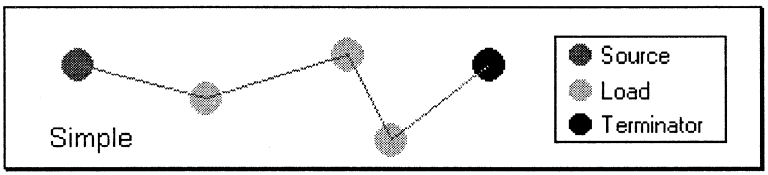
To quickly create a set of From-Tos for the entire net, use the Auto-Generate buttons at the bottom of the From-To Editor. These buttons create a set of From-Tos for the entire net, arranged in that particular topology.

Shortest

By default Advanced PCB arranges the pin-to-pin connections in the net to give the shortest overall connection distance. Pressing this button will remove any user or auto-generated From-Tos, instructing Advanced PCB to arrange the pin-to-pin connections with the shortest topology.

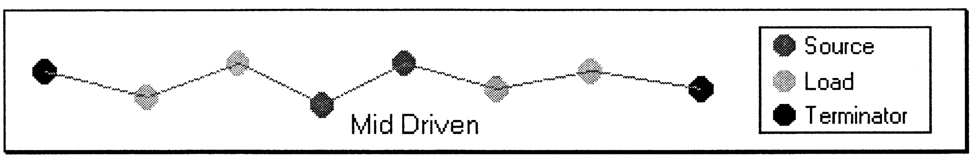
Daisy-Simple

In a simple daisy chain topology all the nodes (pins) are chained together, one after the other. The order they are chained is calculated to give the shortest overall length. If a source and terminator pad are specified then all other pads are chained between them to give the shortest possible length. Edit the pad (double click on it) to set it to be a source or terminator. If multiple sources (or terminators) are specified they are chained together at each end.



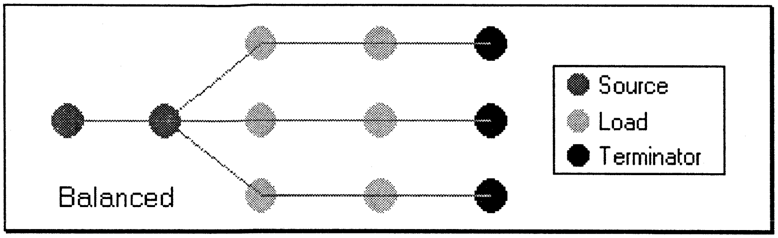
Daisy-Mid Driven

In a mid driven daisy chain topology the source node(s) are placed in the center of the daisy chain and the loads are divided equally and chained off either side of the source(s). Two terminators are required, one for each end. Multiple source nodes are chained together in the center. If there are not exactly two terminators a simple daisy topology is used.



Daisy-Balanced

In a balanced daisy chain topology all the loads are divided into equal chains, the total number of chains equal to the number of terminators. These chains then connect to the source in a star pattern. Multiple source nodes are chained together.



Star

This topology connects each node directly to the source node. If terminators are present they are connected after each load node. Multiple source nodes are chained together, as in the balanced daisy topology.

Displaying Pin-to-Pin Connections

To make working with the ratsnest more manageable Advanced PCB allows you to selectively show and hide the pin-to-pin connection lines. Select the View-Connections sub-menu;

Net

Show/Hide the entire set of pin-to-pin connections for the selected net. When you choose this option, a cross hair cursor appears. If you know the location of a pad on the net, click on that pad. If you do not, click in free space and a dialog will pop up, prompting for the net name. If you are unsure of the net name type *?* and click OK to list all loaded nets.

Component Nets

Show/Hide the entire set of pin-to-pin connections for all nets which connect to the selected component.

All

Show/Hide all currently loaded (unrouted) connections.

Changing Net Attributes

Like pads, tracks, vias and other design objects, each net has a set of attributes. To edit the attributes of a net set the Browse mode of the PCB Panel to Nets, select the net and press the Edit button. The Change Net dialog box will pop up, allowing you to edit the attributes of the selected net. The editable attributes include;

Color

Each net will be assigned the default net color when the netlist is loaded. The color for this net can be changed here.

Hide

Selectively hide the connection lines for this net, or combine with a global edit to hide the connection lines for a number of nets.

➡ The Layers Tab of the Document Options dialog box includes a check box to turn the Connect layer on or off. If this is off no connections are shown, regardless of the Hide attribute of each net.

Globally Editing Loaded Nets

Net attributes can be globally changed using global change options. You can match nets to be globally edited by using wildcards. For example, you can apply changes using the wildcard "D*" which will copy the changes to nets with name "D1," "D2," etc.

Identifying Nets

Use the PCB:IdentifyNet process to identify any net. Position the cursor over the connection line and press ENTER or LEFT MOUSE. The name of the net will be displayed on the Status Bar.

Exporting the Netlist

For information on how to export the netlist refer to the *Export Options* chapter.

Engineering Change Orders

Changes you make to the PCB that affect the netlist are written to an Engineering Change Orders file *filename*.ECO. Enable the ECO feature and specify the ECO text file name in the Options Tab of the Preferences dialog box.

Changes that are recorded in the ECO file include: creating a new net, renaming a net, adding a node to a net, deleting a node from a net, adding a component, deleting a component, changing a component footprint pattern and re-designating a component.

Protel's .ECO file format is fully-compatible with PADS .ECO files.

Netlist Formats

Protel Netlist Format

The first part of a Protel netlist describes each component:

[	Marks the start of each component description.
U8	The Component Designator (label).
DIP16	The Package Description (footprint). A footprint with this name must be available in the open PCB libraries.

74LS138	Part Type, (or comment).
(blank)	These 3 lines are not used.
(blank)	
(blank)	
]	Marks the end of the component description.

The second part of a Protel netlist describes each net:

(	Marks the start of each net.
CLK	Name of the net.
U8-3	First component (by designator) and pin number. Pin numbers in Advanced PCB library footprints must be an exact match.
J21-1	Indicates the second node in the net.
U5-5	Third node.
)	Marks the end of the net.

Protel Netlist 2.0 Format

This format is similar to the standard Protel netlist, with the addition of several fields that include schematic part fields (used for documentation and simulation) plus layout directives which provide net attributes. Advanced PCB version 2.0 or later load this format.

PROTEL NETLIST 2.0	The netlist header.
[	Begin component delimiter.
DESIGNATOR	(Each field is first named)
U1	Component designator.
FOOTPRINT	
DIP20	Library pattern (footprint).
PARTTYPE	
AmPAL16L8	Part Type field (when placed).
DESCRIPTION	
Description	Description field from schematic.
PART FIELD 1	(Field name can be defined in schematic)
Part Field 1	Part fields (1-16) from schematic.
(etc.)	
LIBRARYFIELD1	
Library Part Field 1	Library fields (1-8) from schematic lib.
]	End component delimiter.
(	Begin net delimiter.
VCC	Net name.
U1-20 AMPAL16L8-VCC POWER	
	First node in net.
	Includes: Component -pin designator.
	(single blank space)

	Part type-Pin name.
	(single blank space)
	Pin electrical type.
U2-14 4001-VCC POWER	
	Last component-pin node in net.
)	End net delimiter.
{	Begin Layout Directive delimiter.
TRACK	(Each field is first named).
10	Size of tracks (mils).
VIA	
50	Diameter of vias (mils).
NET TOPOLOGY	
SHORTEST	Net Topology for routing.
ROUTING PRIORITY	
MEDIUM	Routing priority.
LAYER	
UNDEFINED	Routing layer.
}	End Layout Directive delimiter.

Netlist Parameters

Designators and Package Descriptions (Type) are limited to 12 alphanumeric characters. Part Types can be up to 32 characters long. Net names can be 20 characters. Pin numbers in netlists are limited to 4 alphanumeric characters. No blank spaces may be used within these strings.

Any number of components or nodes can be included in a Advanced PCB netlist, limited only by available memory.

Other Netlist Formats

The Protel Hierarchical netlist format has the same information structure as the Protel 2 format, except that the component and net details for each sheet are recorded separately in the netlist file.

Netlists from schematic capture packages (other than Protel) usually have many similarities to the Protel format. However, the order in which component or net information is displayed may vary, and package names (e.g. DIP16), component designators and Pin identifiers may require editing to match Advanced PCB field restrictions. Often, translation of the netlist is an option in the schematic package. Netlists created using either a Protel or "Tango" output option will usually be fully compatible with Advanced PCB.

Design Rules

PCB design is no longer a matter of simply placing tracks to create connections. High speed logic combined with smaller and more complex packaging technologies place new demands on the PCB designer. It is no longer possible to satisfy all the design requirements by only considering the clearance between tracks, pads and vias. Today's designs can also require that you apply specific requirements to individual nets, components or regions of the board as well as considering such issues as crosstalk, reflections and net lengths. To successfully complete designs such as these you need a tool that can be configured to monitor and test for these requirements.

Advanced PCB incorporates a large set of design rules. These include clearances, object geometry, parallelism, impedance control, routing priority and routing topology. Each rule has a *Rule Scope* that defines how it is applied. The scope allows you to apply a rule to objects, nets, net classes, components, component classes, layers, regions, through to the whole board.

What are Design Rules?

You design your PCB by placing components, tracks, vias and other design objects. These objects must be placed in the workspace with regard to each other. Components must not overlap, nets must not short, power nets must be kept clear of signal nets, different nets must be routed at different widths, certain nets must have equal lengths, and so on.

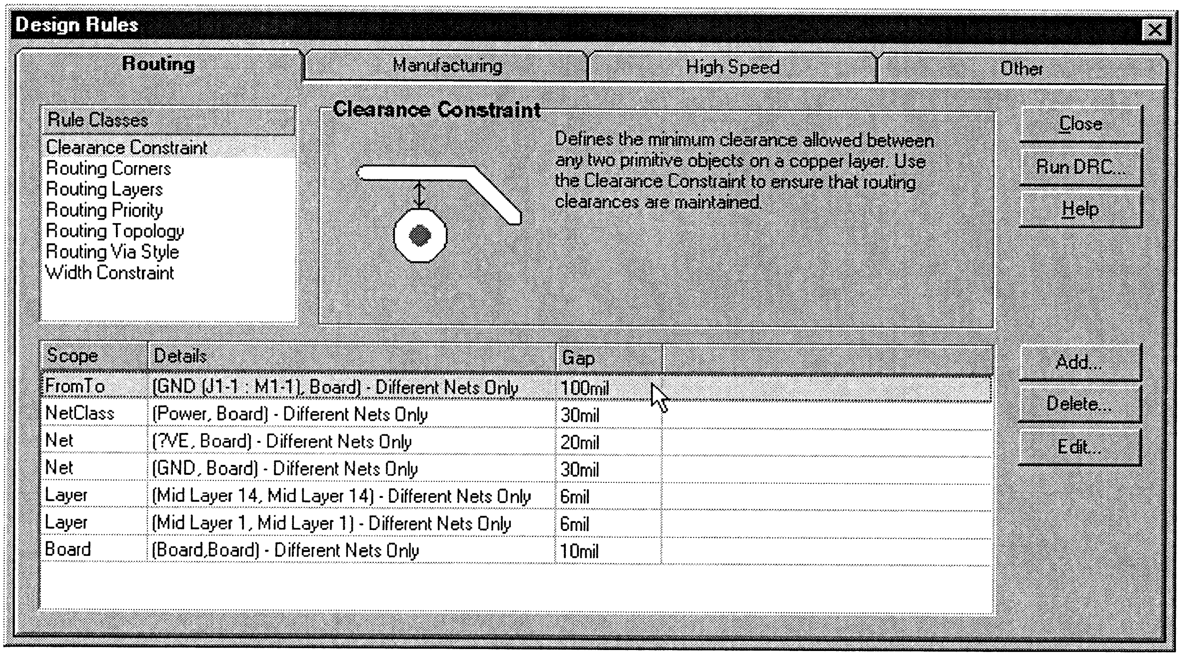
To allow you to remain focused on the task of designing the board, Advanced PCB can monitor these design requirements for you. You instruct Advanced PCB of your requirements by setting up a series of design rules. Advanced PCB monitors the placement of objects in the workspace and as soon as an object is placed in violation of a design rule it is highlighted.

Defining the Design Rules

Where are Rules Defined?

Rules are listed, added and edited in the Design Rules dialog box. Select the Design-Rules menu item to pop up this dialog box.

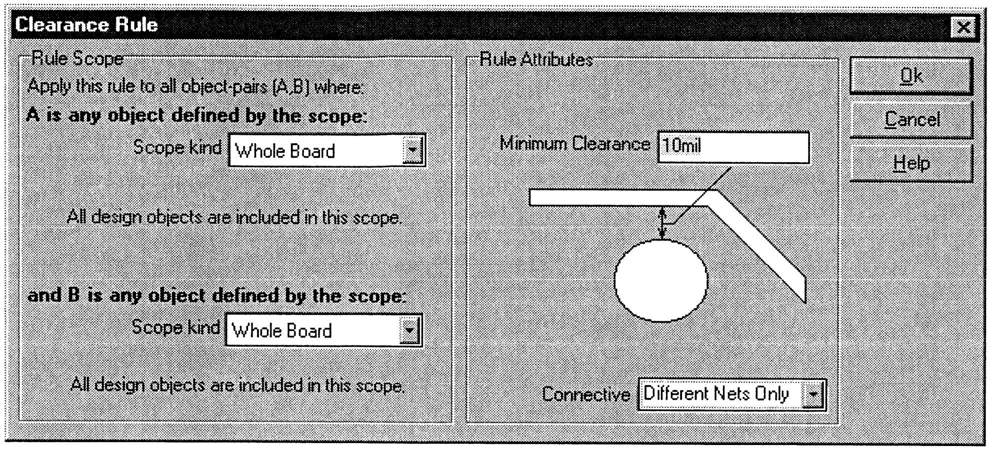
The set of rules available in Advanced PCB is divided into four groups and each group has a separate Tab in the Design Rules dialog box. On each Tab there is a set of rules listed on the left. Next to this list is a description of the currently selected rule. Position the cursor and click to select a different rule. If there are any instances of the selected rule already defined (including new PCB defaults) they are listed in the lower half of the dialog box.



Examine and edit the rules in the Design Rules dialog box.

Adding a Rule

Locate and select the rule you require in the Design Rules dialog box and press the Add button (shortcut; double click LEFT MOUSE on the rule). After pressing the Add button a dialog box for that particular rule will pop up. This is where you configure the rule and set the scope.



Each rule is configured in its Rule dialog box.

What is the Rule Scope?

The scope, or extent of each Rule is determined by the Rule Scope. The scope allows you to define the set of target objects that a particular instance of a rule is to be applied to. By setting the scope you could apply a rule to the whole board, or you could target a particular net, component or pad.

For example, your design might include mains level voltages requiring a clearance of 100 mils, and logic level voltages requiring a clearance of 10 mils.

These requirements can be achieved by defining two copper clearance rules, both with the rule scope set to region. One of the rules has the clearance set to 100 mils, with the region set to cover the area of the board containing the higher voltages; the other rule has the clearance set to 10 mils and the region set to cover the remainder of the board.

Unary and Binary Rules, and Setting Their Scope

There are two types of design rules, *unary rules* and *binary rules*. Unary rules apply to one object, or each object in a set of objects. Binary rules apply *between* two objects, or between any object in one set to any object in the second set. An example of a unary rule is the solder mask expansion rule. This rule applies individually to each pad identified by the rule scope. An example of a binary rule is copper clearances, which applies between any copper object in the first set and any copper object in the second set, as identified by the two rule scopes. When you configure a unary rule you setup one rule scope, when you configure a binary rule you setup two rule scopes.

Multiple Rules of the Same Kind and their Order of Precedence

The Rule Scope allows you to identify exactly what you want to apply a particular rule to. For example, you can apply a clearance constraint to a net, to a region of the board or to an individual pad.

As well as using the scope to define the set of objects that you want to apply the rule to, you can also use the scope to override one rule with another rule of the same kind.

Each rule can be applied as many times as you require. For example, you can apply a solder mask expansion rule to the whole board, apply a second solder mask expansion rule to a particular component and apply a third solder mask expansion rule to an individual pad on the same component. So that Advanced PCB knows which of these three rules to apply to this pad, there is an *order of precedence* for rules of the same kind which have different scopes. The order of precedence for rule scopes from highest priority to lowest is;

- Region (highest priority)
- Pad
- From-To
- From-To Class
- Net
- Net Class
- Component
- Component Class
- Object Kind
- Layer
- Whole Board (lowest priority)

This allows you to use a “most-general-to-most-specific” strategy when you apply rules. Apply general rules to the whole board, then build up your requirements by applying more specific rules with a higher precedence scope. Advanced PCB will analyze the rules applied to each design object and identify and apply only the rule with the highest precedence.

➡ So that you do not have to remember the order of precedence, the rule scopes are always ordered from highest priority to lowest priority in the drop down list boxes.

Contentions Due to Duplicate Rules

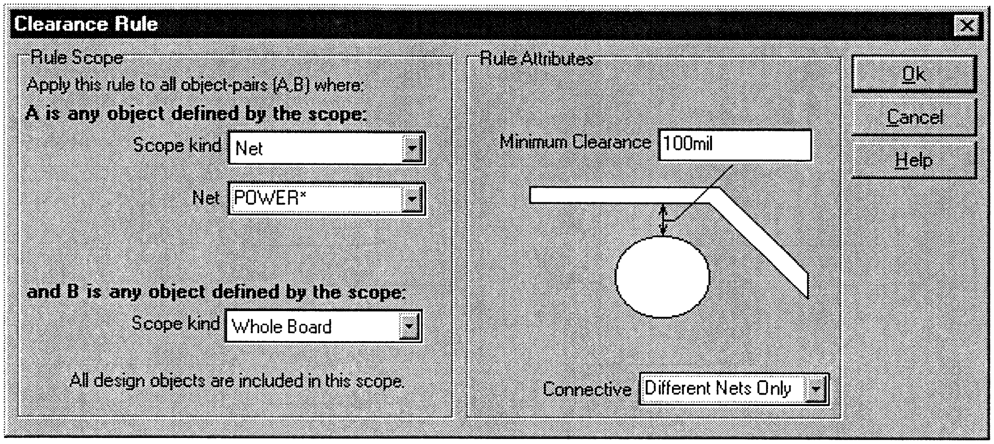
When an object is covered by more than one rule with the same scope (for example a pad covered by two solder mask expansion rules, both with the scope set to region, but whose regions overlap) a *contention* exists. Advanced PCB has pre-defined strategies to resolve each possible contention. The basic approach is to err on the side of safety. How this is interpreted for each rule is document with each rule description.

Strategies for Setting the Rule Scope

There are many ways of using the rules to satisfy your design requirements. Often you will want to identify a set of objects to apply a rule to. Net classes, component classes and from-to classes allow you to create user-defined sets of objects.

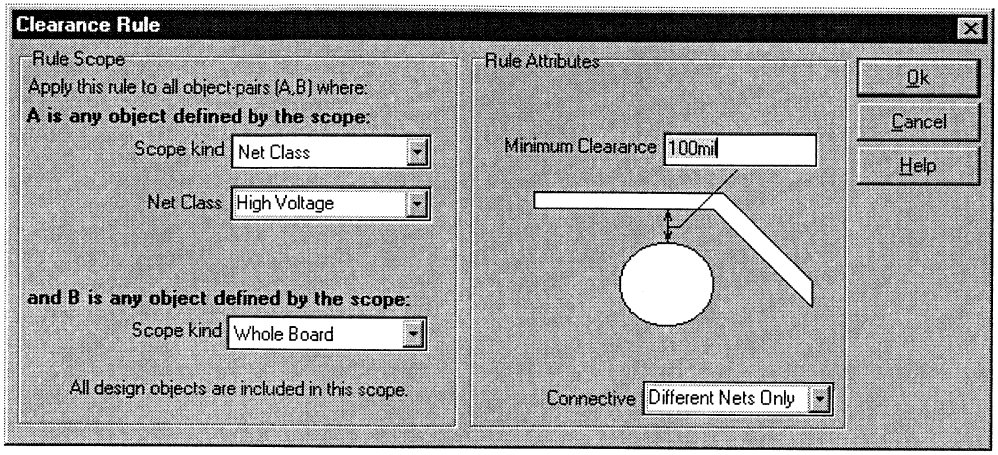
Wildcards can also be used to define a set of objects. Both the any single character (?) wildcard and the any characters (*) wildcard are supported.

Recall our earlier requirement to keep higher level voltage nets separate from the signal nets, which could be satisfied by two clearance rules with the scope set to region. The following figures shows two alternative approaches, the first using the Net rule scope and wildcards, the second using the Net Class rule scope. Note that the Net rule scope example assumes that all the high voltage nets have a net name starting with the string POWER.



Using a wildcard to identify a set of nets.

This clearance rule then specifies that "any net whose name starts with the string POWER must be at least 100 mils away from any other copper object on the Whole Board". As well as ensuring that the clearance is maintained between any POWER* net and any signal net, this rule also ensures that any POWER* net is at least 100 mils away from any other POWER* net.



Using a Net Class to identify a set of nets.

This clearance rule specifies that "any net in the net class HIGH VOLTAGE must be at least 100 mils away from any other copper object on the Whole Board". As well as ensuring that the clearance is maintained between any net in the HIGH VOLTAGE net class and any signal net, this rule also ensures that any HIGH VOLTAGE net is at least 100 mils away from any other HIGH VOLTAGE net.

How Rules are Applied

Advanced PCB applies the rules in the following situations;

On-line Design Rule Check (DRC)

A violation of the rule is flagged as soon as the violation occurs during placement. It is flagged by outlining the objects in violation in the current DRC color. The On-line DRC feature can be disabled in the Options Tab of the Preferences dialog box.

Batch DRC

Selecting the Tools-Design Rule Check menu item will pop up the Chose Rule Set to Check dialog box. Enable those Rule Types you wish to test and press the OK button. All instances of the enabled rule types will be tested.

Note that you can set the number of violations to report. Use this to keep the reports manageable.

During a software operation

Certain rules are monitored during a software operation including; polygon pour, autorouting, auto placement and output generation. Examples of these include; the mask expansion rule which is monitored during output generation and the routing via style rule which is monitored during autorouting.

Exporting the Design

Certain rules are included to support features in Protel's Advanced Route and the Specctra™ autorouters. The requirements specified by these rules are exported with the design.

Each Rule Definition specifies when that particular rule is applied.

Rule Definitions

Acute Angle Constraint

Definition

Specifies the minimum angle permitted at a track corner. Acute angles can be a problem when manufacturing, resulting in over-etching of the copper at the corner.

Setting the Scope

Apply to the Whole Board, to a Layer, to a Net Class, to a Net, to a From-To Class, to a From-To or a Region.

How Duplicate Rule Contentions are Resolved

The rule with the largest angle is obeyed.

Rule Application

Batch DRC.

Copper Clearance Constraint

Definition

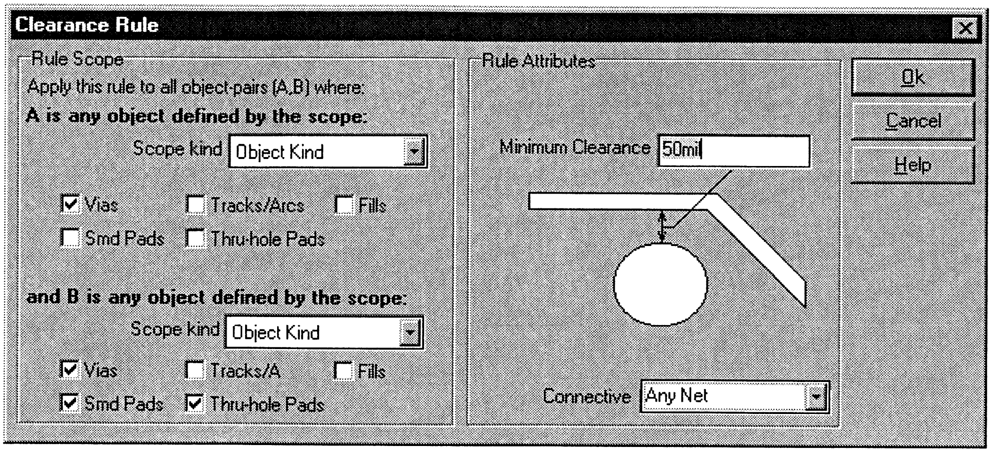
Defines the minimum clearance allowed between any two primitive objects on a copper layer. Use the Clearance Constraint to ensure that routing clearances are maintained.

Setting the Scope

Apply to the Whole Board, to a Layer, to an Object Kind (pads, SMD pads and vias), to a Component Class, to a Component, to a Net Class, to a Net, to a From-To Class, to a From-To or to a pad.

Connective Checking

Typically this would be set to Different Nets. An example of when Any Net could be used is to test for vias being placed too close to pads or other vias on the same net, or any other net. The following figure illustrates how to achieve this;



An example of setting the Connective Checking to Any Net.

How Duplicate Rule Contentions are Resolved

The rule with the largest clearance is obeyed.

Rule Application

On-line DRC, Batch DRC and during autorouting.

Daisy Chain Stub Length

Definition

Specifies the maximum permissible stub length for a net with a daisy chain topology.

Setting the Scope

Apply to the Whole Board, to a Net Class or to a Net.

How Duplicate Rule Contentions are Resolved

The rule which specifies the smallest stub length is obeyed.

Rule Application

Batch DRC.

Matched Net Lengths

Definition

Specifies the degree to which nets can have different lengths. Advanced PCB locates the longest net (based on the scope) and compares it to each of the other nets specified by the scope.

The Matched Length Rules dialog box also allows you to specify how you would like to match the length of nets which fail the matched length requirements. Advanced PCB will add accordion sections to the nets to equalize their lengths.

If you would like Advanced PCB to attempt to match net lengths by adding accordion sections, set up the Matched Length Rules dialog box and then select the Tools-Equalize Nets menu item. The matched lengths rule will be applied to the nets specified by the rule and accordion sections will be added to those that fail. The degree of success depends on the amount of space available for the accordion sections and the accordion style being used. The 90 degree style is the most compact and the Rounded style is the least compact.

Setting the Scope

Apply to the Whole Board, to a Net Class, to a Net, to a From-To Class, or a From-To.

How Duplicate Rule Contentions are Resolved

The rule which specifies the smallest tolerance is obeyed.

Rule Application

Batch DRC.

Maximum Via Count

Definition

Specifies the maximum number of vias permitted.

Setting the Scope

Apply to the Whole Board, to a Net Class, to a Net, to a From-To Class, to a From-To or a Region.

How Duplicate Rule Contentions are Resolved

The rule which specifies the smaller number of vias is obeyed.

Rule Application

Batch DRC.

Minimum Annular Ring

Definition

Specifies the minimum annular ring allowed on a pad. The annular ring is measured radially, from the edge of the pad hole to the edge of the pad.

Setting the Scope

Apply to the Whole Board, to an Object Kind (pads and vias), to a Component Class, to a Component, to a Net Class, to a Net, to a From-To Class, to a From-To, to a pad or to a region.

How Duplicate Rule Contentions are Resolved

The rule with the largest annular ring is obeyed.

Rule Application

Batch DRC.

Min-Max Length Constraint

Definition

Specifies the minimum and maximum lengths of a net.

Setting the Scope

Apply to the Whole Board, to a Net Class, to a Net, to a From-To Class, or a From-To.

How Duplicate Rule Contentions are Resolved

The rule which specifies the tightest range is obeyed.

Rule Application

Batch DRC.

Parallel Segment Constraint

Definition

Specifies the distance two track segments can run in parallel, for a given separation. Note that this rule tests track segments, not collections of track segments. Apply multiple parallel segment constraints to a net to approximate crosstalk characteristics that vary as a function of length and gap.

Setting the Scope

Apply to the Whole Board, to a Layer, to a Net Class, to a Net, to a From-To Class, or a From-To.

Apply to the Whole Board, to a Layer, to a Net, to a Net Class or to a Region.

How Duplicate Rule Contentions are Resolved

Duplicate rules do not create contentions for this rule.

Rule Application

On-line and batch DRC.

Paste-Mask Expansion Rule

Definition

The shape that is created on the paste mask layer at each pad site is the pad shape, expanded or contracted radially by the Expansion specified in this rule.

Setting the Scope

Apply to the Whole Board, to a layer, to a Component Class, to a Component, to a Net Class, to a Net, to a pad or to a region.

How Duplicate Rule Contentions are Resolved

The rule which specifies the smallest expansion is obeyed.

Rule Application

During output generation.

Polygon Connect Style

Definition

Specifies the style of the connection from a component pin to a polygon plane. Two connection methods are available, direct connections (solid copper to the pin) or thermal relief connections.

If Relief Connect is selected you then define; how wide the thermal relief copper connections are, the number of connections and the angle of the connections.

Setting the Scope

Apply to the Whole Board, to a Net, to a Net Class, to a Component, to a Component Class, to a Pad or to a Region.

How Duplicate Rule Contentions are Resolved

The rule which specifies direct connection is obeyed first.

Rule Application

During polygon pour.

Power Plane Clearance

Definition

Specifies the radial clearance created around vias and pads that pass through but are not connected to a power plane.

Setting the Scope

Apply to the Whole Board, to an Object Kind (pads and vias), to a Net, to a Net Class, to a Component, to a Component Class, to a Pad or to a Region.

How Duplicate Rule Contentions are Resolved

The rule which specifies the largest expansion is obeyed.

Rule Application

During output generation.

Power Plane Connect Style

Definition

Specifies the style of the connection from a component pin to a power plane. Two connection methods are available, direct connections (solid copper to the pin) or thermal relief connections.

If Relief Connect is selected you then define; how wide the thermal relief copper connections are, the radial width of the expansion measured from the edge of the hole to the edge of the air gap, and the width of the air-gap. Note that power planes are constructed in the negative in Advanced PCB, so a primitive placed on a power plane layer creates a void in the copper.

Setting the Scope

Apply to the Whole Board, to a Net, to a Net Class, to a Component, to a Component Class, to a Pad or to a Region.

How Duplicate Rule Contentions are Resolved

The rule which specifies Direct Connections is obeyed.

Rule Application

During output generation.

Routing Corners Rule

Definition

Specifies the corner style to be used during autorouting. The corner style can be a 45 degree chamfer or rounded (using an arc). The setback specifies the minimum and maximum distance from the corner location to the start of the corner chamfer or arc.

Setting the Scope

Apply to the Whole Board, to a Layer, to a Net Class, to a Net, to a From-To Class, to a From-To or a Region.

How Duplicate Rule Contentions are Resolved

The order that duplicate rules are obeyed is; Rounded, 90/45 degrees, 90 degrees.

Rule Application

During autorouting.

Routing Layers Rule

Definition

Specifies the layers to be used during autorouting.

Setting the Scope

Apply to the Whole Board, to a Net Class, to a Net, to a From-To Class, to a From-To or a Region.

How Duplicate Rule Contentions are Resolved

The rule with the minimum number of layers is obeyed.

Rule Application

During autorouting.

Routing Priority Rule

Definition

Assign a routing priority from 0 to 100. 100 is the highest priority and 0 is the lowest. The Routing Priorities are relative values which are used to set the order that the nets will be autorouted.

Setting the Scope

Apply to the Whole Board, to a Net Class, to a Net, to a From-To Class, or a From-To.

How Duplicate Rule Contentions are Resolved

The rule with the highest priority is obeyed.

Rule Application

During autorouting.

Routing Topology Rule

Definition

The topology of a net is the arrangement, or pattern of the pin-to-pin connections. By default Advanced PCB arranges the pin-to-pin connections of each net to give the shortest overall connection length. A topology is applied to a net for a variety of reasons; for high speed designs where signal reflections must be minimized the net is arranged with a daisy chain topology; for ground nets a star topology could be applied to ensure that all tracks come back to a common point. The following topologies can be applied with the Routing Topology rule;

Shortest

This topology connects all the nodes to give the shortest overall connection length.

Horizontal

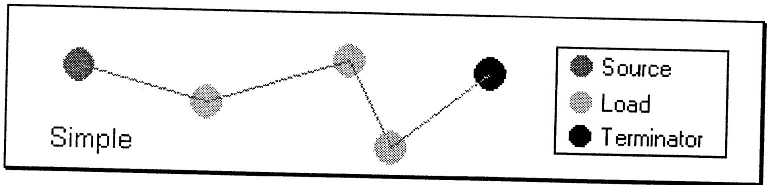
This topology connects all the nodes together, preferring horizontal shortness to vertical shortness by a factor of 5:1. Use this method to force routing in the horizontal direction.

Vertical

This topology connects all the nodes together, preferring vertical shortness to horizontal shortness by a factor of 5:1. Use this method to force routing in the vertical direction.

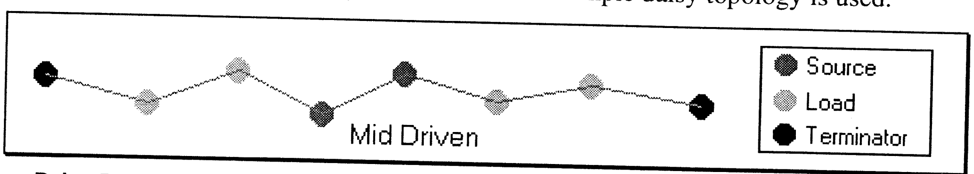
Daisy-Simple

This topology chains all the nodes together, one after the other. The order they are chained is calculated to give the shortest overall length. If a source and terminator pad are specified then all other pads are chained between them to give the shortest possible length. Edit the pad to set it to be a source or terminator. If multiple sources (or terminators) are specified they are chained together at each end.



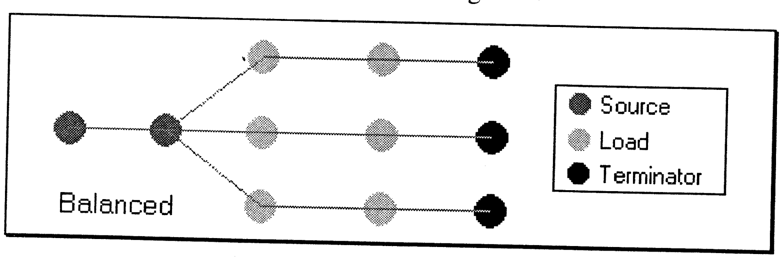
Daisy-Mid Driven

This topology places the source node(s) in the center of the daisy chain, divides the loads equally and chains them off either side of the source(s). Two terminators are required, one for each end. Multiple source nodes are chained together in the center. If there are not exactly two terminators a simple daisy topology is used.



Daisy-Balanced

This topology divides all the loads into equal chains, the total number of chains equal to the number of terminators. These chains then connect to the source in a star pattern. Multiple source nodes are chained together.



Star

This topology connects each node directly to the source node. If terminators are present they are connected after each load node. Multiple source nodes are chained together, as in the daisy-balanced topology.

Setting the Scope

Apply to the Whole Board, to a Net or to a Net Class.

How Duplicate Rule Contentions are Resolved

The rules are obeyed in the following order; Star, Daisy-Balanced, Daisy-Mid Driven, Daisy-Simple, Horizontal, Vertical, Shortest.

Rule Application

During autorouting.

Routing Via Style Rule

Definition

Specifies the via to be used during autorouting. Vias can be thru-hole, Blind (from a surface layer to an inner layer) or Buried (between two inner layers).

Setting the Scope

Apply to the Whole Board, to a Net Class, to a Net, to a From-To Class, to a From-To or a Region.

How Duplicate Rule Contentions are Resolved

The rule with the largest via size is obeyed.

Rule Application

During autorouting.

Routing Width Constraint

Definition

Defines the minimum and maximum width of tracks and arcs on the copper layers.

Setting the Scope

Apply to the Whole Board, to a Layer, to a Net Class, to a Net, to a From-To Class, to a From-To or a Region. The Whole board scope tests primitives on all copper layers.

How Duplicate Rule Contentions are Resolved

The rule with the tightest range is obeyed.

Rule Application

During autorouting and Batch DRC.

Short Circuit Constraint

Definition

Include this constraint to test for short circuits *between* primitive objects on the copper (signal and plane) layers. A short circuit exists when two objects that have different net names touch.

Setting the Scope

Apply to the Whole Board, to a Net or to a Net Class.

How Duplicate Rule Contentions are Resolved

The rule that does not allow short circuits is obeyed.

Rule Application

On-line, batch DRC and during autorouting.

Solder-Mask Expansion Rule

Definition

The shape that is created on the solder mask layer at each pad and via site is the pad or via shape, expanded or contracted radially by the amount specified by this rule. To tent a via set the Expansion to a negative value equal to or greater than the via radius. To tent all vias when the design includes different size vias, set the Expansion to a negative value equal to or greater than the largest via radius.

Setting the Scope

Apply to the Whole Board, to a Layer, to an Object Kind (pads, SMD pads and vias), to a Component Class, to a Component, to a Net Class, to a Net, to a pad or to a Region.

How Duplicate Rule Contentions are Resolved

The rule which specifies the largest expansion is obeyed.

Rule Application

During output generation.

Un-Routed Nets Constraint

Definition

The Un-Routed Nets Constraint tests the completion status of each net identified by the scope. If a net is incomplete then each completed section (sub-net) is listed along with the routing completion. The routing completion is defined as the (connections complete)/(total number of connections) x 100.

Setting the Scope

Apply to the Whole Board, to a Net or to a Net Class.

How Duplicate Rule Contentions are Resolved

The first instance of the rule is obeyed.

Rule Application

Batch DRC.

Vias Under SMT Constraint

Definition

Specifies whether vias can be placed under SMD pads during autorouting.

Setting the Scope

Apply to the Whole Board, to a Net, to a Net Class, to a Component, to a Component Class, to a Pad or to a Region.

How Duplicate Rule Contentions are Resolved

The rule which specifies that vias are not allowed is obeyed.

Rule Application

Batch DRC and during autorouting.

Examples of Using the Design Rules

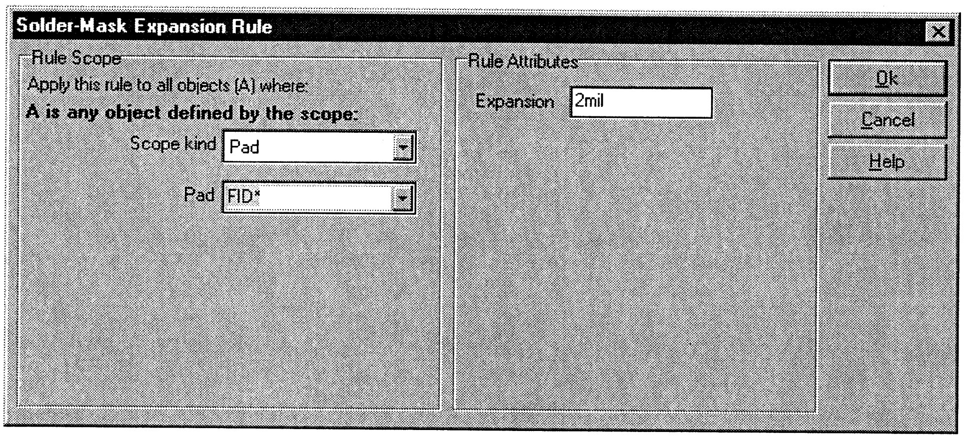
Handling Mask Expansions Around Fiducial Marks

Fiducial marks are copper features used for the optical alignment of a PCB by an automatic assembly machine. It is very important to ensure that the solder mask and paste masks are as per your assembly plants requirements for fiducial marks.

To use fiducial marks in your design;

1. Create the fiducial mark as a component in the library. Use a single layer pad and give the pad the designator name “FID”. Save the component with a name of your choice.
2. Place the fiducial components as required in your design.
3. Add a Solder Mask Expansion rule in the Design Rules dialog box. Set the rule scope kind to Pad and enter “FID*” in the Pad field. Set the appropriate expansion value in the attributes section.

This will ensure that any pad with a designator starting with the string FID has this expansion applied to the solder mask. If you need a different expansion value on an individual fiducial component, change the pad designator for that fiducial to another name and apply another rule to just that pad.



Setting the solder mask expansion for all fiducial pads

You will also need a Paste Mask Expansion rule for fiducials, to ensure that solder paste is not applied to them. To close the opening in the paste mask;

4. Add a Paste Mask Expansion rule in the Design Rules dialog box. Set the rule scope kind to Pad and enter “FID*” in the Pad field.

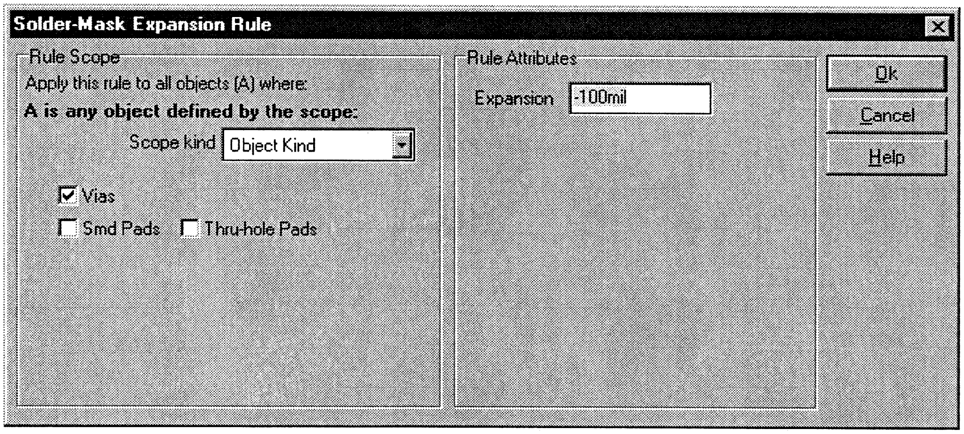
5. Set the expansion value to a large negative number, greater than the radius of the largest fiducial used in the design.

Using a negative number in the Expansion field instructs Advanced PCB to radially contract the opening in the mask by this amount. As long as you supply a contraction value greater than the radius of the largest fiducial, there will be no openings in the paste mask at the fiducials.

Closing the Solder Mask Over Vias

Advanced PCB automatically creates openings in the solder mask at all through hole pads and vias. To close the solder mask over vias;

1. Add a Solder Mask Expansion Rule in the Design Rules dialog box.
2. Set the Scope kind to Object Kind and enable the via check box only.
3. Set the Expansion value to a large negative number, greater than the radius of the largest via in the design.



Closing the solder mask over all vias in your design

Using a negative number in the Expansion field instructs Advanced PCB to radially contract the opening in the mask by this amount. As long as you supply a contraction value greater than the radius of the largest via, there will be no openings in the solder mask at the vias.

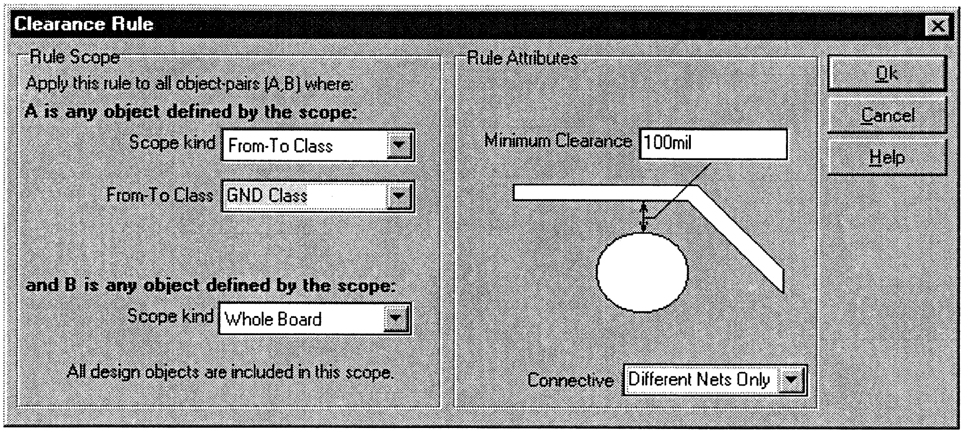
Applying a Clearance Rule to Part of a Net

Design rules can be applied to a particular part of a net.

1. Identify the critical parts of the net by defining From-Tos for those parts of the net.

To create a From-To select the Design-From-To Editor menu item. Define the From-Tos necessary to identify the critical parts of the net. For more information on defining From-Tos, refer to the *From-To* topic in the chapter *Working With a Netlist.*

If there is more than one From-To required to identify all critical parts of the net, create a From-To Class. Classes are created in the Object Classes dialog, select Design-Classes to pop up this dialog box. Once you have identified the critical parts of the net you are ready to add the design rule.



Defining a separate clearance requirement for a critical part of a net.

2. Add a Clearance Constraint rule in the Design Rules dialog box.
3. Set the Scope kind to From-To, or From-To Class if you created a class.
4. Set the Clearance as required. This specifies the minimum distance allowed between any object in this part of the net, to any other object on the board.

Component Placement

Once the board boundaries and keep out requirements have been defined and the netlist successfully loaded into the workspace, the components can be laid out. When the netlist is initially loaded the components are "stacked" with their reference points all located at the last cursor position. The components can then be arranged manually, with the aid of the interactive placement tools, via the auto place tools, or by using a combination of techniques.

Good component placement is a fundamental part of the design process. The manufacturability and routability of the board is highly dependent on the component placement.

➡ In this manual, the term *component placement* refers to the process of arranging or positioning components in the workspace.

Manual Placement

There are a number of set up options which affect the behavior of Advanced PCB during manual placement. These are all configured in the Preferences dialog box (Tools-Preferences).

Snap to Center

Options Tab. When this is enabled components will be "held" by their reference point during a move, with it disabled they will be held wherever the cursor is clicked.

Draft Thresholds

Display Tab. The Strings threshold determines at what zoom level the component designators will change from text to an outline rectangle. To display designators as text when zoomed further out, set this to a smaller number.

To arrange the components manually, select the Edit-Move-Component menu item. The Status Bar will prompt "Select Component". Click on the component you wish to move. The component will float on the cursor and can now be positioned on the board.

➡ During manual placement it is important to re-optimize the netlist regularly, to "update" the connection lines to the current placement. Use the N shortcut key to re-optimize the netlist while a component is floating on the cursor.

Rotating and Flipping Components

Components can be rotated in a number of ways.

They can be rotated when they are floating on the cursor. Press the SPACEBAR to rotate anti-clockwise, hold SHIFT while pressing the SPACEBAR to rotate clockwise. The angle the component rotates is specified in the Options Tab of the Preferences dialog box.

To rotate a group of components, first select the components. The selection can be rotated via the Rotate Selection process launcher (Edit-Move-Rotate Selection). This will first prompt for a rotation angle and then for a Reference Point about which to rotate the selection. The selection can also be rotated by selecting the Move Selection process launcher and then rotated with the SPACEBAR.

To flip a component so that it can be placed on the bottom of the board, press the L shortcut key while the component is floating on the cursor. To flip a placed component, double click to edit the component and change the Layer attribute in the Attributes Tab.

Locking Components

Placement critical components, such as edge connectors, can be locked in place. To lock a component in place, double click on the component to pop up the Change Component dialog box, then enable the Locked check box in the Attributes Tab.

Interactive placement

Advanced PCB includes interactive placement tools, which are especially useful for locally optimizing the placement after using the Auto Place tools. Select the Tools-Align Components menu item, or press the A shortcut key to access the tools.

Aligning Components

The Align Left, Right, Top and Bottom process launchers are used to line up a group of components. Select a group of components, then choose one of the Align menu items. You will be prompted to select one component. The rest of the group will align with the selected item.

Distributing Components

The Distribute Horizontal and Distribute Vertical process launchers can be used to distribute selected components in even horizontal or vertical rows. They will be moved apart horizontally/vertically on the current snap grid until they no longer overlap.

Expanding or Contracting Components

These tools spread apart or squeeze together selected components in increments of one snap grid step.

Centering Components

This option aligns components using the center of the component, rather than the reference points. For example, to place a horizontally oriented bypass capacitor over an IC, place the bypass somewhere near the top of the IC. Select both components, then choose Horizontal alignment. You will be prompted to select one component. Select the IC and the bypass will be horizontally centered over its top.

Shoving Components

This option allows you to "drop" a component into a position which is already occupied by other components. To use this feature, first move the component to the desired location then choose Shove. All the components that surround that location and are in contact with that component will be moved aside to make room for the component, clearances permitting.

The Shove Depth setting (described below) defines the extent of possible changes to other placed components.

If a shoved component hits the edge of the Keep Out perimeter, there will be a "bounce back" effect and that component will back away from the edge and shove the other components until there is no overlap. To avoid shoving a component enable its Locked attribute (Change Component dialog box).

Setting the Shove Depth

This option allows you to set the extent of possible changes. A setting of "1" means that the components which violate the "target" component will move until they are clear of the target only. Setting the depth to "2" means that the process is repeated, allowing the newly violated components to move, and so on. For obvious reasons, it is wise to save an intermediate result prior to shoving placed components, particularly if the design is intricate.

Moving Components to a New Grid

The Move to Grid process is used to move all placed components to a specified snap grid. This is useful if you are changing routing or placement models. For example, if you set your board to route on a 25 mils grid and find that you need the increased density of a 20 mils grid, then you can select Move to Grid, enter the new snap grid and all the component reference points will be moved to the new grid.

Auto Placement

Advanced PCB includes an iterative "global" auto placement tool which places all components in the netlist inside the predefined keep out area. Refer to the *Auto Component Placement* chapter for information on using this tool.

Auto Place from a File

Advanced PCB can position components on the board based on the locations specified in a pick-and-place file. This will move components that have already been loaded into the workspace to the location specified for their designator in the pick-and-place (.PIK) format file.

Lock any components which are not to be moved. Select the Tools-Place From File menu item. You will be prompted to supply a .PIK file name. Any components listed in the .PIK file will have their positions updated, if different from the current position.

The Mid X and Mid Y coordinates in the pick-and-place file are used as the reference for position changes, other coordinates are ignored. Different components in the PIK file can use different units (mil or mm) if required.

Routing Your Design

Routing your design is the process of translating the logical connections into physical connections. These physical connections can include; tracks, vias, pads, arcs, fills, polygons and power planes. Typically, the majority of physical connections are created with tracks and vias.

Advanced PCB includes features specifically tailored to speed this process of translating a logical connection into a physical connection. These features include;

Intelligent manual routing

Place the tracks to create the connections where you choose, you do not have to route the connections along the path shown by the connection lines. Route to a different pin on the net, or create a T-Junction. When you terminate a track the net is analyzed and connection lines are added and removed as required.

Electrical grid

To ease the accurate placement of electrical objects such as tracks and vias, Advanced PCB includes an electrical grid. The electrical grid defines a range within which a moving electrical object (such as a track, pad or via) will attract to another electrical object. The electrical grid overrides the snap grid, allowing you to easily connect to an off grid object.

Violation free object placement

Advanced PCB includes a routing mode where you can only place primitives such that they do not violate any clearance design rules. This feature allows you to route hard up against existing objects, without fear of violating any clearance rules.

On-line Design Rules

A number of the design rules are monitored as you route. These include; the short circuit rule, clearance constraints, track width constraints, routing via style and the parallel segment constraint. Violations are flagged immediately, allowing you to design without errors.

Automatic Loop Removal

Existing tracks can be quickly re-routed. Simply route new track segments and the redundant segments are automatically removed.

Seven track placement modes with look-ahead

The track placement mode defines the way track corners are placed, and includes arcs and 45 degree tracks. Each mode includes a look-ahead segment which you can use to predict the placement of the next segment and accurately terminate the current segment.

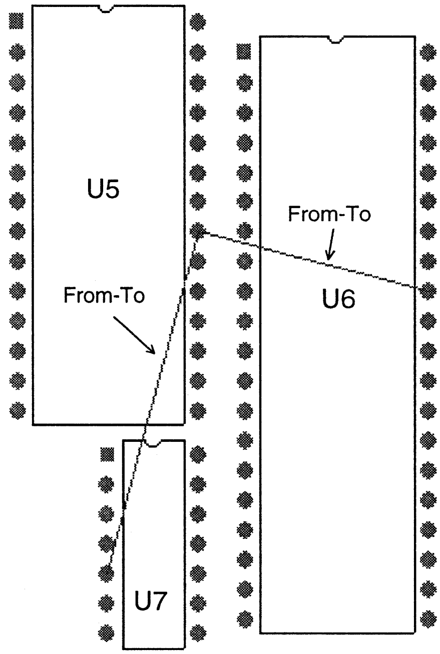
Automatic Via Insertion

Toggle to another copper layer while routing by pressing the *, + or - shortcut keys. A via is inserted automatically.

How Advanced PCB Manages the Connectivity

```
(
RD
U7-5
U6-32
U5-21
)
(
N00023
XTAL2-2
U9-10
U9-9
R4-2
)
(
N00024

XTAL2-1
U9-13
R3-1
)
```



Net RD in the netlist and how it is displayed unrouted on the PCB.

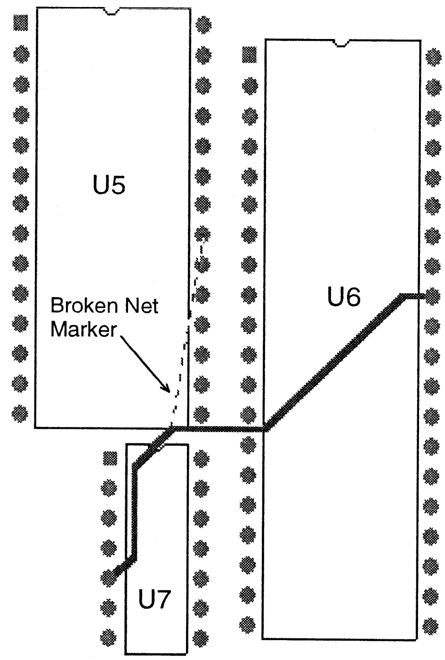
Advanced PCB is a connectivity driven design environment. At all stages of routing your design Advanced PCB monitors and manages the netlist connectivity.

Consider the fragment from a netlist shown in the figure above. RD is the net name given to the connections between pin 5 of U7, pin 32 of U6 and pin 21 of U5. On the schematic this connectivity is represented with wires which join these three pins together. When the netlist is loaded into the PCB workspace Advanced PCB analyses the nets and creates two From-Tos for the net RD, represented by two thin lines on the Connection Layer, as shown in the figure above.

Your task as the designer is to translate these two From-Tos from symbolic connections on the Connection Layer into "physical" connections on the signal layers. You do this by placing tracks to create the physical connections. Whenever you stop placing tracks, Advanced PCB examines the entire net to determine what parts of the net are complete and if the net is still broken. If any breaks are found it inserts Broken Net Markers to join the sub-nets, maintaining the connectivity of the net.

Because Advanced PCB monitors the completion status of the net you are routing automatically, you can route without regard to the arrangement of the From-Tos. For example, you might have commenced routing at U7-5 and then decided to route to U6-32 instead of U5-21. Once you complete this connection, Advanced PCB analyses the entire RD net and adds a Broken Net Marker from the unrouted pin to the closest point on the other sub-net, as shown in the figure below.

To review, your task is to translate connections from symbolic *connection lines* to "physical" tracks, and Advanced PCB's task is to monitor your progress and update the From-Tos and Broken Net Markers as required. There are two distinct advantages to this methodology. The first is that you can route a track to any primitive on the net, you do not have to route between the two pins connected by the From-To. Advanced PCB monitors your progress and adds and removes the Broken Net Markers automatically. The second is that the net connectivity is "unbreakable", you cannot accidentally break it into two unconnected parts. If you delete a track segment Advanced PCB detects the break and immediately adds a Broken Net Marker to restore the net connectivity.



After analyzing the net, Advanced PCB adds the Broken Net Marker

When Advanced PCB analyses the net and determines that it must add a Broken Net Marker, it adds it based on the topology of the net. By default, all nets have their topology set to *shortest*. For these nets the Broken Net Marker is added where the two sub-nets are closest.

➡ Remember, the Broken Net Markers indicate the completion state of the net, not the location where you must route the connection.

If the net has a topology applied the Broken Net Marker is added to maintain the topology. A topology is applied to a net either through a Topology Rule, or by defining fixed From-Tos. For more information on net topology and fixed From-Tos refer to the *Net Topology* topic in the chapter *Working With a Netlist* and the Routing Topology design rule in the *Design Rules* chapter.

Preparing to Route

Preparing the design for routing is an important part of the design process. Here are some tips to improve the routing result.

Setting the Grids

Traditionally PCB's were designed on a standard grid. The grid was calculated to allow objects to be placed quickly and accurately, without the possibility of violating the design requirements. For example; a design that used through-hole components with pins spaced in multiples of 100 mils could be routed on a 25 mil grid. This allowed for 12 mil tracks, 13 mil clearances and one track to pass between the pins of an IC. For information about standard routing setups refer to the *Routing Models* topic in the *Autorouting* chapter.

Changes in packaging technologies, where both imperial and metric pin spacing are used, make it difficult for today's designer to specify a standard grid that fits all the component and design requirements. This has become a major shortcoming of the traditional grid based PCB design environment.

Advanced PCB includes a number of features to aid the designer in overcoming this limitation. These include: an electrical grid which allows one electrical object to snap-to the hot spot of another electrical object, even if it is off grid; look-ahead track placement, allowing you to predict where you want the next track segment to go and accurately terminate the current segment; and violation free object placement with automatic clipping. With these features Advanced PCB behaves as a *shape based manual router*, allowing you to route quickly and accurately to any object, at any point in the workspace.

While these features may initially sound complex, they are quite easy to put to work. If your design is not suitable for one of the traditional routing models, or the density requires you to pack the tracks and vias more tightly than a grid based routing model allows, then set the snap grid to a small value, such as 5 mils or 1 mil. Continue reading this chapter for an explanation of how to route in this mode.

For more information about setting the snap grid and the electrical grid refer to the *Grids* topic in the chapter, *Setting Up the PCB Workspace.*

Move Components onto the Grid

To maximize the number of routing channels available, as many component pads as possible should be on the snap grid. Check if the components are on grid by selecting the Edit-Select-Off Grid Pads menu item (shortcut; S, G). To move all the components onto the snap grid, select the Tools-Align Components-Move To Grid menu item (shortcut; A, G). The Component Move dialog box will pop up allowing you to specify the grid.

Check the Routing Density

To help in the process of determining how the board should be routed; that is, the track and grid sizes, the via size, the number of layers, and so on., Advanced PCB includes a Density Map feature. Select the Tools-Density Map menu item. After a few moments the board will be "painted" with a colored map. The green color represents "cool", or less dense regions, the red color represents "hot", or most dense regions. If there are large areas of red you may wish to analyze the current component placement and try to remove these "hot" zones. If this is not possible your design may require more routing layers.

Enable the Routing Layers

Advanced PCB has 16 signal layers (top, bottom and 14 mid layers), as well as four internal power plane layers. If your design requires the use of internal signal layers and you intend to use blind and buried vias, then the signal layers should be used as *layer pairs.* Refer to the *Vias* topic in the *Design Objects* chapter for more information on using blind and buried vias and layer pairs. Enable the routing layers in the Layers Tab of the Document Options dialog (Design-Options).

Setup the Design Rules

Before you begin routing you should add the required design rules. If you have not already done so, review the *Design Rules* chapter and setup the appropriate rules.

Routing Manually

➡ Before you commence routing, review the *Tracks Placement Modes* topic in the *Design Objects* chapter. A good understanding of the all the track placement features is essential to use the full potential of Advanced PCB while routing.

Because Advanced PCB monitors the net connectivity for you, routing is very straightforward. You place tracks, vias, fills and arcs to create the physical connectivity, Advanced PCB monitors the connectivity and updates the connection lines accordingly.

If you select Place-Track and then click on a design object that has a net name, the track you are placing will adopt the net name, becoming part of that net. If you click on a connection line, Advanced PCB will jump to the nearest pad, and then keep the connection line attached to the end of the track you are placing.

When you quit from placing the track, Advanced PCB examines the net and updates the connection lines. The connection line may or may not stay on the end of the track. Advanced PCB adds the connection lines between all the parts of the net that are not physically connected, based on the net topology. The default topology is shortest, so the connection lines will be added between the closest points on the sub-nets. For more information on net topology refer to the Net Topology topic in the chapter *Working With a Netlist* and the Routing Topology design rule in the *Design Rules* chapter.

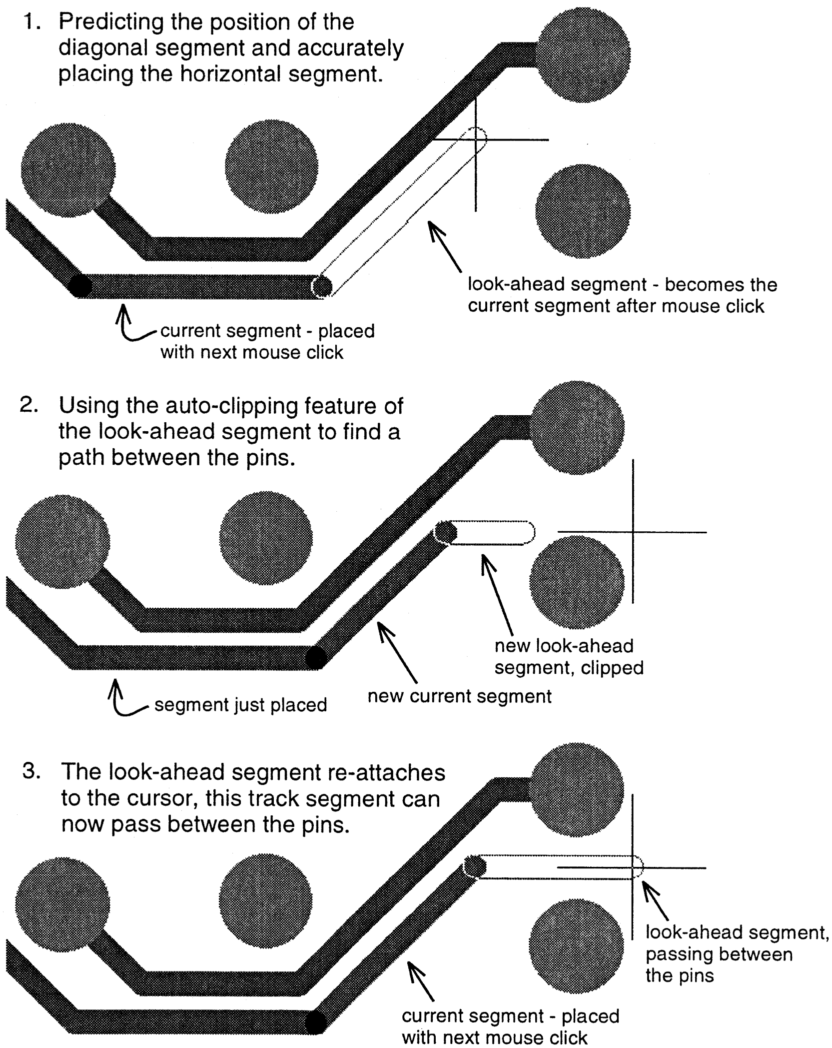
Set the default track size used when manual routing in the Default Primitives Tab of the Preferences dialog box. If you would like a particular net to have a different track size, apply a Width Constraint design rule to that net. Refer to the *Design Rules* chapter for information on applying design rules.

Use the following shortcuts to speed the routing process;

- Press the BACKSPACE key, to remove track segments while routing a connection.
- Press the * key to toggle through the routing layers while routing.
- Press the TAB key to pop up the Change Track dialog box to edit track attributes while routing.
- Use the SPACEBAR to change between the Start and End placement modes. Press SHIFT+SPACEBAR to change track placement modes.
- While routing you may need to refresh the display. Select View-Refresh, or press the END shortcut key.

Placing Tracks and Looking-Ahead

Advanced PCB incorporates a sophisticated “look-ahead” feature that operates as you place tracks. The track segment that is connected to the cursor is a look-ahead segment (shown in outline/draft mode). The segment between this look-ahead segment and the last-placed segment is the current track that you are placing (shown in final mode). This is shown in the following diagram.

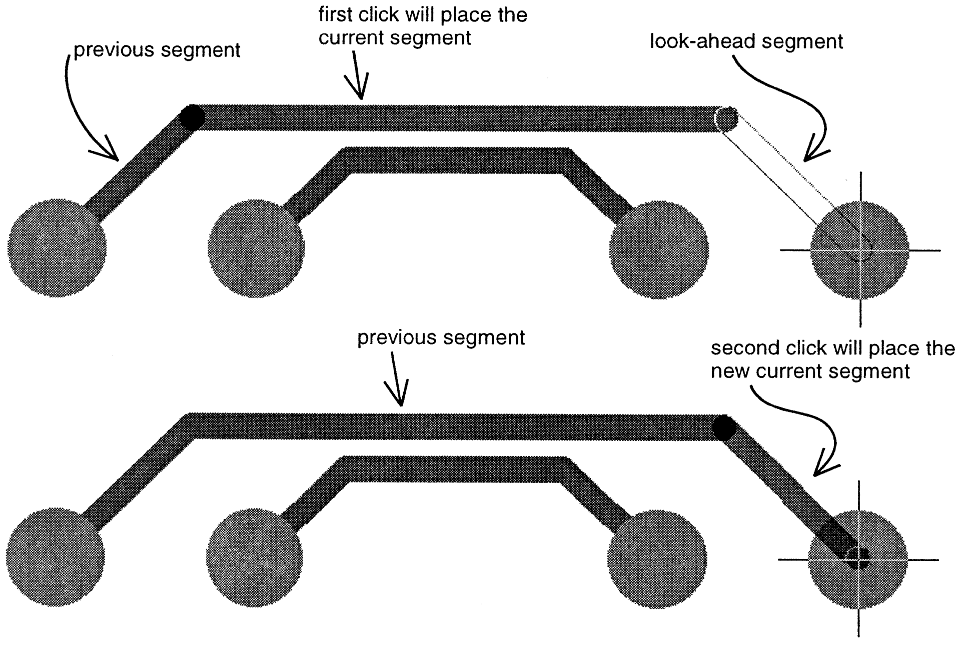


Using the look-ahead feature to predict the next segment and accurately position the current one.

Use the look-ahead segment to work out where you intend to place the next segment and to determine where you wish to terminate the current segment. When you click to place the current segment, its end point will be positioned exactly where you need to commence the next segment. This feature allows you to quickly and accurately place tracks around existing objects and plan where the next track segment can be placed.

As you use the look-ahead segment to guide your routing, you will notice that the track end does not always remain attached to the cursor. It "avoids" electrical objects that belong to another net. This feature allows you to only place primitives where they do not violate any clearance design rules, as shown in step 2 of the previous diagram. In step 2 the cursor has been moved to the right of the pads, but the look-ahead segment is clipped back to the point where no violations would exist. As soon as the cursor is moved up to a point where the look-ahead segment can pass between the pads without causing a violation, it extends across to the cursor. Set the Routing Mode to Avoid Obstacles in the Preferences dialog box (Tools-Preferences).

Consider another example of manual routing shown in the following diagram. You wish to route horizontally across, and then diagonally down to a pad. Previously this was a process of trial and error, judging exactly where to terminate the horizontal segment and commence the diagonal one. The look-ahead segment allows you to bring the cursor down onto the target pad, clicking once to terminate the horizontal segment, then clicking a second time to terminate the new diagonal segment.



Using the look-ahead feature to predictively place tracks

➡ Remember, the segment displayed as an outline is the look-ahead segment, not the segment you are currently placing. If you are trying to place a segment and nothing happens when you click, you are probably trying to place the look-ahead segment. This can happen when you have the placement mode set to End when it should be Start, or it is set to Start when it should be set to End. Press the SPACEBAR to toggle between the Start and End placement modes.

Re-routing

Re-routing is a normal part of the design process. Perhaps a new component has been added, a footprint changed, or you are "cleaning up" after autorouting. Advanced PCB includes a powerful loop removal feature to assist in the process of re-routing tracks. To re-route an existing track;

1. Check that the Loop Removal feature is enabled in the Preferences dialog box.
2. Select the Place-Track menu item.
3. Click on a track or pad to commence re-routing.

The entire track will highlight and a track segment will appear attached to the cursor.

4. Re-route the track along its new path and bring it back to meet the old track. This can be along its length or at a pad.
5. When you have finished the re-route, click RIGHT MOUSE or press ESC.

This new track will create a loop. Advanced PCB will now remove the old part of the loop. The re-route can include new vias, or the removal of old vias. You can route the new track across existing tracks and create violations, with these other tracks being re-routed later.

➡ Loop Removal is enabled in the Options Tab of the Preferences dialog box.

Power Planes

Power planes are special "solid" copper internal layers. Advanced PCB has four internal power planes. If your design is netlist-based, you can assign a net to each of these layers. It is also possible to "share" a power plane between a number of nets by splitting it into two or more isolated areas.

There are two styles of connecting each pin to a power plane: either a direct connection or a *thermal relief* connection. Thermal relief connections are used to thermally isolate the connected pin from the solid copper plane when the board is soldered. Advanced PCB allows you to define the thermal relief shape of each or all pads connecting to the power plane.

Special support is also provided for connecting SMD power pins to power plane layers. SMD pads on a net that is connected to a power plane are automatically "tagged" as connected to the appropriate plane. The autorouter completes the physical connection for these pads by placing an SMD "stringer" - a short track and multi-layer pad which is relief or direct connected to the plane layer.

Connecting to a Power Plane

To connect a net to a power plane;

1. Enable the Internal Plane that you intend to use in the Layers Tab of the Document Options dialog box (shortcut; O, L).
2. Select the Design-Internal Planes menu item to pop up the Internal Planes dialog box.
3. Select the desired net from the drop down list box for the plane you are using.

All the From-Tos for that net will disappear. A small cross will appear at each pad on the net, on the appropriate power plane layer. The cross will look like a "+" for a relief connection, or an "x" for a direct connection.

➡ The properties of the pad-to-plane connections are controlled by the Power Plane Connect Style design rule. Refer to the *Design Rules* chapter for information on using this rule.

Pins that Do Not Connect to a Power Plane

Pads not connecting to the plane are isolated from the plane by a region of no-copper. This region of no-copper is specified as a radial expansion around the pad hole by the Power Plane Clearance design rule. Refer to the *Design Rules* chapter for more information on using this rule.

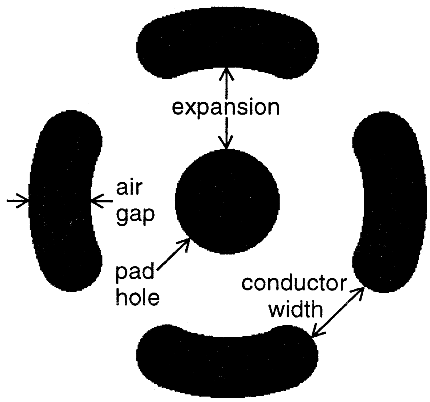
Viewing a Power Plane

Power planes are created in the negative. Object placed on the power plane layer become voids in the copper, while the "black" regions on your screen will become solid copper. To view how the pins connect to a power plane;

1. Enable only the plane layer, pad holes layer and the multi layer.
2. Click on the power plane layer Tab at the bottom of the workspace.

3. If necessary, redraw the screen by pressing the END key.

Relief connected pads are displayed as shown in the adjacent figure. As direct connected pads have solid copper to the pin, they simply show black up to the pad hole. Note that in this book, the white page represents copper and the black represents no-copper.



Creating a Split Power Plane

When you wish to share an internal power plane between two or more nets, you divide, or "split" the plane into regions. Typically the net with the greatest number of pins is first assigned to the internal plane, then regions are defined for the other nets that you wish to connect via this plane. Each region is defined by placing special boundary tracks to encompass all the pins on that net. Any pins which cannot be encompassed in the split plane region continue to display a connection line, indicating that these pins must be connected on a signal layer.

Power planes are constructed in the negative, so the special boundary tracks that you place become a strip of no-copper, creating the separation between this net and the adjacent net(s) on the plane.

To create a split plane,

1. Turn the internal plane layer on in the Layers Tab of the Document Options dialog box.
2. Make the internal plane the current layer by clicking on its layer Tab at the bottom of the workspace.
3. Select the Design-Internal Planes menu item to pop up the Internal Planes dialog box.
4. Connect the pin with the greatest number of pins to the desired internal plane.
5. To add a Split Plane press the Add button. The Split Plane dialog box will pop up. Set the Track Width, Layer and Connect to Net as required.
6. Click OK when the Split Plane dialog box is set up. The dialog box will disappear and a cross hair will appear on the cursor.
7. Click to define each point on the boundary, coming back to the start to create a closed boundary.

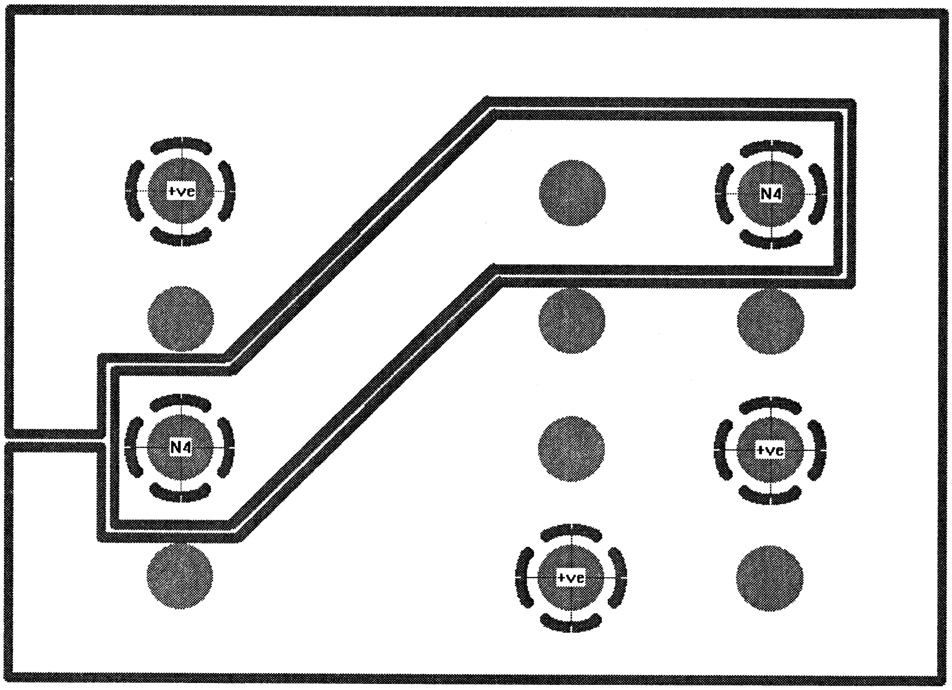
➡ Use the SPACEBAR to change between the Start and End placement modes. Press SHIFT+SPACEBAR to change track placement modes. The boundary can also include arcs.

Once the boundary is closed the Internal Planes dialog will reappear. The new split plane will appear in the Split Planes list, click on it to display the region.

Multiple Split Planes

If the plane is to connect to more than two nets, continue to add split planes to create the other regions on the plane. Adjacent boundary tracks can be placed on top of each other if desired. Typically they would be placed with the tracks overlapping by a small amount to create a continuous no-copper region.

To define a region within a region, the outer region must "wrap around" the inner region, as shown in the following figure.



A split plane within a larger split plane - note how the larger plane wraps around the inner plane. The boundary tracks have only been made thinner to clearly show the two split planes.

➡ A region should not overlap into another region, always wrap them around each other.

Tips for Defining a Split Plane

To make the pads that you wish to encompass within each split plane more visible the following is suggested.

1. Display only a minimum of layers; the Keep Out layer, the multi layer, any mechanical layers needed and the power plane which is being used.

2. Hide all the connection lines (View-Connections-Hide All).
3. Set the display mode of pads and components to draft mode in the Show/Hide Tab of the Preferences dialog box.
4. Enable the Highlight In Full option and the Use Net Color For Highlight option in the Options Tab of the Preferences dialog box.
5. Set the color attribute of each net on the split plane to a different color. To do this; set the Browse mode in the Panel to Nets, select the net in the list and press the Edit button to pop up the Change Net dialog box.
6. Select the Edit-Select-Toggle Net menu item and click on the one of the pads on one of the nets. Repeat this for the other net connecting to the plane. Extend Selection must be enabled to select more than one net at a time (Options Tab of the Preferences dialog box). These two sets of pads will appear in different colors and should now be easy to identify.

Modifying a Split Plane

The boundary of a split plane can be modified after it has been defined. The following modifications are supported;

1. The boundary track width, the layer that the split plane is on, and the net that is connected can all be changed. To change one of these attributes double click inside the boundary of the split plane to pop up the split plane dialog box.
2. The location of the boundary tracks can be changed. Select the Edit-Move-Polygon Vertices menu item. You will be prompted to select a Polygon, click inside the split plane to be modified. The boundary track editing handles will be displayed. Click on an editing handle to move the handle.

Design Verification

Design verification is the process of ensuring that your design is correct and complete. It is a fundamental and integral part of the board design process. The verification process must ensure that the board logically matches the schematic. It must also ensure that the board will be physically functional. Electrical objects such as tracks, pads, vias, etc. must not violate their clearances.

Advanced PCB includes a powerful Design Rule Check feature which verifies that the design meets the requirements specified by the design rules. It tests for routing violations such as clearance errors, unrouted nets, width errors, length errors and also conditions that effect manufacturing.

To enhance productivity while designing the board, Advanced PCB includes an online DRC feature, which tests for copper clearance violations as you route. To clear the violation simply move the object causing the problem and continue routing.

➡ To be able to verify that your design is correct and complete you must have some reference to test against. The reference that Advanced PCB uses is the set of design rules defined in the Design Rules dialog box. The best practice is to define the rules that are appropriate for your design at the beginning of the design process. Refer to the *Design Rules* chapter for further information.

Design Rule Check

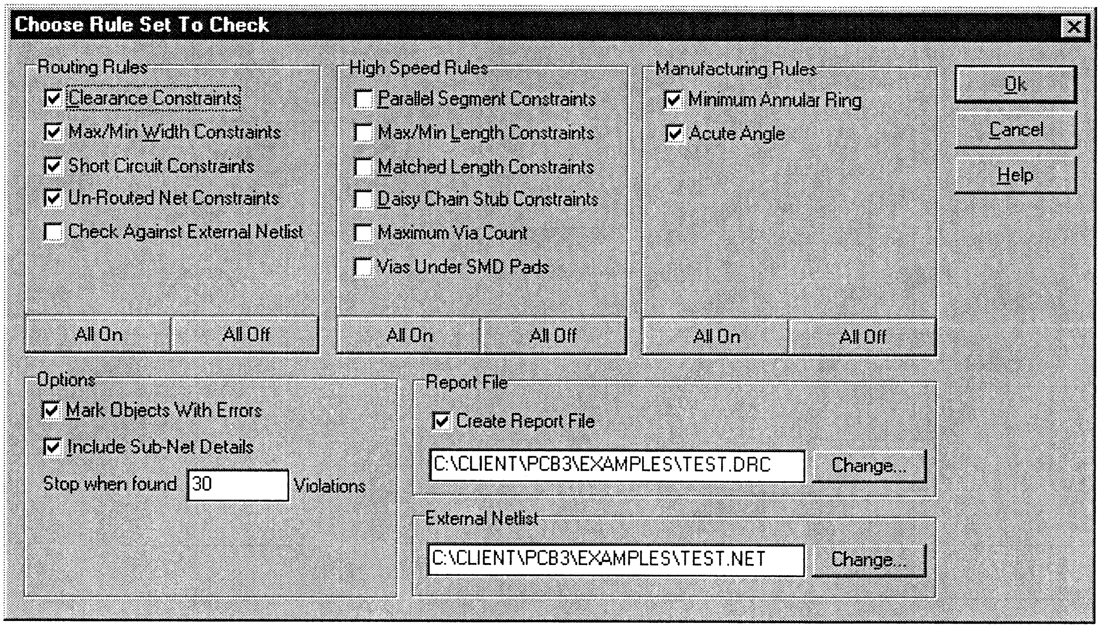
Design Rule Checking (DRC) is a powerful automated feature that checks both the logical and physical integrity of your design. This feature should be used on every routed board to confirm that minimum clearance rules have been maintained and that there are no other design violations. Because Advanced PCB allows you to make changes to the layout at any time it is particularly important that you always perform a design rule check prior to generating final artwork.

Online DRC

The online Design Rule Check feature is enabled in the Options Tab of the Preferences dialog box. Turn this on when manually routing to immediately highlight clearance and parallel segment violations.

Setting Up for a Batch Mode Design Rule Check

Select the Tools-Design Rule Check menu item to pop up the Choose Rule Set to Check dialog box.



Design rule checking validates both physical and logical layout integrity prior to artwork generation.

The batch DRC includes the following features;

Rules

Enable the set of rules that you wish to test. Refer to the Design Rules chapter for a description of each rule.

Options

Mark Objects with Errors

Enable this to highlight primitives with clearance or parallel segment violations in the current DRC Errors color.

Include Sub-net Details

This option works with the Unrouted Net Rule. Enable this option if you require sub-net details.

➡ The Unrouted Net Rule should only be enabled when all connections have been routed, as a connection line is effectively an “open circuit”.

Stop When Found XX Violations

Advanced PCB will stop testing the design after finding this many violations.

Report File

Enable the Create Report File option to automatically create a DRC report file and open it in Text Expert. If no filename is specified the report will be created with the name *filename*.DRC.

External Netlist and Check Against External Netlist

If you enable the Check Against External Netlist option, Advanced PCB will compare the internal PCB netlist against the external netlist specified in the External Netlist field. A report listing any differences between the two netlists will be created with the name *filename*.REP.

Running the Batch DRC

Once you have configured the Choose Rule Set to Check dialog box press the OK button to start the DRC process. Once the check is complete the report file will be opened in Text Expert.

The DRC Report

The DRC report lists each rule that was tested, as specified in the Choose Rule Set to Check dialog box (rules that are not present in the design are not tested). Each violation that was located is listed with full details of any reference information; such as the layer, net name, component designator and pad number; as well as the location of the object. Clearance, length and width errors are highlighted by outlining the object in the current DRC color.

Resolving Design Rule Violations

DRC reports can appear quite daunting to the new user of Advanced PCB. The secret to keeping the process manageable is to develop a strategy. One strategy is to limit the number of violations that are reported. Set the Stop when found feature in the Choose Rule Set to Check dialog box to a small number, say 10. Another strategy is to run the DRC in a number of stages. If you find that the design contains a large number of violations, begin by enabling the rules one at a time. Remember to only enable those rules which you have included in your design. With experience you will develop a preferred approach to testing the various design rules.

➡ Use Cross Probing to work between the PCB file and the report file. Tile the two document windows, select the desired reference in the report file and press the Cross Probe button on the text editor toolbar. The reference can be the component designator, the pin, an X Y location, or a string. The cross probed object will appear centered in the PCB window.

- ➡ Use the Jump shortcuts to quickly locate components, pads, nets, error markers and specific locations. Press the J shortcut key to pop up the Jump menu.
- ➡ A broken net is effectively an "open circuit". Broken into two sub-nets indicates one break, broken into three sub-nets indicates two breaks, and so on. Set the Browse method in the PCB Editor Panel to Nets and use the Zoom button to quickly locate and highlight all parts of the broken net. To identify one sub-net choose the Edit-Select-Physical Net menu item and click on an object in the net. Select-Physical Net will only highlight the sub-net, where as the zoom feature in the Panel will highlight the entire net. Use this difference to quickly locate the break in the net.
- ➡ If your report has one net broken into many sub-nets and a second net reporting extra pins that should be on the first net, there is probably a short between the two nets. Include the Short-Circuit Rule to locate the short.

Generating Output

Completing the PCB layout is only the first part of the process that culminates in the fabrication and assembly of your PCB. The link between your design and the finished board is the artwork that you generate using the Plot/Print, Gerber and NC Drill features that are built into the Advanced PCB design system. The system includes support for a wide variety of "hard copy" options for this critical stage of the design process, whether your computer is connected directly to a printer or if you are sending Gerber files by modem to a fabrication house. Advanced PCB provides a wide range of output options when you are ready to turn your layout into artwork.

Which Kind of Artwork?

Generally, printer output, including impact printers and laser printers, will be suitable for check prints. These prints will allow you to confirm the contents of your output files but will not be used for final artwork. With the improved resolution of laser printers some users find printer output suitable for producing simple prototype boards, providing the level of detail is fairly coarse. The linear distortions inherent in print output can cause problems with layer registration and pad hole-to-pin alignment if this approach is pushed too far.

Most users will need access to higher-resolution output devices to achieve the quality necessary for PCB manufacture. Traditionally this has meant the use of pen plotters or photoplotters. Advanced PCB supports all output devices for which Windows drivers are available. Additionally, Advanced PCB provides special options for plotting directly to HP-GL (Hewlett Packard) and DM-PL (Houston Instrument) and compatible pen plotters, as well as complete support for Gerber (RS-274A) standard plotters.

Postscript Options

High-resolution Postscript "imagesetter" output is now widely available from graphic design and typesetting bureaus. This equipment is capable of producing film positives at resolutions as high as 2540 dpi (dots per inch) and can provide a low-cost alternative to Gerber plots.

However, users should be aware that there are some limitations to using this approach for PCB artwork. The resolution of these systems does not necessarily translate into positional accuracy or linearity, particularly when measured over a large area. There are also film size restrictions.

Photoplotting

Gerber format photoplotting provides the highest resolution output and is generally considered the method of choice for production PCB tooling as it provides the best quality artwork for board production. Photoplots will be required when the design is either large in total area, or of high-density with fine line details.

Working With a Design Bureau

When you start designing, you should have a clear idea of the output requirements mandated by both the PCB technology and production methods you will be using.

If you intend to use the services of a plotting bureau or PCB manufacturer take the time to consult with them before you start generating artwork. Bureaus and manufacturers often have specific requirements that must be reflected in the files or artwork that you submit. For example, you may wish to either “step and repeat” or panelize your Gerber files for efficient quantity fabrication.

To accomplish this, you have to know the film size accepted by the photoplotter, clearance requirements, etc. as well as the manufacturing tolerances involved. Planning for numeric (N/C) control drilling, requires similar consideration.

In some instances, the bureau or fabrication facility will prefer to work directly with “raw” Gerber files (or even PCB files) rather than panelized Gerbers. Understanding these requirements will help you to plan the entire design process for efficient and trouble-free completion.

Print / Plot Layers

Layers that can be printed, pen plotted or Gerber plotted include:

Solder masks (Top and Bottom)

These masks match the pads and vias in your design. The mask layers are automatically generated based on the pads. Enlargements of these masks are required by manufacturers to accommodate manufacturing tolerances and can be specified (in mils or mm) by including the Solder Mask Expansion design rule. Multiple rules can be added if different pads have different requirements. Refer to the *Design Rules* chapter for information about the Solder Mask Expansion rule.

Paste masks (Top and Bottom)

Paste masks are similar to Solder masks but are used to create solder paste screens when the “hot re-flow” technique is used to mount SMD components. Like Solder masks, Paste masks are plotted in reverse, for efficiency. These masks can include an expansion or contraction at each pad by including the Paste Mask Expansion design rule. Multiple rules can be added if different pads have different requirements. Refer to the *Design Rules* chapter for information about the Paste Mask Expansion rule.

Drill Guide

A small cross is plotted at each drill site, to provide a visual target for drilling. Specify the size of the cross in the Drill Plots Tab of the Setup Output Options dialog box (Setup Printer, Layers button).

Drill Drawing

This layer places a marker at each drill site. This marker can be an alphabetical character, a string specifying the hole size or a graphical symbol. Add the .LEGEND special string to this layer to include a table of hole count and drill size, for each hole size. There is a limit of 16 different hole sizes when the Graphic Symbols option is used.

Top layer

This is the "component side" signal layer. All the tracks, arcs, fills, strings and top layer pads that have been placed on this layer are included when printing, as well as multi-layer pads and vias.

Mid layers (1–14)

There are 14 mid-layers available in Advanced PCB. All primitives that have been placed on these layers are included when printing, as well as multi-layer pads and vias.

Bottom layer

Also known as the "solder side" of the PCB. Components can also be placed on this layer. Tracks, arcs, fills, strings and bottom layer pads placed on this layer are printed, as well as multi-layer pads and vias.

Top Overlay, Bottom Overlay

Also called the silkscreen layers, these are normally used for component outlines and component text.

Internal Power Planes

These special mid layers consist mainly of solid copper in the manufactured board, and are therefore printed in "reverse" (in the negative), for efficiency. Fills, tracks and arcs can be placed on this layer wherever you wish to create a void in the copper. For example, many manufacturers recommend that you place a track around the perimeter of your board to keep the copper clear of the trim area.

Mechanical Layers 1–4

These layers can be used to create Fabrication and Assembly drawings, showing dimensions, trim marks, datum marks, holes, assembly instructions and other mechanical details of the board. The contents of these layers can be added to other layers when printing.

Pad Master

These layers can be used to print / plot all pads on the top or bottom layers.

Mid Layer Pads

Unconnected Mid layer pads can be removed from Mid layers (1–14) prints/plots by disabling the Include Unconnected Mid Layer Pads option in the Layers Tab of the Setup Output Options dialog box.

Keep Out Layer

The contents off the Keep Out layer can be printed / plotted if desired.

Setting Up

There are three approaches to generating output from Advanced PCB. The first is Gerber file generation, where the PCB information is translated into a set of Gerber files, from which a photoplotting bureau can produce a set of phototools. The second method of producing output is via the Protel plotter driver which supports both HP-GL and DM-PL languages. The third method of producing output is via a Windows driver. This gives access to the vast range of printing technologies and devices supported by Windows drivers.

To select and setup the output type, select the File-Setup Printer menu item. The Printer Setup dialog box will pop up. This dialog box will list all currently available output drivers. Adding another Windows compatible output device is as simple as installing the Windows driver. Consult your Windows documentation for further information on adding printers and plotters.

The Protel drivers are built into Advanced PCB. The Protel plotter driver is not a Windows driver, it sends the plot data directly to the specified port. It is a general purpose HP-GL / DM-PL plotter driver which was included when it was not possible to obtain true Windows vector plotter drivers. While the driver is still included in Advanced PCB, there are now true generic Windows vector plotter drivers available. These allow the plotter to be accessed in a true Windows fashion, including over a network. Contact your plotter manufacturer or your Protel agent for information on Windows plotter drivers.

When a driver is installed, it will appear twice in the list of output devices, as a "Final" option and as a "Composite" option.

Final Output Drivers

The final output options print each enabled layer on a separate sheet. If desired these separate layers can be panelized onto a single sheet. This option is used when "final artworks" are to be generated.

Composite Output Drivers

The composite output options print all enabled layers together. This allows you to simulate the board, with the desired layers stacked directly on top of each other. These are also known as "check plots". Depending on the capabilities of the printer, it may be possible to print each layer in a different color or a different gray scale.

Layers Button

The response when the Layers button in the Setup Printer dialog box is pressed depends on whether the selected output is Gerber, final or composite.

Gerber and Final Output

Pressing the Layers button pops up the Setup Output Options dialog box. This dialog box has four Tabs;

Layers

Enable each layer you would like to generate output for. Note that the contents of each mechanical layer can be added to all plots.

Drill Plots

This tab controls the setup for drill drawing layer and drill guide layer printing. Select the preferred Drill Drawing Symbol to be used on the drill drawing. Include the .LEGEND special string on the drill drawing layer to add a table of hole / drill sizes.

Mirroring

Layer mirroring will flip bottom side layers along the horizontal axis so the manufacturer can use the emulsion side down when exposing the resist for all board layers, yielding the most consistent result.

Composite Output

Pressing the Layers button pops up the Setup Composite Print Layers dialog box. For each layer there is a check box to enable that layer and a “color” box. This “color” box will display the color or gray scale assigned to that layer, depending on the capabilities of the output device. Click on this box to change the color or gray scale assigned to that layer.

Options Button

The response when the Options button in the Setup Printer dialog box is pressed depends on whether the selected output is Final, Composite or Gerber.

Final

Pressing the Options button pops up the Final Artwork.... Dialog box.

Scale

The output can un-scaled, scaled manually or scaled to fit the page. If the Fit Layer On Page option is enabled the print or plot will be expanded / contracted to fit on the page size set up for your printer, maintaining the aspect ratio.

➡ Make sure that you have set the Portrait/Landscape mode in the printer driver dialog box to best fit you PCB shape when using the Fit Layer On Page option.

The X and Y correction factors are provided to correct repeatable errors in your printer. These two values are multipliers for all coordinates sent to the printer. The x-correction is multiplied with all horizontal values and the y-correction is multiplied with all vertical values. To calculate corrections, print or plot track segments of known dimensions, as large as possible, then measure the result. It can be helpful to make reference measurements at various locations, and make multiple test plots to be sure that repeatable errors, not mechanical problems, are being isolated. A correction factor can then be calculated to cancel the repeatable linear errors on either axis.

Options

Each layer can be printed on a separate page, or panelized onto the same page to conserve paper or film. The show holes option will work with raster type printers but generally does not work with vector type printers. This is due to the way the data is passed to and processed by the Windows driver.

Setup

Press the Setup button to pop up the printer driver setup dialog box. This dialog is part of the printer driver and may have different options for each printer.

➡ The printer driver dialog box should not be used to set the scale for your PCB artwork, as it is only accurate to around ±1%. The Scale options in the Final Artwork or Composite Artwork dialog boxes should be used for scaling the output.

Protel HPGL Plotter Final Driver

This Final Artwork dialog box includes a number of other options. It allows you to select the plotter language to be used and also specify the port (or file) the plot data is to be sent to.

The Serial button will pop up the Setup Serial Communications dialog box. As this driver communicates directly to the port, the serial communications parameters can be defined here. Ensure that the plotter has the matching setup.

The Pens button pops up the Plotter Pens Setup dialog box, which allows the speed and thickness for up to eight pens to be specified.

The Use Software Arcs option is included for plotters which do not support arc commands in the plot data. If your plotter is giving erroneous plot results with arcs, enable this option. Advanced PCB will replace the arcs with very small straight line segments.

Composite

Pressing the Options button pops up the Composite Artwork.... Dialog box.

Scale

The output can un-scaled, scaled manually or scaled to fit the page. If the Fit Board On Page option is enabled the print or plot will be expanded / contracted to fit on the page size set up for your printer, maintaining the aspect ratio.

➡ Make sure that you have set the Portrait/Landscape mode in the printer driver dialog box to best fit you PCB shape when using thc Fit Board On Page option.

The X and Y correction factors are provided to correct repeatable errors in your printer. These two values are multipliers for all coordinates sent to the printer. The x-correction is multiplied with all horizontal values and the y-correction is multiplied with all vertical values. To calculate corrections, print or plot track segments of known dimensions, as large as possible, then measure the result. It can be helpful to make reference measurements at various locations, and make multiple test plots to be sure that repeatable errors, not mechanical problems, are being isolated. A correction factor can then be calculated to cancel the repeatable linear errors on either axis.

Primitives

Each primitive type can be rendered in Final (solid) or Draft (outline), or can be Hidden (not printed).

Other Options

Select Color/Gray if the printer supports color or gray scale printing, otherwise select monochrome.

The show holes option will work with raster type printers but generally does not work with vector type printers. This is due to the way the data is passed to and processed by the Windows driver.

Setup

Press the Setup button to pop up the printer driver setup dialog box. This dialog is part of the printer driver and may have different options for each printer.

➡ The printer driver dialog box should not be used to set the scale for your PCB artwork, as it is only accurate to around ±1%. The Scale options in the Final Artwork or Composite Artwork dialog boxes should be used for scaling the output.

Protel HPGL Plotter Composite Driver

This Composite Pen Plot dialog box includes a number of other options. It allows you to select the plotter language to be used and also specify the port (or file) the plot data is to be sent to.

The Serial button will pop up the Setup Serial Communications dialog box. As this driver communicates directly to the port, the serial communications parameters can be defined here. Ensure that the plotter has the matching setup.

The Pens button pops up the Plotter Pens Setup dialog box, which allows the speed and thickness for up to eight pens to be specified.

The Use Software Arcs option is included for plotters which do not support arc commands in the plot data. If your plotter is giving erroneous plot results with arcs, enable this option. Advanced PCB will replace the arcs with very small straight line segments.

Gerber

Pressing the Options button pops up the Gerber Output dialog box. Remember to set up the required layers prior to setting up the Gerber options and generating the Gerber files.

The Gerber setup options are defined later in this chapter, in the section Generating Gerber Files.

Generating a Print or Plot

Select the File-Print menu item when you are ready to print, plot or generate Gerber files. The print or plot will be generated based on the current settings in the Printer Setup dialog box.

As the print is being generated it will be displayed on the screen – you will be able to see the format of the panel files if using the Panel option or you will see the layers drawn superimposed, if the composite option is selected.

The print can be interrupted at any time. Press the Cancel button and a prompt will appear allowing you abort or continue printing.

While Advanced PCB is printing it is possible to minimize the window and continue printing in the "background". While printing large files using Print Manager, a message may pop-up from Print Manager telling you that it cannot write to the printer and that the print operation has been suspended. This indicates that the printer cannot accept the data as fast as Print Manager can supply it. If this happens, give the printer a few minutes to catch up, then open the Print Manager program and click Resume button and the rest of the file will be downloaded.

➡ You should be aware that rotation of fonts is not supported for all printers and the substituted fonts will only be used if the text on your printed circuit board is in a standard horizontal orientation, and within the size capability of the printer. Other printers such as Postscript printers can support rotation of fonts at any angle.

Postscript Printing Tips

Postscript printers and "imagers" generally produce output between 300 and 2540 dpi. Because of the high resolution obtainable from these devices, many users are interested in producing artwork quality Postscript prints as a lower-cost alternative to Gerber plots. However, there are a few limitations which should be considered before printing.

High-resolution laser imagers print directly onto film or sensitized paper. While these devices are quite accurate horizontally, they do not always achieve consistent linearity, particularly on devices where the film or paper moves off a roll, then through the printing mechanism via a series of rollers.

Some typesetting / graphic arts bureaus now use Postscript imagers that use cut, rather than roll film, mounted on a large drum. These imagers suffer much less from linearity problems and may provide a suitable alternative to Gerber plots for non-critical designs.

To test any Postscript device, create a file with vertical and horizontal tracks of known length and carefully measure the output with a rule of known accuracy. This will allow you to apply a correction factor scale setting to either axis, which should minimize the problem. The amount of linearity error may not always be constant, so you should check each final artwork print for accuracy before committing the art to fabrication.

Another problem with 300 or 600 dpi "desktop" laser printers is the "overspray" and "bleed" effects created when the toner is fused to the paper. Small particles adhere to the paper on either side of lines, etc., creating the potential for unwanted effects in your artwork.

When designing for laser print artwork, you should keep the clearances generous, and again, print at a reasonable scale to minimize scale and bleed effects.

The print quality obtainable with a laser printer is largely determined by the paper. A number of special papers are currently available (primarily for the graphics arts trades) which reduce this toner "bleed" into the paper, hence making the outline sharper. Some of these special papers are slightly heavier and treated to resist the waxes and glues used for paste-up, making them easier to handle. Be especially careful keep these paper laser prints clean.

Postscript compatible photo-typesetting equipment has the advantage of being able to provide output at very high resolutions (up to 2540 dpi). These devices can also print a direct film positive to A3 (or "B") size.

However, the concern with linearity, described above, applies to these devices as well. The problem of linear accuracy will already be familiar to imagesetting bureaus who provide color separations to the graphics arts industry.

➡ Some Postscript printers will "time out" and discard the current data when they don't receive the end of page marker within a specified time. This can cause problems where you seem to be missing pages from your plots. If you experience this problem using a Postscript printer or any other printing device then you should go to the Control Panel, select the printer icon, select the printer and click the Configure button. Change the Transmission Retry to 500 seconds, or some other large number. This will allow the printer sufficient time to catch up before the Print Manager gives up.

Pen Plotting Issues

There are two basic choices if you wish to produce draft or final artwork on pen plotters:

1. You can plot via a Windows plotter driver. This option is preferred if your device is well-supported by a current Windows driver. This should not be a problem for newer devices that use raster, rather than vector plot routines, such as the newer large format "ink jet" type plotters. There are also true vector plotter drivers available now, contact you plotter supplier or your Protel agent for further information.

2. You can bypass the Windows plotter drivers by using Advanced PCB's built in plotter driver. This option allows direct control over vector-type HP-GL and DM-PL plotters. If you use a conventional pen plotter, this will produce high quality pen plots from Advanced PCB which will be generated efficiently and with superior plot quality than standard Microsoft Windows plotter drivers.

➡ If you plan to use a bureau, or are plotting from another computer, you will need to generate your plot as a file. To do this with a Windows driver you must map the plotter to File instead of a printer port. Refer to your Windows User Guide for information on how to do this. To plot to a file with the Protel plotter driver, select the driver in the Printer Setup dialog box and press the Options button. In the Pen Plot dialog box set the Output Port to File. You will be prompted to name the file when you generate the plot.

Producing good quality pen plots

There are many aspects of producing PCB tooling that will have a direct impact on the finish and reliability of the finished product. Advanced PCB is capable of designing to tolerances of 0.001 mil (0.000001 inch). However, if your final output is only accurate to 10 mils, there is little point in using "fine line" clearances in your design. Therefore, it is important that you consider the available technology for each stage of production before you design.

Particular care is required when planning the use of internal power and/or ground plane layers. Adequate clearance must be maintained around all non-connected pin holes to ensure that shorts do not result from slight misalignment of layers when the board is manufactured. The recommended minimum for this clearance is 0.4mm (.016") for boards less than 305mm (12") in a side. Larger boards require a clearance of at least 0.5mm (.020").

Many manufacturers require a copper free area around the edge of internal power plane layers to prevent shorting between these layers when the board is laminated. You can place tracks on these mid layers to "hold back" the solid copper, wherever desired.

Pen plotters can be used to produce very sophisticated design artwork, when the many variables affecting plot quality are understood and applied to the process. But, there are inherent problems with pen plotting that need consideration. In many cases there will be distinct advantages, in terms of final costs (when productivity is considered), in using the services of a photoplotting bureau, even when good pen plotting facilities exist in-house.

The variables that directly effect plot quality include:

- Accuracy of the plotter – particularly its "repeatability" or ability to return accurately to specific coordinates, over the entire plot area.
- Type and condition of plotting pens.
- Plotting film or paper.
- Type and age of the ink selected.
- Environmental factors – i.e. temperature and humidity.
- Pen speed and pen size settings.

Other factors include the experience of the operator and the maintenance and storage of equipment and materials. There are a number of simple rules you can follow to make sure that the best quality possible is obtained.

Perhaps the most important factor is the quality of the paper (or drafting film) and the pens that you use. Use inexpensive paper and fiber tip pens for check plots – save the best pens and film for the final plot.

Plotter pens and plotting inks

There are a wide range of plotting pens on the market. Felt, and plastic tipped pens are convenient to use, but only suitable for draft plots. Pens used for master artwork must be capable of providing a consistent ink flow, must not dry out when the pen is lifted off the film for short periods and must be of the correct diameter for the selected plot scale.

The pens that have been found to be the most suitable are those with tungsten carbide, cross-grooved points. A latex-based ink will provide a dense plot without the ink running or drying out in the pen. Your local plotter supplier will make specific recommendations.

Drafting film

Your choice of drafting film is not as critical as the choice of pen or ink, but good quality film is recommended. For best results, use single-matte or double-matte polyester film of around 3 mils thickness. Your local plotter supplier will make specific recommendations.

Setting up the plotter

The options available will depend upon the type and model plotter selected. Guidelines should be documented in your plotter manual. Most plotters will require the following setup decisions:

Pen speed

Determining the best pen speed is largely a matter of trial-and-error. Some users may find they have to choose a slower speed to get properly "filled" tracks and pads. The condition of the pen points, freshness of ink, etc., can have a significant impact on plot quality. Some plotters have force and acceleration options in addition to pen speed. Consult you plotter manual for recommended setting for the paper or film and pen combination you intend to use.

Assigning pens

If generating pen plots for a multi-pen plotter, you can assign different pens to different layers for plotting a color check plot. Pen size and pen number assignments are made from the Pen Plot dialog box.

Setting the pen speed

Pen speed is a critical, and often overlooked factor in plot quality. It will be worthwhile to make a series of experimental plots to determine the optimum settings for your combination of plotter and materials. You may also improve the plot result by making small adjustments to the pen size selection. Slight changes will adjust the amount of "overlap" obtained when filling in pads, fills and wide tracks – with further adjustments needed as the pen wears during normal use.

Communications with Serial plotters

Most plotters are controlled via an RS232-C (serial) interface. A cable connects the plotter and computer to provide two-way communication. Correctly configuring this combination of computer software, serial port, cable and plotter can be a challenge, even for experienced engineers.

If you are installing a serial plotter for the first time, this section explains the relevant RS232-C conventions.

The RS232-C standard defines the signals for bi-directional communication where there is no inherit distinction between the computer and the output device. In the jargon of serial communications both devices are referred to as DTE, or Data Terminal Equipment. Signals, such as Transmitted Data are assigned to the same pins in both devices, unlike the parallel standard where each pin has a single function.

Each serial "terminal" needs an intermediary device or devices to connect the "transmitted" data pin of one DTE to the "received" data pin of the other, and vice versa, and to also correctly configure the handshaking signals.

These intermediate devices are called Data Communications Equipment (DCE), which connects to DTE, transmits and receives the data over a channel but is neither the source nor the final destination of the data. A modem is a DCE – it both modulates data for transmission over a single voice channel and demodulates it back to digital data.

Baud rate, data bits, etc.

Once a correct serial connection between the computer and plotter is achieved, the correct communications parameters must be selected.

Your plotter manual should indicate the default settings of the plotter and will contain information on changing the communications setup. Some plotters do not have default settings, as such, but use DIP switch settings which must be configured before the plotter is operated.

Match these parameters using the Setup Serial Communications dialog box. Once set, these settings are stored with your Advanced PCB preferences.

A baud rate of 2400 BPS is standard for many plotters and a good place to start, if you don't know the specific recommendations for your plotter. This is an intermediate baud rate and should yield error-free data transmission with cables up to 50 feet (15 meters).

Your plotter manual should also document its interface and handshake settings.

Solving plot communication problems

If you are confident that you have the right cabling and parameter settings and you still can't plot successfully, check the following items:

1. Inspect the cable connections and make sure that no wires have broken. Also check that your Windows settings match the plotter baud rate, parity, etc..
2. Confirm that you are using the selected serial port.
3. If your plot progresses normally at first, then starts putting stray lines or arcs all over the layout, this generally indicates improper handshaking. You may also have a problem with one or more pin assignments and your cabling may need modification.
4. Another possible solution is to keep the plotter cable as short as possible and keep it away from power cords and other "noise" sources.

If you are using a long cable, you may have to reduce the baud rate to obtain error-free transmission. Due to the distributed resistance and capacitance of cables, there is a trade-off between cable length and baud rate for reliable data transmission.

Remember, if you change the communications settings at the plotter, you will have to match the new settings in the Set-up Serial Communications dialog box.

Erratic plotter behavior can also be the result of plot file corruption. If you have been unable to solve your plotting problem, try plotting one of the (supplied) demonstration files, as a cross-check.

Generating Gerber Files

Advanced PCB allows you to generate Gerber format photoplot files from the current board layout. This Gerber generation process is highly automated and very efficient, requiring minimal user input. Advanced PCB will even automatically generate the aperture table, used in photoplotting the file. You can also import the Gerber files back into the PCB editor – a convenient way to verify files prior to photoplotting. Additionally, you can batch load photoplot files, converting the batch back into a PCB file. This provides a powerful way to translate PCB files from other design system into the Advanced PCB format.

About Photoplotters

Photoplotters are similar to pen plotters in many ways, the primary difference being that photoplotters use light to plot directly onto photosensitive film. The many advantages of this approach has led to the widespread adoption of photoplotting in the electronics industry.

Because the etching of printed circuit boards is generally based upon photographic techniques, the production of positive and negative photo-tools (or films) is an inherent part of the process. When the original artwork is a pen plot, a number of intermediate steps have to be performed to produce the final tools. Pen plots are generally plotted at least 2:1 scale to achieve reasonable accuracy and then photographically reduced.

Photoplotters provide sufficient accuracy to generate a precision 1:1 plot in a single operation. Photoplotting bureau services are widely available and all designers should carefully consider its advantages. To make the best use of photoplotting, it's helpful to understand some key concepts.

Vector vs. Raster Plotters

Photoplotters fall into two general categories, vector and raster.

Vector plotters generally use an aperture "wheel" or "slide" to create the combination of "flashes" and "strokes" to "draw" an image. These make images in much the same way as pen plotters. They select a drawing instrument (or aperture) and describe a vector in the drawing space. The result can be seen as a line the width of which is defined by the aperture. Apertures are a collection of defined shapes which allow the plotter to plot varying track widths, pad shapes, etc. Flashes occur when there is no movement of the light source, strokes occur whenever there is movement while the light source is on. Some plotters use separate apertures for strokes and flashes in order to maintain consistent exposure. Others control the light intensity – all apertures serving for both uses.

Raster plotters do not use a system of fixed apertures. They read the Gerber file, storing an "image" of the whole plot, which is then scanned onto the film, line-by-line, not unlike a television image. Raster photoplotters can synthesize a virtually unlimited variety of different apertures, providing a great amount of flexibility to the designer.

Some photoplotters use the Postscript language. Photoplot files for these devices will be prepared using an appropriate Postscript driver. For information about Postscript printing, see the *Postscript Printing Tips* section, elsewhere in this chapter.

You will want to know something about the "target" photoplotter, in order to make efficient use of its capabilities when you design.

➡ Contact your photoplot bureau before generating any photoplots. Matching, wherever possible, available plotting options at the edit level can save considerable time and expense when generating Gerber photo-tools.

Photoplotter Languages

Nearly all photoplotters are controlled by a vector-based plotting language, developed specifically for this task, generically referred to as "Gerber" – a registered trademark of the Gerber Scientific Company. This language has become an industry standard (also known as RS-274). While the language has evolved to accommodate changes in both plotting equipment and design tasks, a number of potential difficulties and limitations must be considered by the designer when planning a job for Gerber output.

A Gerber format file describes a plot as a series of draft codes (or commands) and coordinates. The draft codes control the aperture to be used, turning the light "on" or "off" and so on. Coordinates define the position of the various flashes and strokes on the plot. This information is stored as an ASCII text file.

The structure of Gerber files can vary due to a number of "optimizations" that have been added to the format over time, to address the changing capabilities of plotting hardware. Your photoplot bureau may need to know details regarding Protel's use of Gerber format, so we have described it in some detail, below.

Protel Gerber files are divided into individual commands, followed by carriage return code then a line feed code. Each record is terminated by the character "*".

The records may refer to an absolute location or a draft code which changes apertures. Thus a record might be "X800Y775*" which instructs the plotter to move to a particular coordinate or "D16*" which is a draft code or command, such as a new aperture selection.

Some plotters reserve draft codes D01–D09 for uses other than aperture selection, for example:

- D01 turns the light source on.
- D02 turns the light source off.
- D03 flashes the light source.

On some older plotters the special code "G54" needs to be sent before each change of aperture code. The last Gerber record is terminated by the special record M02*, which is followed by another block, containing the character "hex 08," then 509 "spaces" (hex 20), then a carriage return and a line feed.

You can inspect any Gerber file with a text editor or word processor capable of loading an "un-formatted" text file.

About Apertures

All Gerber format photoplotters use apertures – which describe the available tools used to draw on film. In the case of a vector plotter these apertures correspond to various sizes and shapes of holes in an aperture wheel or slide. Light is projected through these apertures onto the film emulsion.

Raster plotters are not limited to a set of specific aperture sizes and shapes. Raster imaging systems interpret the aperture information in the generated Gerber file and the entire plot image is synthesized and represented by a bit map and plotted line-by-line, not unlike a television image.

Using Apertures

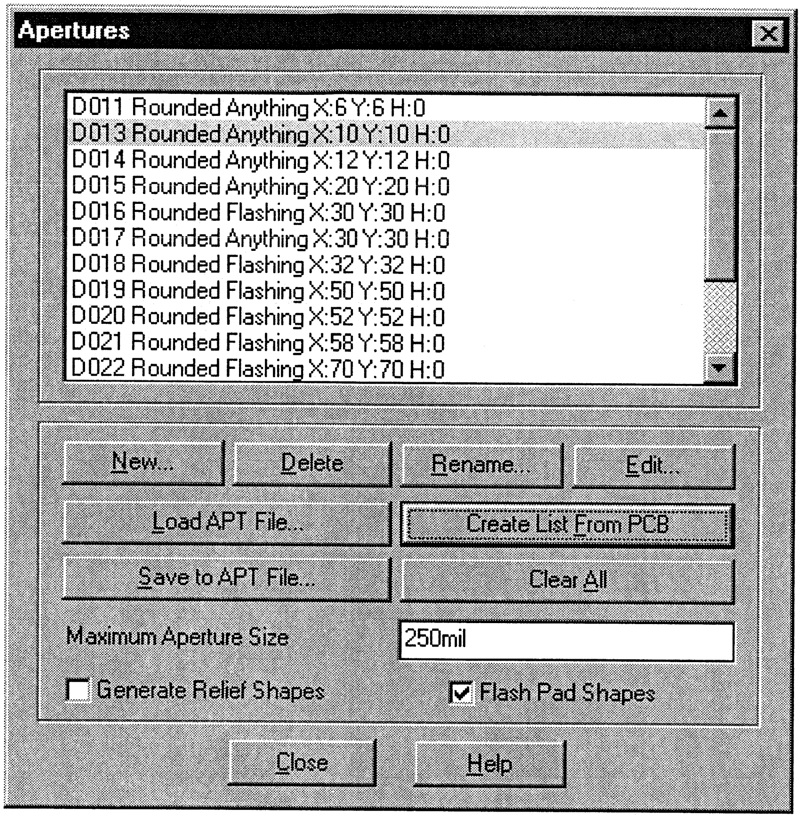
The apertures that will be used to translate your PCB file into a set of Gerber files are stored in a file with the extension .APT. Apertures can be regarded like plotter pens. Aperture descriptions include a shape, such as a 50mm square, and use – flash, stroke or anything (either flash or stroke).

Before you can generate a Gerber file, you can either load an aperture file that matches the capabilities of the target plotter, or you can let Advanced PCB automatically create an aperture file, extracted from the primitives (tracks, pads, etc.) in the current PCB file. When targeting a vector plotter, the apertures in the .APT file must correspond to the apertures available on the actual aperture wheel or slide to be used. The photoplotting bureau will supply the aperture table to suit their vector plotter. Raster plotters use the aperture file to translate draft codes directly into an image "map". If the target plotter is a raster device, you can generate the apertures from the PCB and supply the generated aperture table with the Gerber files. Your photoplotting bureau will supply the required file generation details.

When you use an existing aperture file, Advanced PCB scans the primitives (tracks, pads, etc.) in the PCB file and matches these with aperture descriptions in the loaded .APT file. If there is no exact match of aperture to primitive, Advanced PCB will automatically "paint" the primitive with a suitable smaller aperture. If there is no aperture suitable to "paint" with, a .MAT match file will be generated listing the missing apertures and Gerber file generation will be aborted.

If targeting a vector plotter, use primitives (track and pad sizes and shapes) for which there is a matching aperture. If the designer is familiar with the aperture set supported by the target photoplotter and tailors the choice of objects placed on a PCB design accordingly, the photoplotter will be able to faithfully reproduce the file in the most efficient manner.

Loading and Editing Apertures



All aperture manipulation is done in the Apertures dialog box.

Select the Design-Aperture Library menu item to load, create, or edit the apertures used by the photoplot output routines. When you choose this menu item the Apertures dialog box will pop up. Any currently loaded apertures will be listed. These apertures would be used if you generated a photoplot at this time. If there are no apertures loaded, the list is empty. Options allow you to work with new or existing aperture files. Changes are applied to the aperture file currently loaded into memory. These changes do not become permanent until you use the Save to APT File button.

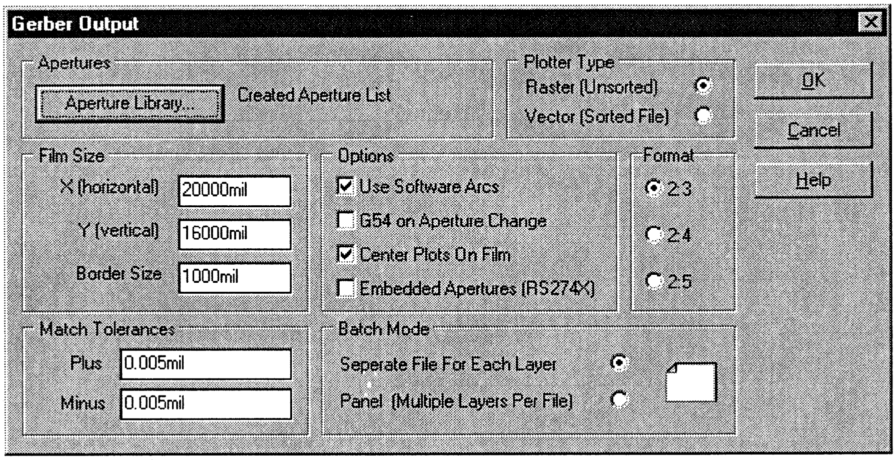
➡ You can define up to a maximum of 1000 different draft codes, in the range D00-D9999, although some of these codes (usually D00-D09) may be “reserved” when targeting some plotters, so use of these codes is not generally recommended.

➡ When creating new apertures do not include the “D” in the Draft Code Number dialog box.

Gerber Output Setup

Setting up to generate a Gerber plot file is similar to setting up pen plots or prints. The previous section (Plot/Print) provides an introduction to basic plotting and printing concepts. We recommend that you review this section if you haven't already done so.

To generate Gerber files, select the File-Setup Printer menu item (shortcut; F, R) to open the Setup Printer dialog box. Select the Protel Gerber RS274 on File output option. Press the Layers button to setup the required output layers. Layers are discussed in detail earlier in this chapter. Press the Options button in the Setup Printer dialog box to pop up the Gerber Output dialog box.



Prior to generating Gerber plot files it is vital that the designer understand the special requirements and limitations of the target photoplotter.

Apertures

If there is no aperture file listed, press the Aperture Library button to pop up the Apertures dialog box. Apertures are discussed earlier in the section, setup the apertures as required.

Plotter Type

These options process the generated Gerber file for efficient plotting on either a vector or raster machine. The Vector Plot option will slow down plot file generation, and the plot will be highly optimized for these machines. Using this option saves (expensive) bureau plotting time where you normally pay an hourly charge. The Raster Plot option leaves the plot unsorted and the plot file is more quickly generated. There is no advantage in sorting a plot file for a raster plotter.

Film Size

These fields set up the film size used by the target photoplotter. The default is twenty inches by sixteen inches. Your bureau will tell you the available sizes, but be careful to specify the x and y values in the proper orientation. Set the border size required on each film. This will define the border around each plot, so there will be a space at least equal to twice the border size between panelized plots. The system will spread the plots out to equalize the borders around all the plots if there is more space available.

Use Software Arcs

Some photoplotters do not support the Gerber arc drawing command, where the arc is generated by the plotter – also known as "hardware arcs". Advanced PCB can generate a series of short line segments to draw the arcs – referred to as "software arcs". Hardware arcs are preferable, if supported by the target plotter. Consult with your photoplot bureau. Hardware arcs are used for arcs that are in multiples of 90 degrees only – as the Gerber language can only describe arcs of one-to-four quadrants. For any small arc angle (less than 90 degrees), software arcs are generated, regardless of whether or not Use Software Arcs is selected.

G54 On Aperture Change

Some early model photoplotters require a specific G54 command to be inserted in the control code before every aperture change command. Consult with your photoplot bureau first, and then turn this on or off as they suggest.

Center Plots on Film

If this is enabled Advanced PCB will calculate the size of the plot and calculate the Gerber file coordinates such that the plot will fall in the center of a film of the specified size. If this option is disabled the Gerber file coordinates are calculated relative to the Absolute Origin (bottom left of the workspace). Ensure that the design is located in a suitable position in the workspace for the chosen film size when using this option.

Embedded Apertures (RS274X)

If this option is enabled the apertures required to produce each photo-tool are written into the Gerber file, according to the RS274X Gerber file specification. If embedded apertures are used a separate aperture file does not need to be sent with the Gerber files. Check if your photoplotting bureau supports RS274X embedded apertures.

Format

Specifies the numeric format for the coordinates in the Gerber file. The syntax is A:B, where A is the number of digits before the decimal point and B is the number of digits after the decimal point. These are in inches. This means for the 2:3 format, the largest number in the Gerber file could be 99.999 inches and the smallest number could be 0.001 inches (1 mil). Metric Gerber files are supported. In metric 2:3 (inches) becomes 5:3 (mm), 2:4 becomes 5:4 and 2:5 becomes 5:5.

Match Tolerances

While generating a photoplot file, Advanced PCB looks for an aperture to match each item in the plot layers. If no exact match is available in the current aperture list, the Match Tolerances are checked to see if a slightly smaller or larger aperture can be used.

For example, circular 62 mils pads will be efficiently plotted using a 62 mils aperture. But what happens if the photoplotter does not have a 62 mils aperture? Matches involve accepting an aperture that is close to the size of the original primitive. For example, a 60 mils round aperture might be available, and be close enough in size as to be acceptable. A Match Tolerance Minus value of "2" would allow this aperture to be used. The pads on the actual PCB would then end up being 60 mils.

If no exact aperture exists, or none fall within the tolerance range, Advanced PCB will attempt to "paint" with a smaller aperture to create the required shape. This requires that a suitable smaller aperture is available, and that this aperture can be used for "painting". Some photoplotters restrict the "use" of individual apertures to either "flash" or "stroke". Other plotters allow unrestricted use. This difference in plotter capabilities is one key reason why it is important that Gerber plots be planned for the target plotter.

➡ Match tolerances will only be of use if you are working with a fixed, or supplied aperture file. They will not be required if the apertures have been created from the PCB. If match tolerances are not required they should be left at the default of 0.005 mil.

Batch Mode

You can choose Separate File for Each Layer and each layer will be centered on a separate sheet of film. Use the Panel Files option to automatically panelize the layers in your plot onto a single sheet of film. If there is insufficient film space for your layers (per your Film Size settings above) the system will generate as many files as required for the plots you have specified. The first file will be called *filename*.P01, the next one .P02 and so on.

The Plot Generation Process

Check that the following points are satisfied before generating the Gerber files:

1. You have targeted a specific photoplotter and are aware of its output capabilities and file format requirements.
2. A list of suitable apertures is indicated in the Gerber Output dialog box. This list will be a created or loaded aperture file for the target plotter, or one created directly from your PCB file (raster plotters only).
3. You have pre-determined the contents of each plot layer (pads "on" or "off," etc.).
4. The Gerber Output dialog box is correctly configured.

➡ If you are uncertain about any of these points, we recommend that you review the preceding information, or contact your photoplot bureau or PCB manufacturer. Plotting bureaus and manufacturers are good sources of general design advice, which can save hours of frustration, and prevent costly mistakes.

In the Printer Setup dialog box press the print button to generate the Gerber files. The Photoplot Output Filename dialog box will pop up. Note that it offers the design filename, with no extension. Predefined extensions are automatically added. Refer to the end of this chapter for a list of file extensions.

As the plotting sequence begins, the board is redrawn to fill the whole screen. There are a number of stages to the Gerber generation process, which can be identified by the messages on the Status Bar;

Removing Duplicates - if this option is enabled (Options Tab of the Preferences dialog box) the design will be scanned for identical primitives in identical locations. These duplicates will not be written to the Gerber file. Use this option if the target photoplotter is a vector plotter. It is not required if the photoplotter is a raster plotter as these machines automatically deal with duplicates. Enabling the remove duplicates option can lengthen the Gerber generation process considerably for a large design.

Building the Shape List and Matching Apertures - will then be displayed and a number of shape descriptions will quickly flash at the extreme left end of the line. This lists unique shapes that are contained in current board file, which will be matched to the available apertures.

Checking Internal Pads and Vias - If the "Include Unconnected Mid Layer Pads" option has been disabled, this message will appear as the mid layers are searched and unconnected pads and vias are removed.

Sorting - If generating a vector photoplot, the sorting routines runs.

Once sorted, the plot file is generated for each enabled layer. All primitives are drawn on the screen, the order of display matches the order which they will be plotted. At the end of the process, the prompt "Photoplot is Finished" will be displayed.

➡ Advanced PCB does not include rotated rectangular shapes in the Gerber generation process. If rotated rectangular shapes are present in the design they will be painted using a suitable round aperture when the Gerber file is generated. This can be verified when Gerber files generated from designs containing these objects are imported.

Identifying Gerber Plot Files

Gerber plot file names are automatically appended with a unique extension that identifies layer and plot type. For example, the Top layer plot of a file called "TEST" will be saved as "TEST.GTL," to indicate "Gerber-Top layer". Because each design can generate several plot files, these tags help identify sets of output files.

➡ Using a unique extension for different layers and file types allows you to retain a common filename (e.g. TEST) to simplify identification later.

We recommend that you follow this convention which conforms to general industry practice. Advanced PCB Gerber extensions, added automatically as you generate plot files are;

Top Overlay	.GTO
Bottom Overlay	.GBO
Top Layer	.GTL
Bottom Layer	.GBL
Mid Layer 1, etc.	.GM1
Power Plane 1, etc.	.GP1
Mechanical Layer 1, etc.	.GF1
Top Solder Mask	.GTS
Bottom Solder Mask	.GBS
Top Paste Mask	.GTP
Bottom Paste Mask	.GBP
Drill Drawing	.GDD
Drill Guide	.GDG
Pad Master, Top	.GPT
Pad Master, Bottom	.GPB
Keep Out Layer	.GKO
Gerber Panels	.P01, .P02, etc.

Gerber plotting summary

The cost of generating photoplots is generally determined by the time required to plot a given piece of artwork. If the designer matches, wherever possible, the output capabilities of the plotter, the cost of the plots will be less.

Ideally, the designer would use only pads, vias and tracks types that matched the available apertures for the target plotter. This minimizes the amount of filling (or "painting") required to complete the plot and guarantees that the plotted will exactly duplicate the edited file. If you are targeting a raster plotter, you can allow Advanced PCB to generate you aperture list directly from the completed layout by using the automatic aperture creation option in the Apertures dialog box. If your targeting a vector plotter, where apertures are restricted to the actual wheel (or slide) selections, you work, wherever possible, with primitives that match the available selections.

Working with the available apertures at the edit level also speeds up Gerber file generation. However, if aperture selection is restricted by your target photoplotter, Advanced PCB will automatically match all the primitives on each plot layer with your aperture list. It will also optimize the plot to make the best use of the available apertures when plotting.

NC Drill

Introduction

Advanced PCB allows you to produce output files for Excellon format numeric control drilling equipment, directly from the current Advanced PCB document window. NC drill files are binary files, read directly by the drilling equipment, which include coordinates and drill tool assignments for each hole in the PCB file.

Advanced PCB generates both ASCII and Excellon format drill files. If both plated and non-plated holes are present in the design they are assigned to different tools.

About NC Drill Files

The use of numerically controlled (NC) drill equipment provides several advantages for board designers and manufacturers. Supplying the fabrication house with an NC file saves the cost of manually programming the drilling process and removes the danger of missing holes on complex boards. There are however a number of potential problems to be considered before generating NC drill files.

➡ Board artwork must be highly accurate, preferably photoplotted, when using NC equipment. NC holes will be accurately positioned by the drilling equipment. If your plot is not accurate, holes will not align with the pad centers over the area of the board. Consider a 300mm long printed circuit board. If the plotter has a scale (linear) error of 0.3%, then the total error on the board could be up to 0.9 mm. If the drill is manually programmed from the plot master then the error would not matter, since it would be very small over the length of any single component. If the drill was driven directly from the NC Drill a cumulative drilling error of up to 0.9mm would result.

Generating NC Drill Files

Refer to the *NC Drill* topic in the *Reports* chapter for details on how to generate the NC drill files.

Reports

The following reports are available in the PCB Editor.

Selected Pins

All the selected pins are listed in the Selected Pins dialog box. This provides a convenient way to verify the connections within a net. Use the Mask to narrow the list of pins being displayed. Press the OK button to generate a report file.

Board Information

General Tab

This Tab includes; a tally of each primitive type used in the design, the dimensions of the board (based on the most distant primitives in the workspace), the total number of pad and via holes and the number of clearance errors on the board.

Components Tab

Lists all components currently placed on both top and bottom layers. Includes designator and comment, if any.

Nets Tab

Lists all currently loaded nets, by name. Includes a tally of nets loaded.

Pwr / Gnd

Press the Pwr/Gnd button to pop up the Internal Plane Information dialog box. Each Tab lists the nets connecting to that plane. Select a net to display the pins on that net which are connected to this plane.

Bill of Materials

Select the Reports-Bill of Materials menu item (shortcut; R, I) to launch the PCB BOM Wizard. This Wizard generates a Bill of Materials (BOM) for the active design, which is automatically loaded into the Spread Sheet Editor. If the Wizard does not launch you will need to install the PCBBOM Wizard Server. Refer to the chapter *A Quick Tour of EDA/Client* for tips on installing a server.

Project Hierarchy

The Reports-Project Hierarchy process launcher (shortcut; `R, P`) generates a listing of currently open documents, reflecting any project structure that may exist between open documents. This report is output in ASCII text in the following format:

```
Project Hierarchy Report For C:\CLIENT\DESIGNS\RTD.PCB
C:\CLIENT\DESIGNS\RTD.PCB
Project Hierarchy Report For C:\CLIENT\DESIGNS\RTD.PRJ
C:\CLIENT\DESIGNS\RTD.PRJ
    C:\CLIENT\DESIGNS\RTDADC.SCH
        C:\CLIENT\DESIGNS\RTDINTA.SCH
        C:\CLIENT\DESIGNS\RTDINTB.SCH
```

The report will be opened automatically when the Reports-Project Hierarchy menu item is chosen, using the text editor specified in the Setup Run Options dialog box (Text Expert is the default text editor).

Netlist Status

For each net this report lists; the layers used for routing and the total net length.

NC Drill

Advanced PCB generates both ASCII and Excellon format drill files. Select the Reports-NC Drill menu item to automatically generate the drill files, using the current hole attributes for all pads and vias in the active PCB document window. If both plated and non-plated holes are present in the design they are assigned to different tools. Three types of files will be generated:

1. The .DRR file is a report file that lists the files generated for each layer pair. For each of these it details; the tools used, the hole sizes in both metric and imperial, the number of holes and the tool travel.
2. Excellon format file with the extension .DRL. If the design is multi-layer and blind and buried vias have been used, there will also be a drill file for each fabrication layer, with the extensions DR1, DR2, etc.
3. An ASCII version of each of the Excellon format files with the extensions .TXT, TX1, TX2, etc.

You should supply all of these files to your board fabrication house, together with a print out of the .DRR file.

As mentioned above, if there are blind or buried vias in your design Advanced PCB will generate additional drill files with modified extensions that identify each layer pair. For example if you have a board with four signal layers, and you have use blind/buried vias then you will receive three different NC drill file sets. One set of EIA and ASCII for each of the two layer pairs and one set for through-hole vias. You should supply all of these to your board manufacturer along with clear instructions detailing the layer/file assignments and order of assembly.

The drill control files (Extension DRL, DR1, etc.) are written in the EIA character (binary) format in the EXCELLON language. The data is specified in mils or mm with trailing zero suppressed. Drill tools are numbered from 1 up to 255.

Pick and Place

This report contains information which can be used by a pick and place machine. It is generated in both text and CSV formats. The report includes;

Mid X, Mid Y

The location of each component mid point, referenced to the current origin. The component mid point is calculated as the mid point of the set of component pins.

Ref X, Ref Y

The location of each component reference point, referenced to the current origin.

Pad X, Pad Y

The location of pin 1 of each component, referenced to the current origin.

TB

Layer component is mounted on, (T)op or (B)ottom.

Rotation

Rotation of component, relative to its original orientation in the library.

Measure Distance

Use this feature when you need to perform an accurate measurement in the PCB workspace. After selecting the Reports-Measure Distance menu item you will be prompted to Select Measure Start Point. Position the cursor, click LEFT MOUSE, move the cursor to the end point and click LEFT MOUSE again. A dialog will report; the point-to-point distance measured, the X distance and the Y distance. Change the snap grid if you cannot accurately position the cursor at the required points (shortcut; G). You may need to temporarily disable the Electrical Grid if you find that the cursor snaps to the center of electrical objects.

Linking to Advanced Schematic

Netlists

The Netlist that you create from your schematic and load into Advanced PCB is the primary link from Advanced Schematic to Advanced PCB. This file passes the component information and the net information from your schematic into the PCB design environment, allowing you to layout and route the board with the confidence that what you are working with is an accurate representation of what you specified in the schematic.

The Protel Design System supports full *forward annotation* of design changes made in Advanced Schematic into Advanced PCB. Updated netlists can be used to make engineering changes to fully or partially routed PCBs. Refer to the chapter *Working With a Netlist* for details on how to forward annotate your design.

Routing Directives

The Protel 2 netlist format can carry net specific design information, such as track width and via size, from the schematic to the PCB. Add this information in your schematic by attaching a PCB layout directive to the net. The routing directive information is translated into design rules when the netlist is loaded into Advanced PCB. For information about using PCB layout directives refer to the *Advanced Schematic User Guide.*

Bi-directional Cross Probing

Cross Probing is a technique of locating matching objects in different documents. For example, you are studying a net in the schematic and would like to examine where it has been routed on the PCB. To cross probe this net from the schematic to the PCB, press the cross probe button on the Schematic Sheet Editor main toolbar. When you click on the net name on the schematic sheet the net will highlight on the PCB. Tile the document windows to easily observe the highlighted net.

Advanced PCB supports cross probing of most objects, including nets, pins and parts to and from Advanced Schematic. You can also cross probe to documents open in other EDA/Client Editors, such as the text editor. If required you can cross probe to a specific object kind and also to a particular editor. Cross probing is performed by the Client:CrossProbe process. For information on the Client:CrossProbe process parameters press the Info button when you are setting up the toolbar button to launch this process.

For further reading on processes and parameters refer to the *Understanding Processes* chapter. For information and examples about creating a toolbar refer to the *Resource Management* chapter.

Re-Annotation

In Advanced PCB, the components on the board can have their designators re-assigned on a positional basis (they can be *re-annotated*). If this is done, all the designator changes are written into a *was/is* file. This is a simple ASCII file which lists what each designator *was* followed by what it now *is*.

To re-annotate the PCB, select the Tools-Re-Annotate menu item. The Positional Re-Annotate dialog box will pop up. In this dialog box you specify the starting point and the direction of annotation. The four directional options work as follows;

1. Numbers from bottom to top, starting in the lower left corner.
2. Numbers from top to bottom, starting in the upper left corner.
3. Numbers from left to right, starting in the lower left corner.
4. Numbers from left to right, starting in the upper left corner.

The annotation routine uses a "band width" of 50 mils, stepping across in bands as specified by each option. For example, the first option is titled "Ascending X then Ascending Y. This option sets the first band 50 mils wide in the X direction, then scans up the band in the Y direction. It then sets the next band in the X direction and again scans up the band in the Y direction, and so on. The center of the component body is used as its location reference.

➡ When a board is re-annotated designators are re-assigned so that there are no gaps in the designation sequence.

The re-annotation results are written to an ASCII file, *filename*.WAS.

After re-annotation has been performed on the PCB, the designator changes must be *Back Annotated* into the schematic. Refer to the Advanced Schematic User Guide for information on back annotating the schematic.

➡ Once the schematic has been back annotated, the netlist no longer matches either the schematic or the PCB. It is advisable to always keep all three files that hold designator information consistent. After re-annotation in Advanced PCB, immediately do a back annotation in Advanced Schematic and create a new netlist.

➡ Never re-annotate more than once in Advanced PCB without then doing a back annotate in Advanced Schematic. If you do, the *was* list of designators in the was/is file will no longer be a list of the schematic designators, but a list of your last re-annotation in Advanced PCB. A total miss match of designators can occur, with a painful process of manual recovery. If you are unhappy with the results when you re-annotate in Advanced PCB, use Undo.

Advanced Topics

Understanding Processes

The Advanced PCB environment, whether creating and editing PCBs, or working in the library editor, consists of two basic features: *objects* that are placed in the workspace to build up the design, and *processes,* which are used by the system or the user to create, modify, save and report on the objects.

What is a Process?

A process can be thought of as the software executing a sequence of jobs. This job may be something simple, like refreshing the screen, or it may be more complex, like placing a polygon plane.

Any action or operation that is performed in Advanced PCB is carried out by a process. When the Place-Track menu item is selected, the *PlaceTrack* process is launched. The track is then placed by the user interacting with the *PlaceTrack* process. Selecting the Reports-Netlist Status menu item launches the *ReportNetlistStatus* process, which generates an ASCII file with information about the routing of each net. Menu items, toolbar buttons and shortcut keys launch a process and are called Process Launchers. The action, or job, is performed by the process.

Each process is identified by a unique *Process Identifier*. The process identifier includes the server name and the process name, separated by a colon. For the two process names mentioned above, the syntax is;

PCB:PlaceTrack

PCB:ReportBOM

A definition of each process provided by EDA/Client and each server can be found in the relevant On-line Help file. Each of the process launcher Edit dialog boxes includes an Info button. Press this to pop up the Help file with a description of that process.

Launching a Process

PCB processes are *launched* by passing the process identifier to the PCB server. The server then carries out that process. When the PCB server receives the process identifier it first checks that it is valid and then invokes the process. To pass the process identifier to the server, a process launcher is used. Process launchers include;

- menu items
- toolbar buttons
- keyboard shortcut keys

When a menu item is selected, such as Place-Track, the process identifier linked to that menu item is passed to the PCB Server, along with any parameters that have been defined.

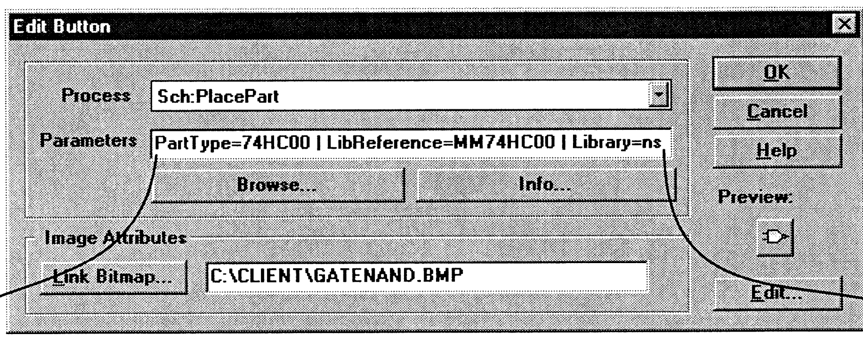
Advanced PCB provides a powerful productivity enhancement by allowing *any process available in Advanced PCB to be mapped to any process launcher*. This allows you to define your own menus, your own toolbars and your own shortcut keys. For tips on editing menus, toolbars and shortcut keys refer to the chapter *A Quick Tour of EDA/Client*. For information about creating and configuring resources refer to the *Resource Management* chapter.

Process Parameters

Processes in Advanced PCB are *parametric*. A parametric process supports the passing of parameter values to the process. This means you can instruct the process on how it is to behave at the same time as you launch the process.

Parameter values can be passed when the process is launched by entering them into the Parameters text field of a process launcher editing dialog box (e.g., the Edit Menu dialog box). They can also be passed by a macro.

The following figure shows a toolbar button, which launches the Sch:PlacePart process. It also passes the parameters shown in the figure which specify;



PartType=74HC00 | LibReference=MM74HC00 | Library=ns03-c.lib | Orientation=0 | Designator=U?|$Description=Quad 2-input positive Nand gate

- Library = The name of the library that the component is in. If the library is not in the standard location, include the full path.
- LibReference = The name of the component in the library.
- Designator = The designator you want to give this component.
- PartType = The part type or comment you want to give this component.
- Orientation = The orientation of the component when it appears on the cursor.
- $Description = This parameter allows you to create your own button tool tip.

When the button is pressed the 74HC00 specified by these parameters will appear on the cursor, ready to be placed.

The syntax for passing process parameter values via the Parameters text field is *parameter = value*. Each parameter is then separated by the vertical bar (or pipe) symbol. It is not necessary to list parameters in any order, nor is it necessary to pass every possible parameter.

For instant access to information on the parameters of a process press the Info button in the Edit Menu, Edit Button or Edit Keyboard Shortcut dialog boxes.

For information on passing parameters from macros, refer to the *Macros* help file or on-line manual.

Resource Management

Advanced PCB allows you to design a Printed Circuit Board on a computer. To do this, components are placed from libraries and tracks are routed to connect these components. Output files are then generated, which can be used by a PCB manufacturer to create the PCB.

All the 'doing' functions in Advanced PCB, such as placing a component or a track, changing the zoom level, redrawing the screen and so on, are performed by processes. To access these processes, Advanced PCB provides a set of resources.

➡ Details about each process can be found in the On-line Help system.

Resources are the mechanism through which you launch processes. To route a track you can press the routing button on the PlacementTools toolbar. The button launches the process and is called a process launcher. The set of buttons, or toolbar, is a resource. Alternatively, you could select the Place-Track menu item. The menu item is the process launcher, the entire menu is a resource. The other method of invoking a process is to use the third set of resources available, shortcut keys. The shortcut key is the process launcher, the shortcut key list is a resource.

All the resources available in Advanced PCB are fully customizable. New menus, toolbars and shortcut key lists can be created and all can be modified.

Advanced PCB Resources

Advanced PCB has three types of resources; menus, toolbars and shortcut keys. All Advanced PCB processes can be launched from any of these resources (in fact any process available in the Client environment can be launched by any resource in any server).

Menus

Advanced PCB menu items are organized to be as consistent with the Windows model as possible. This means that standardized operations, such as opening and saving files, printing or using standardized Windows editing operations such as Cut or Paste are handled in Advanced PCB using the same methods that other Windows applications use. This makes the software more productive in an integrated environment, where you are typically working with a number of Windows applications.

Menus can be edited and new menus created. Any process currently available in EDA/Client can be linked to any menu item. Refer to the chapter *A Quick Tour of EDA/Client* for tips on how to edit menus.

Pop-up menus

Advanced PCB includes special shortcut key assignments for accessing menus. For example, pressing E will pop up the Edit menu, pressing M will pop up the Move sub-menu. This provides a convenient way to access menus and sub-menus directly from the keyboard. The underlined letter in a menu is the shortcut key for that menu or menu item.

Toolbars

All toolbars in the EDA/Client environment can be fixed to any side of the workspace or set to be floating, where they can be re-positioned anywhere in the environment.

New toolbars can be created and existing toolbars edited, linking any of the processes currently available in EDA/Client to any button. There is an example of how to create a new toolbar later in this chapter.

Assigning frequently used processes to tool buttons provides a convenient shortcut, which can speed editing of a PCB or component creation in the library editor.

Keyboard Shortcut Keys

Advanced PCB comes with two keyboard shortcut key lists, one for the PCB Editor and one for the Library Editor. These can be edited and new shortcut key lists can be created.

Keyboard shortcuts can include key combinations, including CRTL, SHIFT and ALT in combination with either one or two keys.

➡ As well as being able to map keyboard shortcut keys directly to processes, keys can also be mapped to menus. If a key has mistakenly been mapped to both a menu and directly to a process, then menu mapping has priority. Refer to *A Quick Tour of EDA/Client* for clues on how to define shortcut keys.

Default Resources

The default resources provided with Advanced PCB are defined in a resource file. Advanced PCB, like all servers that run in EDA/Client, has a default resource file with the file extension RCS. This file contains the definitions of the default menus, toolbars and shortcut key lists for both the PCB Editor and the PCB Library Editor. These resources are known as system resources and can not be removed from the environment. Customized resources are *not* stored in this file.

When the Advanced PCB server is first installed, the resource definitions are read from the Advanced PCB RCS file and added to the Client RCS file. Any modifications or additions made to Advanced PCB resources are done to the copy of the resources in the Client RCS file. It is not necessary to edit this file as all resource customization can be performed from within Advanced PCB.

EDA/Client Resources

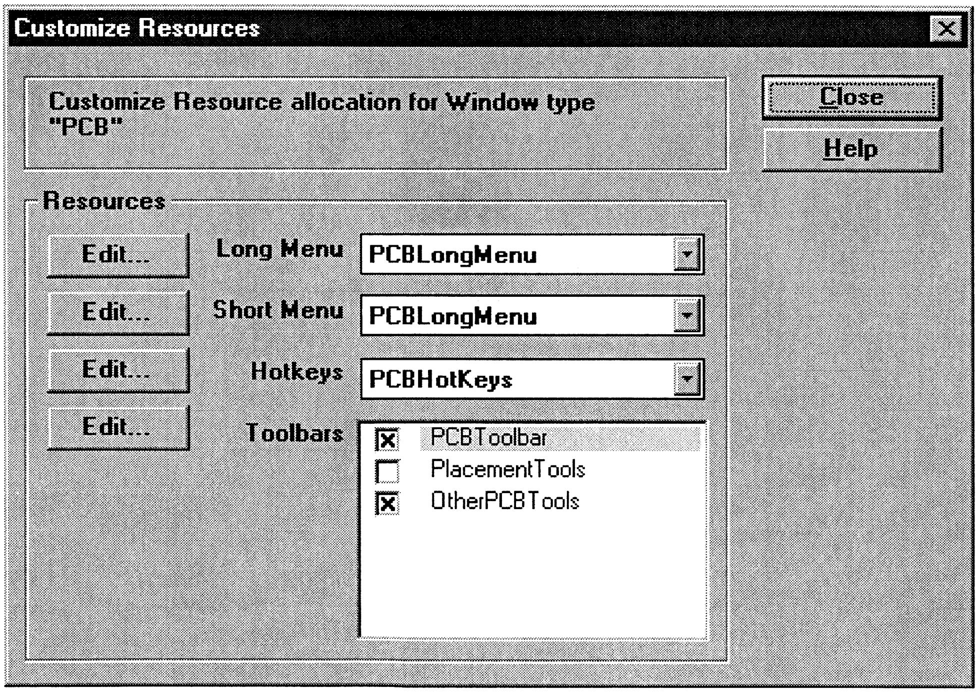
The total pool of resources available in the EDA/Client Server environment includes those provided by Client, as well as any resources provided by installed servers, such as Advanced PCB. The first time a server is installed, its default resources are added to the Client RCS file. Modifications to the resources of any server are recorded in the Client RCS file. As all user customization is stored in this file it is a good practice to backup this file. The CLIENT.RCS file is in the Windows directory.

Managing Resources

Managing resources can be broken down into three areas, *customizing*, *editing* and *configuring* resources.

Customizing Resources

When you would like to select a different set of shortcut keys, change to your custom menu or display a particular toolbar, you need to *customize* the resources.

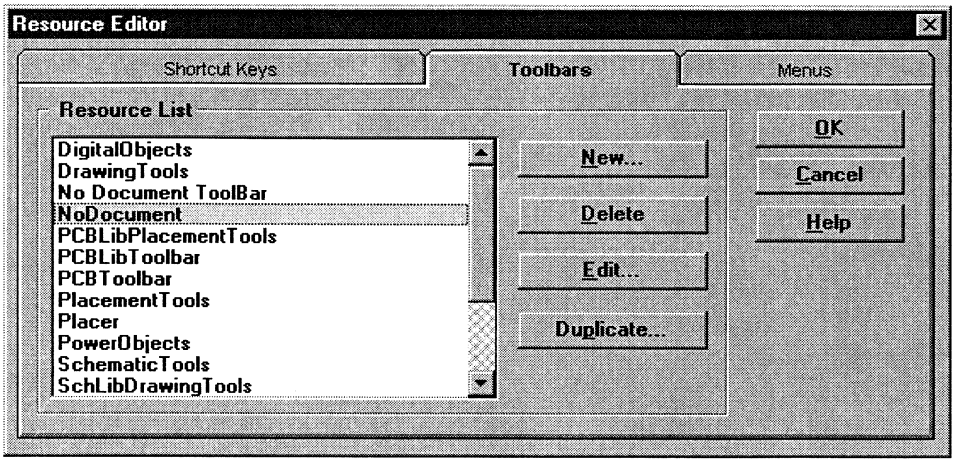


Selecting the Client menu-Customize menu item pops up the Customize Resources dialog box. This dialog box allows you to customize the resources currently available to the *active document editor*. If a PCB is the active document then the Customize Resources dialog box will give you access to the resources currently available to the

PCB Editor. To customize the resources of the active document editor, select the Client menu-Customize menu item.

Customization includes; selecting another resource from the list of resources available to this document editor (perhaps choosing your own specialized menu), editing a selected resource and toggling the display state of toolbars. To add or remove resources to a particular document editor refer to *Configuring Resources* below.

Editing Resources



Selecting the Client menu-Resource Editor menu item pops up the Resource Editor dialog box. This dialog box gives you access to the entire pool of resources currently available in the EDA/Client Server environment. Here you can create resources and remove resources from the environment. It is also possible to edit any of the resources currently available. To add or remove resources to a particular document editor refer to the *Configuring Resources* topic.

➡ To edit a menu, simply double click anywhere in the menu bar. To edit a toolbar simply double click on a toolbar. To edit the current shortcut key list, select the Client menu-Edit Shortcuts menu item.

Example: Creating a New Toolbar

There are two way of creating a new toolbar. An existing toolbar can be duplicated and then modified, or a new one can be built up. This example shows how to build up a new PrimitiveTools toolbar.

1. Select the Client menu-Resource Editor menu item.
2. Click on the Toolbar Tab and press the New button.

The Edit Toolbar dialog box will pop up. The Process List at the left will most likely display a list of processes for all installed servers. You will use the Filter to quickly locate the processes required for the new toolbar.

3. In the Filter field of the Edit Toolbar dialog box type in the word *place*.

The Process List will now display all process identifiers that start with the word *place*. Note that the server name before the colon (:) is not considered by the filter. The * (any characters) and ? (any single character) wildcards can be used in the filter.

4. Select the PCB:PlaceString process identifier and click the Add Process to Toolbar button (shortcut: double click LEFT MOUSE).

A blank button will appear in the Toolbar window with the PCB:PlaceString process identifier next to it.

5. Double click on this new button to pop up the Edit Button dialog box.

Note the Browse and Info buttons. The Browse button allows you to browse for a different process to assign to this button, the Info button will pop up the help file for the current process. The Edit button will open the linked bitmap in the assigned Picture Editor (the default is Paintbrush). To assign a new Picture Editor select the Client menu-Run-Setup menu item. When you enter the name of the new Picture Editor application it may be necessary to enter the full path as well.

6. Press the Link Bitmap button to assign a new bitmap.

The Image File dialog will pop up. All supplied Button bitmaps are stored in the \CLIENT directory.

7. In the File Name field enter the letter "t" in front of *.bmp and press enter.

The Files field will now list all bitmap files starting with the letter "t".

8. Double click on the file text.bmp.

The Image File dialog will close, presenting the Edit Button dialog box again. Note that the text bitmap has been linked and the Preview button now shows the image of the letter T.

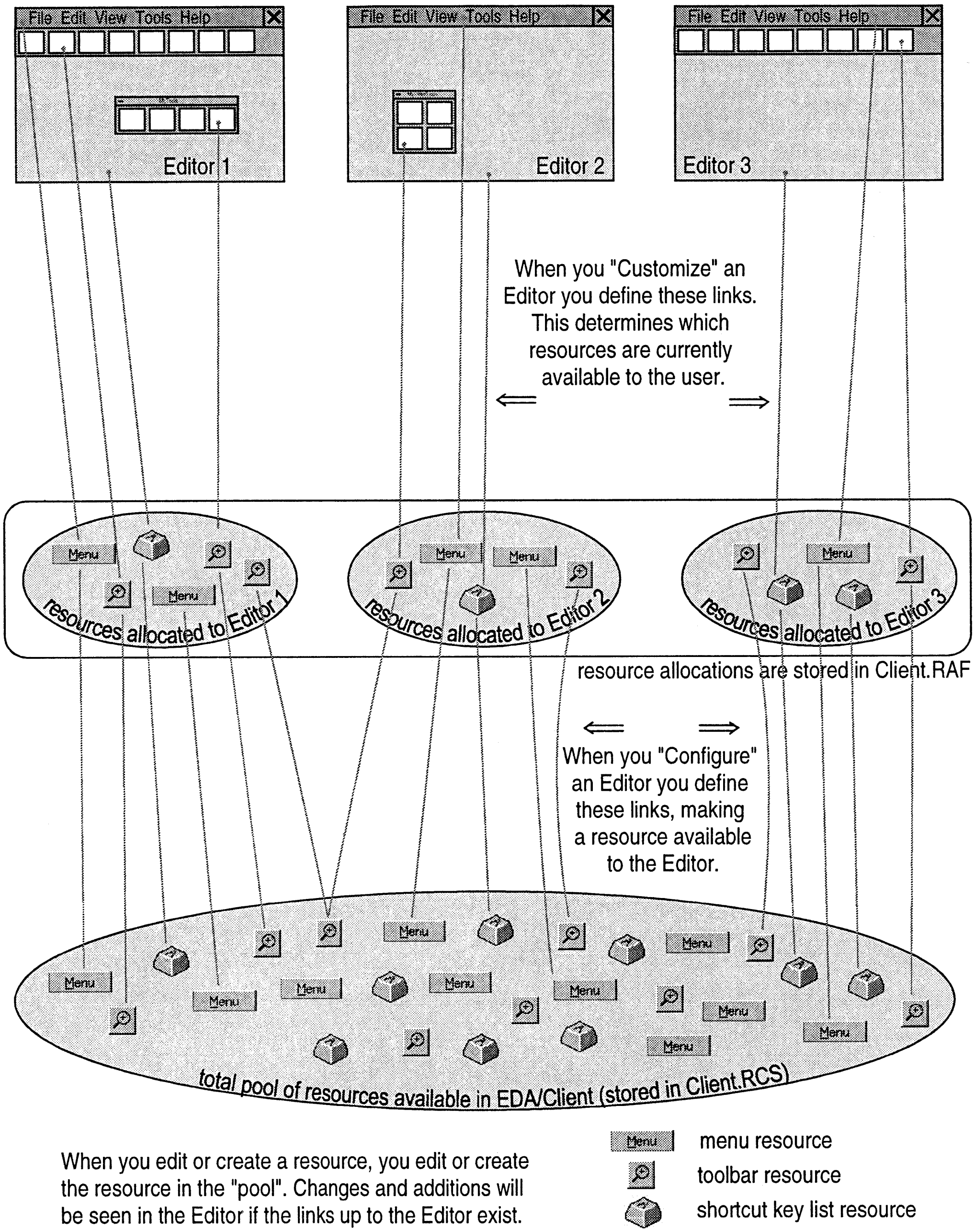
9. Click OK to close the Edit Button dialog box.

The Edit Toolbar dialog box will be pop up. Note that the bitmap is correct.

10. Repeat steps 4 to 9 to add the other five buttons using the following details.

Process Identifier	**Bitmap**
PCB:PlaceArcByCenter	arc.bmp
PCB:PlaceFill	rect.bmp
PCB:PlaceVia	via.bmp
PCB:PlacePad	pad.bmp
PCB:PlaceTrack	track.bmp

The Organization of Resources in EDA/Client



When you edit or create a resource, you edit or create the resource in the "pool". Changes and additions will be seen in the Editor if the links up to the Editor exist.

11. To make the toolbar appear as two rows of three buttons, highlight the PCB:PlaceFill button and press the Separator button. A blank space will appear above this button.
12. The last step is to name the toolbar. In the Name field of the Edit Toolbar dialog box type in the name, PrimitiveTools (or any other name you prefer).
13. Click the Close button to close the Edit Toolbar dialog box.

The new toolbar has been created and will appear in the Resource List in the Resource Editor dialog box.

14. Click OK to close the Resource Editor dialog box.

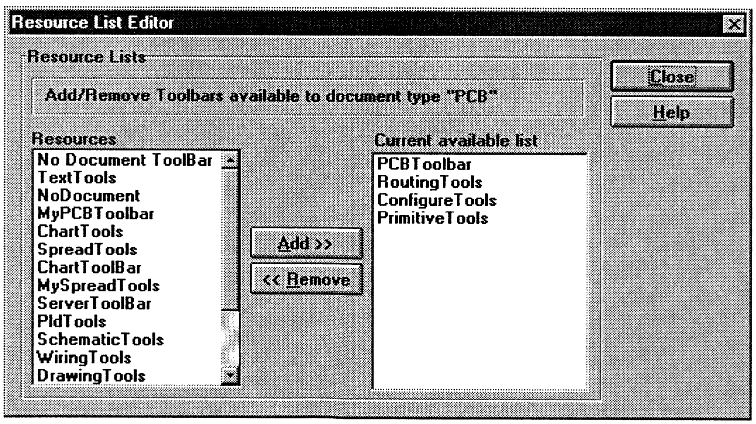
This new toolbar now exists in the pool of resources available in the EDA/Client environment. The next step is to make this resource available to the PCB Editor. This is detailed in the *Configuring Resources* topic below.

Configuring Resources

Configuring resources is the process of defining what resources out of the entire pool of resources in the EDA/Client environment are available to a particular Document Editor.

When you create a new resource with the Resource Editor, that new resource goes into the pool of resources available in EDA/Client. You then *configure* the Document Editor to make the resource available to that Document Editor. As a final step you *customize* the Document Editor to display or hide the resource in the Document Editor.

As an example, consider the case of configuring the resources available in the PCB Editor. If you intend to add your own resource, perhaps a toolbar, first create the resource with the Resource Editor (refer to the *Editing Resources* topic). You then *configure* the resources available in the PCB Editor, and then *customize* the PCB Editor to display the toolbar.



Add and remove resources to the selected document editor

Example: Adding Your New Toolbar to the PCB Editor

To add the new PrimitiveTools toolbar that was created in the *Editing Resources* topic you must re-configure the resources of the PCB Editor.

1. Select the Client menu-Servers menu item.

The EDA Servers dialog box will pop up.

2. Select the PCB server in the list of Installed Servers and press the Configure button.

The Configure Server dialog box will pop up. The Document Types field will list all the Document Editors that this server provides.

3. In the Document Types list select PCB and press the Toolbars button.

The Resource List Editor dialog box will pop up. The Resources list on the left will display all the toolbar resources currently available in the EDA/Client Server environment.

4. Locate your new PrimitiveTools toolbar in the list of Resources and double click on it. The name will now appear in the Current Available List on the right. Click the Close button to close the Resource List Editor dialog box, then close the Configure Server dialog box and the EDA Servers dialog box.

This toolbar is now available in the PCB Editor. If there are toolbars already visible the new toolbar will appear down the left of the workspace. Click on the toolbar and drag it into the workspace if you want it to float in the workspace.

If there are no other toolbars currently visible then you will need to make it visible. Changing the visibility of resources is a *Customizing* function.

5. Select the Client menu-Customize menu item.

The Customize Resources dialog box will pop up. Note that the new toolbar will be in the list of Toolbars.

6. Click in the check box to make the toolbar visible. Close the Customize Resources dialog box.

The new toolbar will appear down the left of the workspace.

Resetting Defaults

It is possible to return the resources of any server back to their defaults at any time. To do this, select the Client menu-Servers menu item. In the EDA Servers dialog box, select the appropriate server and press the Configure button. In the Configure Server dialog box press the Default button. Menus, toolbars and shortcut keys for the selected server will be returned to their default state.

➡ Pressing the Default button will return *all* the resources for *all* the document editors provided by *this server*, back to their defaults. It will not affect any user created resources. To avoid the possibility of inadvertently loosing changes made to resources, do not edit the default resources. Duplicate and rename a resource and then customize this resource. To create your own resources, refer to the Editing Resources section of this chapter. To customize your own resources, refer to the Customizing Resources section of this chapter. To make your own resources available, refer to the Configuring Resources section of this chapter.

Global Editing

As well as being able to edit the attributes of a single object, Advanced PCB also allows you to apply these edits to other objects of the same type. These may be other objects in the current component or in the current document.

Additionally, you can further define conditions that either extend or restrict global changes. For example, changes can be applied to all objects that are selected or all objects that are not currently selected. If desired, you can create a complex set of conditions for applying changes.

Virtually every attribute of an object can be globally edited. A simple example would be changing the size of pads associated with a specific component. In another instance you may wish to change the width of tracks for a particular net. These options (and many more) are possible with global editing. The possible applications for global changes are limited only by the imagination of the designer.

➡ The large number of global change options may make this feature appear somewhat complex at first. However, the principles of applying global changes are reasonably simple, once understood. When mastered, this feature can be an important productivity tool that can save a great deal of manual editing of a PCB.

Each object's dialog box may contain different options since each object type may have unique attributes.

Global Editing Strategies

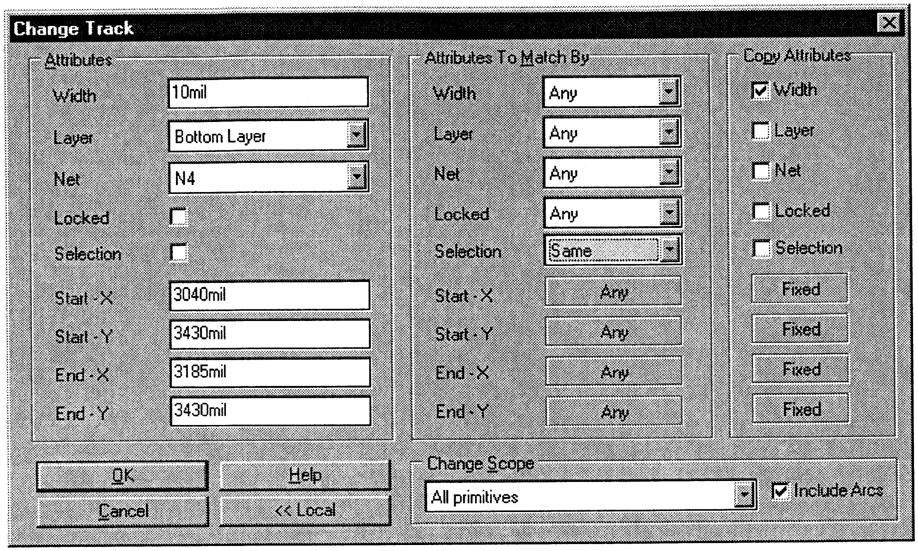
While the presentation of global change options may appear differently in the various object dialog boxes, the strategy used is always the same. This description will outline the approach to global editing.

Current Attributes

When you double click on an object, you are presented with the Change dialog box for that type of object. This dialog box contains the *current* values or settings of the object's attributes.

Change the attributes you would like to alter.

Pressing the Options button will extend the dialog box. It will now contain three distinct regions; Attributes, Attributes to Match By and Copy Attributes.



Note the three columns or regions each dialog box has when the Options button is pressed.

Attributes to Match By

In the center of the dialog box there will now be a column titled Attributes To Match By. In the Attributes To Match By column you define how to identify the other objects in the design which the global change is to apply to.

The Attributes To Match By column will contain either a choice field for each attribute or a text field which you can type in.

The choice field has three options: Same (apply global changes if this attribute is matched in the target object); Different (apply global changes if this attribute is not a match in the target object) and Any (the default) which applies the change irrespective of whether the attribute has the same value in both objects.

➡ If the Match By attributes are all set to *any* and the text fields contain the wildcard symbol (*), then the global change will apply to all objects of this type.

Use combinations of Match By attributes to define a particular set of objects to apply the change to.

Copy Attributes

The third column in the dialog box is titled Copy Attributes. This column will contain either a check box for each attribute or a text field which you can type in.

In this column you specify which of the attributes in the matched objects you want to copy the changes to, and if the attribute has a text field what new text value to copy to the matched objects.

➡ Any attribute can be globally changed if the object's dialog box includes a Match By and Copy field for that Attribute field.

Change Scope

The last parameter to set is the change scope. There are two options here; "All primitives" or "All FREE primitives". Advanced PCB identifies any primitive which is not part of a group object as a *free primitive*.

Examples of Global Changes

The following examples will give you some idea of the potential scope of global changes that can be performed on components and primitives:

Example 1 - Swapping Track Layers

To move all Top layer tracks to the Bottom layer, regardless of track width or selection status:

1. Double-click on any Top layer track to open the Change Track dialog box.
2. Set the Layer attribute to Bottom Layer.
3. Click the Options button to display the global edit parameters.
4. Under Attributes to Match By set Layer to Same. All other attributes should be set to Any.

This tells Advanced PCB to apply this change to *all* tracks on the *same* layer. Note that the match by condition is based on what the Layer was, not what you just changed it to.

5. In the Copy Attributes column the Layer attribute will have been automatically activated.

This tells Advanced PCB to copy the change made to the Layer attribute to all tracks that meet the Match By criteria.

6. Set the Change Scope to All FREE primitives. If your routing includes arcs then enable the Include Arcs option. Click OK or press ENTER.

The initial track that was edited will be changed to the Bottom layer first. The Confirm Global Change dialog box will then pop up.

7. Click YES to accept the global change.

All Top layer tracks will be swapped to the bottom layer. You may need to redraw the screen (press END) to refresh the display.

If you wish to change a particular net to another layer, select the Edit-Select-Physical Net menu item and click anywhere on the desired net. Now repeat the global edit process as described, except in the Attributes To Match By column set the Selection attribute to Same and all others to Any. Only the selected net will move to the Bottom Layer.

Example 2 - Changing Via Sizes

To change all vias on a board to 40 mils:

1. Double-click on any via to open the Change Via dialog box.
2. Type 40 in the Diameter field under Attributes.
3. Click on the Options button.
4. Set all the Attributes To Match By to Any (all vias will be changed).
5. In the Copy Attributes column the Diameter attribute will have been automatically activated.

Leave the other attributes disabled.

5. Set the Change Scope to All primitives and click OK or press ENTER.

The initial via selected will be redrawn with the new diameter and the Confirm Global Change dialog box will open.

6. Click OK.

All vias in the document window will change to reflect the new size setting.

Example 3 - Locking a Net

To prevent the autorouter from modifying a particular net that was manually routed, lock it in place. To do this;

1. Select the net. To select a net choose the Edit-Select-Physical Net menu item and click on the net.

All the track segments that make up the net will highlight in the selection color.

2. Double click on one of the selected track segments.

The Change Track dialog box will pop up.

3. Enable the Locked attribute in the Change Track dialog box.
4. Press the Options button.
5. In the Attributes To Match By column set the Selection attribute to be Same.
6. Set the Change Scope to All FREE primitives. If your routing includes arcs then enable the Include Arcs option.

The global change will be applied to all selected track segments that are not part of a group object.

6. In the Copy Attributes column the Locked attribute should be set.

This instructs Advanced PCB to copy the changes made to the Locked attribute of this track segment to all track segments that meet the Match By criteria.

7. Click OK.

The initial track selected will be changed and the Confirm Global Change dialog box will open.

8. Click OK to lock the remaining selected track segments.

Summary

The three examples above show the most basic application of the global change options. With care and planning you will experience significant productivity benefits from this powerful feature. However, the very power of these options can contribute to some unanticipated results – particularly when complex selections are globally edited.

When in doubt, it's always safest to De-Select All (X, A), then, rather than using the global edit to change the target objects, set it up to simply select the target objects. Visually confirm that the match by criteria has targeted the correct objects, then re-do the global edit, using Selection as the match-by criteria.

Use the auto-backup feature included in EDA/Client and always archive your design, particularly if you intend complex changes. Finally, remember that the Undo/Redo features can allow you to recover several operations, if required.

Import Options

Advanced PCB supports the importing of a number of different file formats. These include; Gerber files, AutoCAD DXF format files, p-CAD, PADS and Tango PCB files, and route files generated by the Specctra autorouter. To import one of these files select the File-Import menu item to pop up the Import File dialog box. Set the file Type as required to import the desired file.

Autotrax (*.PCB)

Advanced PCB imports Protel Autotrax files. All Autotrax design objects are supported.

DOS PCB 3 (*.PCB)

Older Protel DOS PCB version 3.12 files can be imported. All design objects are supported.

Protel ASCII and Protel Binary

These options allow you to import files from all previous versions of Advanced PCB. Always review the design rules after importing an older version file.

DXF Files (*.DXF)

Advanced PCB can import single layer and multi-layer DXF files.

Gerber Files

There are two ways of importing Gerber files, either as a single layer, or all layers simultaneously, in a batch.

➡ The aperture list that was used to create the Gerber file must be loaded to be able to import Gerber data. If no apertures are loaded, 1 mil apertures are assumed.

To import a single Gerber file, or a Gerber panel file, onto the current layer of the Advanced PCB workspace, select File-Import. In the Import File dialog box set the Type to Single Gerber file (*.g??). Locate and select the Gerber file and click OK. The import process translates all flashes in the Gerber file to pads and all strokes to tracks.

There are two exceptions to this rule:

1. If a horizontal or vertical stroke was generated using a square or rectangular aperture then the PCB file will contain an area fill of equivalent size.

2. Advanced PCB does not include rotated rectangular shapes in the Gerber generation process. If rotated rectangular shapes are present in the design they will be painted using a suitable round aperture when the Gerber file is generated. This can be verified when Gerber files generated from designs containing these objects are imported.

To batch import all Gerber files for a particular design set the Type to Gerber Batch files (*.g??). Locate and select one of the Gerber files and click OK. All Gerber files with this filename will be imported, each onto the layer identified by their file extension. For example, the .GTL file will be imported onto the top layer, the .GBL file will be imported onto the bottom layer, and so on. This provides a way of generating a PCB file from various layer files – from any Gerber source that can be read by Advanced PCB.

➡ Batch Gerber Import can produce very large PCB files as each layer will have its own set of pads or flashes, rather than a single multi-layer pad as contained in the original PCB file. Also the components will not be components, they will be made up of tracks and pads, on the appropriate layers. If necessary, these primitives can be selected and replaced by actual components.

The Gerber Import feature can be used to achieve some things which would not otherwise be possible in Advanced PCB. For example, to produce a Drill Drawing with complex information, generate the Drill Drawing Gerber file, then use Import Gerber to load the Drill Drawing onto a Mechanical layer. Add information as you please, then regenerate the "Mechanical" layer file – now a composite of Drill Drawing and Mechanical layer elements.

P-CAD PDIF (*.PDF)

Advanced PCB imports PCAD PDIF version 5 and 6 format files.

Boards should be 100.000 x 100.000 inches or smaller, to fit within the Advanced PCB workspace. Some limitations may be encountered with (through hole) pad stacks - as Advanced PCB limits pads to a single description placed on Multi layer. PCAD pads assigned as multi-layer require no special handling. If a pad stack exists, the top layer shape is assigned to the Advanced PCB Multi layer. The PDIF "Void" primitive is not supported by the current translator.

PADS ASCII (*.ASC)

Advanced PCB imports PADS-PCB and PADS2000 ASCII format files (.ASC).

PADS-PCB and PADS2000 provide you with many options when generating files in PADS .ASC format. Some of these options may yield a file that is incomplete or otherwise unreadable to Advanced PCB. If you experience difficulty in loading PADS files, try re-generating the file using the PADS default (include all) output options. Some problems are due to ambiguities in the .ASC format that cannot be resolved by Advanced PCB. For example, Advanced PCB cannot translate PAD-PCB files that contain numeric net labels of four or less characters, because there is no inherent identification of these strings as net labels. In this case, manual editing of the .ASC file allows Advanced PCB to resolve the nets.

If the file is large, it may take some time for the conversion to complete and the file to open. When you Save the file, it will be stored in the standard Advanced PCB binary format. Because PADS format files have a different layer identification system, you will need to re-assign layers during the initial load sequence. The Setup Layer Translation dialog box will open, allowing you to designate the layers. The complete file, including netlist and component information is preserved, allowing complete freedom to modify existing PADS files in Advanced PCB.

Advanced PCB will convert all "zero size" pads to 1 x 1 mil size, allowing you to (globally) edit or delete these "place holder" pads which are not needed in Protel's system.

You can also manually re-assign any power or ground nets (which are loaded as un-routed) to internal power and ground plane layers (Power Plane layers 1-4) after conversion.

Advanced PCB uses the Top layer, Bottom layer and Multi layer pad shapes (in pad stacks) to assign pad types. Larger internal pads (created for solder masks, etc.) will be lost in conversion. You will be able to use the normal Advanced PCB method of creating these attributes.

➡ Pads edge connector pads require special handling when the pad is assigned to an internal power/ground plane. A partially routed track segment will be connected to the pad. Place a free pad at these locations, then edit the pad to assign it to the desired plane.

Tango ASCII (*.PCB)

Tango PCB Series II files will be transparently imported by Advanced PCB.

This has been tested on Tango file formats, version 1.2A and 1.3A. All physical elements with the exception of multiple point polygons were successfully loaded and all nets and connections were successfully loaded. With polygons, Advanced PCB will only load Tango polygons which have four corners. If it finds a polygon with four corners then it will assume that is a rectangular area fill and load it as such. Advanced PCB also loads the older Tango-PCB files directly.

CCT Spectra and SB Route (*.RTE)

Select this option to import the route file generated by the Specctra Shape Based Autorouter.

Export Options

Advanced PCB supports exporting to a number of different file formats. These include Advanced PCB V2.8, AutoCAD DXF, the HyperLynx board simulation format, IPC-D-350 format, Protel Netlists, and route files suitable for the Specctra autorouter. To export to one of these formats select the File-Export menu item.

Protel ASCII (*.PCB)

This option allows you to export to the Advanced PCB version 2.8 ASCII format. Note that version 2.8 did not support design rules so these will be lost.

AutoCAD (*.DXF)

Exports all objects currently in the workspace into a DXF file.

HyperLynx (*.HYP)

HyperLynx BoardSim for Windows is a post-layout Signal-Integrity Simulator for Digital PCBs. BoardSim reads information directly from the Advanced PCB database and predicts transmission-line effects (like overshoot and ringing) based on the actual board layout. BoardSim includes; direct data import from Advanced PCB, automatic electromagnetic modeling of complex board traces, a PCB-layout viewer for PCB manipulation independent of Advanced PCB, a digital oscilloscope window for displaying signal waveforms, a graphical editor for changing the board stackup and comprehensive libraries of device models.

TO export to the BoardSim for Windows format simply set the Type to HyperLynx (*.hyp) in the Export File dialog box. Set the filename and click OK. The *.hyp file will be generated and stored in the specified directory. This file can now be opened in BoardSim for Windows, ready for signal-integrity simulation.

IPC-D-350 (*.IPC)

This option exports the Advanced PCB database into an IPC-D-350 format file. This is a mechanical format suitable for use by manufacturing, test and assembly equipment.

Netlist (*.NET)

This option will export the current internal netlist. This will be the netlist that was initially loaded into Advanced PCB, plus any modifications performed on it.

Shaped Based Design

Export the current file to the Specctra Shape Based router file format.

Auto Component Placement

Placement is a critical process in the design phase of a printed circuit board. The effort to route your design and its ultimate manufacturing cost are highly dependent on placement quality.

Placement involves specifying the exact location and orientation of each component on the PCB, given the board size and the area of the board on which placement is permissible. The most important objective in placement, after the fitting of all the components on the board without violating any design rules, is to make the job of routing easier, or in certain cases, possible.

In short, the main goals in placement of a PCB are :

- Fitting all the components on the board.
- Avoiding design rule violations.
- Placing components to allow routing completion.
- Meeting any requirements for board assembly and board testing. It is advisable to always consult with your board assembler during the design phase to ensure that their requirements are met.

Advanced PCB includes a "Global" automatic component placement server, which has been especially developed to enhance this difficult and critical phase of the design process. This chapter includes information on setting up and running the placement server, as well as a discussion of the theory of auto placement algorithms.

➡ To perform automatic component placement you must install the Placer Server. Refer to the chapter *A Quick Tour of EDA/Client* for tips on installing a server.

➡ Before using auto placement set the *current origin* back to the *absolute origin* by selecting the Edit-Origin-Reset menu item. This can be important because the auto placement routines use the absolute origin as a reference point and may place components off grid relative to the current (relative) origin you created.

Board Area For Placement

Define the board area available for component placement prior to running the Global Placer Server. This is done by placing tracks on the keep-out layer to define a boundary within which all components are to be placed.

To keep certain regions of the board free of components, create keep-out zones. Place tracks, fills, arcs and polygons on the Keep Out layer to create these keep-out zones. Refer to the chapter *Defining the Board* for further information.

Setting Up the Global Placer

The Global Placer is easy to use, despite the underlying complexity. All you need to know are some very basic facts about the design. No special configuration is required, as the system has been finely tuned over a wide spectrum of boards for optimal performance.

To set up the Global Placer, select the Tools-Auto Place menu item to pop up the Autoplace Preferences dialog box.

Options

Group Components

If this option is enabled the Global Placer will go through all the components on the board and group together those which are tightly connected. The main criteria for grouping is the number of connections between components. The weight given to this criteria is influenced by the number of pins these components have.

The system then performs a relative placement on each group. These groups are treated as "super components" and their relative placement is kept untouched during the main placement cycle.

Although this option is generally useful, it can be counter productive if there is not going to be enough space on the board. This is because the relative placement within each group is not changed during the main placement cycle and space might be wasted in order to accommodate the group.

Rotate Components

If component rotation is allowed, components will be rotated in order to find an optimal orientation. Four orientations are considered, 90, 180, 270 and 360 degrees.

This option must be used with caution, as footprint rotation can have a direct impact on the manufacturability of the design. For example, some pick-and-place equipment cannot handle rotated components. Enabling Rotate Components also makes the placement job more difficult and time consuming – particularly if the design is very dense. Given that, it can also produce a superior result, so some degree of trial-and-error is normally required.

Advanced PCB also allows you to freely rotate components after placement to 0.001 degree accuracy, using the Edit-Move, Edit-Paste Array or the Rotation field in the Change Component dialog box.

Automatic PCB Update

The Global Placer will automatically pass the current component positions to the board in Advanced PCB each time the optimization status is updated (approximately every 10 seconds). You can also manually update at any time while the Placer is running by selecting File-Update PCB.

Placement Grid

This is the grid each component reference point will be placed on. It is typically set to a fraction of the common component pin pitch and/or a multiple of the intended routing grid. Advanced PCB includes a tool to move all components to a new grid if the component placement grid needs to be altered at a later stage (Tools-Align Components-Move To Grid).

Power Nets

The Power Nets option performs two functions;

1. Nets that are specified in the Power Nets fields are no longer considered by the placement algorithm, which can greatly speed the placement process.
2. To associate a bypass capacitor with each large component, specify the names of the nets which the capacitors are across. For example, if your design uses the nets VCC and GND as the power nets, enter VCC in the first field and GND in the second. The Global Placer will attempt to associate a two pin component that is connected across the specified power nets (VCC and GND) with each large component (14 pins or more).

➡ More than one power net can be specified in each text box. Separate net names by a single blank space (total of 28 characters per line maximum).

Clearances

Set the desired clearances for small components (less than 14 pins) and large components (14 pins or more). The component clearance boundaries are calculated from the smallest rectangle that can encompass all the component primitives.

➡ These clearance settings are the *clearance around* each component, which are added to determine the clearance between any two components.

Running the Global Placer

Once you have configured the Autoplace Preferences dialog box and are ready to start the placement process, press the OK button.

The Global Placer Window

The Global Placer Server displays the placement progress in its own window. This includes the components and the keep-out areas.

The main menu for the Global Placer is very short. Options include: File, View, Window and Help. As the Global Placer has its own data structure the PCB database is not changed during the placement process. The File-Update PCB menu item can be used to pass the current placement back to Advanced PCB. This allow you to periodically switch back to Advanced PCB to check the placement quality and use the Density Map tool to examine the "routability" of the design.

The placement window has its own Status Bar, within which the following information is available :

Elapsed time

Time since start of placement.

Optimization

There are a total of 70 cycles in any placement task. The first 40-50 cycles are very fast as most moves are accepted. However, as the "temperature" decreases, more and more moves are made in order to satisfy the requirements of a cycle. This means the cycles get slower towards the completion. Optimization refers to the percentage of completion toward an "ideal" set of costs. Refer to the *Theory* topic at the end of this chapter for a discussion of optimization.

Number of Moves

The Number of Moves, displayed in the status bar, is the total number of times that the system has moved a component to a new position in order to improve the routability of the board.

➡ During the placement process small purple squares will appear on the board. The size of these squares reflect the connection density in that region.

Placement Results

The placement process will make smaller and smaller moves as it progresses towards its optimal solution. It is not necessary to run it to completion. Select File-Close to terminate the placement process. You will be asked if you wish to update the PCB before closing the placement window.

Tips for Better Results

The Global Placer is highly robust and generally does not require any special user direction. However, should any difficulty arise, the following points could prove useful.

Pre-Placing Components

You can pre-place any component before running the Global Placer. To prevent these components from being moved enable the Locked attribute in the Change Component dialog box.

Apart from those components which have to be placed in certain locations on the board such as edge connectors, heat sinks, or a group of analog components, it can be useful to pre-place components which need no restriction on their placement. For example, it might be desirable to pre-place the memory chips. This could facilitate the placement of the other components.

Use of Keep-Out Zones

To keep certain regions of the board free of components, create keep-out zones. These could be placed next to connectors, or in regions which must be kept clear for mechanical reasons. Place tracks, fills, arcs and polygons on the Keep Out layer to create these keep-out zones.

Auto Place and Larger Nets

Large nets slow down the Global Placer. The reason for this is the computation involved with rearranging a net is exponentially proportional to the net size. An interesting observation is that large nets, such as power and ground, can play an insignificant role of in the overall placement process. Therefore it can be advantageous to instruct the Global Placer to ignore these large nets. To do this assign the large nets in the Power Nets region of the Autoplace Preferences dialog box.

➡ Remember, automatic placement is a productivity tool – not a replacement for the judgment and experience of the designer. A little guidance from the designer – for example, locking down connectors and "seed" components, realistic placement grids and clearances, can all go a long way toward ensuring that the Global Placer will both speed and ease the design process.

Interactive Placement Tools

The outcome of the Global Placer is a board in which the relative positions of the components are optimal. Due to its global nature, the Global Placer often produces boards which are not entirely "polished". For instance, there could still be some overlaps after the placement is completed, or some components might not be aligned properly. The interactive placement tools are specifically designed to facilitate the process of tidying the placement. Refer to the *Interactive Placement* topic in the *Component Placement* chapter for information on using the interactive placement tools.

Auto Placement Theory

This section gives an overview of the task of developing an intelligent automatic placement tool, including some of the theory and the various optimization techniques.

Theory of Optimization

Optimization is a formal technique used by mathematicians to find the extremes of a function. In mathematical terms this problem can be stated as follows:

Given the function f(x) with x belonging to the set s and s being the set of all the possible x's find a w from the set s for which f(w) is a maximum (or a minimum).

Solving this problem has turned out to be one of the most difficult areas in mathematics. The complexity of this problem is governed by the type of the function, and the size and dimension of s.

There are hosts of different techniques developed by mathematicians and other scientists to solve this problem. However most of them only work on a small subset of the real-world problems.

If f(x) is an analytical function, using differential calculus maxima and minima of f(x) can be found. This method is the fastest in terms of the computational time required. Moreover it always guarantees a solution can be found. For most real-world problems f(x) is not an analytical function and consequently this method can not be used.

One optimization technique which is almost diametrically opposite to the analytical method, uses an exhaustive search to find the solution. What exhaustive search means is to try all the possible x's and pick the one which gives the maximum for the function (or the minimum). This method also guarantees to produce the best solution. By contrast, it can be applied to any problem regardless of the type of the function. The heavy price is that some geological time (minutes or hours) is needing to carry out the computations involved for most of the real-world problems.

Although the real-world problems seem to be almost impractical for the optimization techniques to tackle, they have however one major handy attribute which allows us to successfully apply an optimization technique to a real-world problem.

That attribute is that in a real-world problem we are often not after the maximum or minimum, rather an approximation to them. In other words, we are not after the x which gives the maximum (or the minimum), rather an x which produces a high enough (or low enough in case of minimization) value for f which can satisfy the problem at hand.

This aspect of real-world problems has given rise to a spectrum of different techniques in between the two extremes, i.e. the analytical and the exhaustive search methods. Most techniques in that range make a compromise between the computational time and the quality of the approximation. They generally attempt to find a good solution by searching a very small subset of the set of all possible points in the solution space. The criterion by which a small subset is selected for search is the most intriguing part of the optimization problem, and often the cause of diversity in the methods of optimization.

Global and Local Optimization

Approximation of a solution is a very loose goal when we do not have any idea about the solution. How can we approximate a number when we do not know anything about the magnitude of that number? All we know is, "the higher the better"(or "the lower the better" in the case of minimization). The answer to this is that most of the optimization techniques only find the best out of the ones which have been tried, and that, they only try the ones which are expected to make an improvement . A method like this can not claim to be able to find one of the best solutions. This problem is commonly known as the problem of Local Maxima (or Local Minima in the case of minimization).

An analogy to this is the case of a blind man wanting to get to the highest point in a valley. Starting from a random point on the valley, he only moves in a direction where he can ascend. It is obvious that the chances for this poor man to get to the highest point in that valley is very small. In fact, a small hill which happens to be near his starting point can fool him and drag him to its top, making him believe he can not get any higher, as there is no point near by the top of that hill which is of a higher altitude. The techniques exhibiting this or similar kinds of problems are commonly referred to as local optimization, or greedy methods. By contrast the methods which find one of the best solutions are commonly referred to as global optimization methods.

Developing a Function for the Problem

In order to apply optimization to the problem of automatic placement we first need to define a function which can measure the "quality" of a particular placement. In this case our variable x is a proposed placement and s is the set of all the possible placements. We then use a suitable optimization method to find the placement which gives us a high value of "quality".

This function, depending on the particular method of optimization employed, is referred to as the cost function, objective function, evaluation function, fitness evaluator, etc. We have used the term Routing-Difficulty Function.

To determine the "quality", this function measures the following :

- Connection length.
- Connection density on the board.
- Component's alignment.
- Design rules violations.

Our goal is to minimize this function.

It is very important to realize that the overall success of any optimization technique is highly dependent on how good this function models the actual problem.

Optimization Techniques

Successful routing is highly dependent on a placement that is optimized for routing the entire layout. the Global Placer attempts to find a global solution rather than a local one. This was likened to a blind man attempting to find the highest point in a valley. He may be able to judge whether he is going up or down, and locate one or more local peaks, but it is very difficult for the blind man to know if any peak is the highest peak in the valley. Auto placement poses a similar problem that "blind" local placers cannot defeat. To find the highest point, we have to employ a global optimization technique which will ultimately minimize the Routing-Difficulty function.

There are only a few global optimization techniques and almost all of them have been developed very recently. Amongst them are the simulated annealing method, simulated evolution method and genetic algorithms. These methods all, have received inspiration from nature as their names suggest. In the annealing process of a metal, atoms array themselves in a way that crystals of low energy are formed. In natural evolution, only those species which are of better fitness survive and in genetics through cross-over and mutation, on average, better chromosome are produced from generation to generation.

The optimization technique which was chosen is the Simulated Annealing method.

Simulated Annealing

the Global Placer uses an AI-based methodology called simulated annealing. This methodology mathematically imitates some metallic characteristics.

In a metal raised to a temperature above its melting points, the atoms are in violent random motion. As with all physical systems, the atoms tend toward minimum energy state (a single crystal in this case), but at high temperatures the vigor of atomic motion prevents this. As the metal gradually cooled, lower and lower energy states are assumed until finally the lowest of all possible states, global minimum.

A simple way of describing the application of this to the PCB placement problem is to consider each component as a metallic atom during the annealing process. Now all we need is some artificial temperature, base on which, we can control the random movements of components. Starting at a high temperature, allowing components to move freely, we gradually drop the temperature and thereby applying some loose restriction on the moves. As the temperature drops lower we apply more restriction until we get close to the freezing point around which only moves which result in improvements are allowed.

Autorouting

Routing is the process of creating the "physical" connectivity in your design. Advanced PCB supports two methods of creating this physical connectivity.

The first approach is manually, where *you* place tracks and vias to create the connectivity. The second approach is where the tracks and vias are placed automatically, that is, the connections are *autorouted.* As designs become more complex and the components and routing objects (tracks and vias) continue to shrink, autorouting is becoming the preferred approach.

Traditionally autorouting has been the domain of high-end workstations, running very expensive EDA tools. With the maturing of desktop EDA tools, combined with the rapid improvement in the performance of PCs, viable autorouting technologies are now available on the designer's desktop at realistic prices.

Advanced PCB supports autorouting at two levels;

1. Advanced PCB includes an autorouting server, Route2. This autorouter includes; a memory routing pass, SMD specific routing passes, a rip-up-and-retry maze router, a via reducing smoothing router and a miter pass. Route2 has been designed to autoroute through hole and SMD designs of "medium" density. This includes digital PCBs with through holes and two layers (up to 0.8 sq. in per 14 pin equivalent) or four layers (up to 0.6 sq. in per 14 pin equivalent), and SMD boards with components predominantly on one side.

2. Interface to high-end autorouters - autorouting technologies are developing and there are now very powerful high performance autorouting "engines" which can be interfaced to from Advanced PCB. These include Protel's Advanced Route[3], a gridless shape based router that incorporates the latest neural learning algorithms. These neural learning algorithms include the best experience of actual board designers, in the form of neural knowledge. This knowledge is then applied as the router "learns" your design and adjusts its internal costing accordingly. As well as being a powerful autorouter now, this adaptive modeling approach offers great potential as the understanding of neural networks grows and computational power increases. Advanced PCB can also interface to the Specctra shape based router.

The success of autorouting is dependent on a number of different factors:

- The quality of the component placement.
- The density of the PCB.
- The design rules employed and number of copper layers available.
- The manufacturing technology available.
- The aesthetic requirements of the individual designer.

This chapter begins with an introduction to autorouting technology, which will help you to get the most from the autorouting tools available in Advanced PCB. It then details how to setup and run the autorouting tools available in Advanced PCB.

An Introduction to Autorouting

Autorouters are generally judged first on the completion percentages achieved for a given layout. Designers have to consider a number of other measurements of router success. If the board is a standard through-plated digital board with two signal layers, then completion percentage may be the only requirement. If the board is mainly analog, RF, or power supply, then completion percentage may not be as important as "star pointing," connection lengths, shielding and so on. Criteria such as cost and manufacturability are also important for production boards.

For mixed analog and digital boards, the designer may need to consider manual routing of the analog section and then autorouting the remaining digital sections. Digital boards with complex bus structures may benefit from manual routing of the bus on one or two layers, and then autorouting the remaining random logic.

The class of boards that can be (fully) autorouted successfully with Advanced PCB include digital PCBs with through holes and two layers (up to 0.8 sq. in per 14 pin equivalent) or four layers (up to 0.6 sq. in per 14 pin equivalent), and SMD boards with components predominantly on one side. Adding extra layers will allow greater component densities. Hand routing busses and other difficult connections will allow successful routing on much higher density boards.

Autorouting Strategies

In order to get the best results from autorouting, it is useful if the designer has a basic understanding of autorouters and some of the strategies employed in autorouting.

There are two main classes of autorouter:

1. Routers that route one connection at a time until they either finish or run out of options.
2. Routers that, after finishing one pass of the board, modify the design in some way to get the failed connections in, repeating this process as many times as possible until the board is routed. These "iterative routers" (sometimes referred to as "100% routers") fall into two main approaches, "rip-up" where blocked connections are temporarily un-routed then re-routed and "shove aside" routers which add the ability to move tracks aside to create additional routing channels.

Within both classes of router, there are a number of different techniques used to actually find a path between two points on a partially routed board:

Memory (or Pattern) Routers

These routers, sometimes called heuristic routers, look for a known commonly occurring pattern of pins, and try to insert a standard pattern of tracks and vias to complete the connection. For example, memory bussing is a common pattern, usually solved with the characteristic wave pattern. Simple L patterns, formed from a horizontal and vertical track segment (one on each layer) connected by a via, is another common pattern.

Pattern routers are usually very fast, produce very high quality routes but will only achieve a low completion rate on all but the most simple boards. For these reasons, pattern routers are usually run first, to pick up the easy connections, and provide a high quality solution.

Line Probe Routers

These routers "probe" with a test track from both ends of a connection until an obstacle is encountered. The probes turn to one side until the obstacle is avoided and then resume toward the target. When they meet, the connection is complete. There is usually a limit to how far sideways the probe can go.

Line probe routers are reasonably fast and provide high quality routes with low via counts, but suffer from a major problem. They are prone to run into blind alleys and often "miss" very simple solutions. As a result, they tend to slow down and fail as the board gets fuller. This is because they are not evaluating all possibilities, but simply following the most direct path. However, when combined with a heuristically guided route shape, such as a Z route (two vias, two verticals an one horizontal track etc.) they can provide very high quality solutions to fairly difficult routing problems.

Wave Expansion Routers

Commonly referred to as "flood," "maze" or "Lee" routers. These routers use an exhaustive search to find a solution to a connection if one exists. The flooding process is a metaphor for pouring water over the board from one end of the proposed connection, where all free spaces are considered as channels for the water, and all obstacles are considered as islands. The flood will spread out in all directions until, if physically possible, the flood will reach the other end of the connection. This approach can be enhanced in a number of ways.

The quality of the route can be enhanced by setting a "cost" for changes in direction, swapping layers and moving against the layer bias direction. The solution will then be a balance of these parameters, controlled by the weighting of the various costs. Expanding points on the wave front that are closer to the target and in the same direction as the target, before others, will speed up the search.

There are two basic kinds of wave router, gridded and gridless. In the gridded type, each point of space on the board is represented as a point in a two dimensional array, either occupied or empty. The problem here is that the memory requirements go up as the square of the grid resolution. In other words, doubling the number of grid points quadruples the memory requirements.

In gridless wave routers, the expansion is performed by expanding the current point (or front) of the wave in a rectangular shape, until obstacles are hit. When an obstacle is encountered, the intersection of the obstacle and the rectangle is calculated and another rectangle is expanded from the free section of the edge of rectangle. This approach is very processor intensive, but completely grid independent due to the use of polygon (rather than point) calculations.

Although tremendously powerful and flexible, wave type routers are relatively slow and "memory hungry".

Preparing to Autoroute

Prior to running the autorouter there are a number of issues to consider. These include;

Choosing the Placement Grid

The quality of routing and the degree of completion is heavily influenced by the component placement. As well as how appropriately the components are placed relative to one another, it is extremely important that they are placed on an suitable grid. Using an appropriate placement grid will affect the number of routing channels available to the autorouter. Also, off grid pads slow the router significantly.

The choice of placement grid depends on the routing technology that you intend to employ, that is, the track, via and clearance parameters. The choice of routing technology depends on the common component pin pitch, the density of the design and the number of routing layers available. Ideally, you will attempt to route the board with the fewest number of routing layers and with the largest tracks and clearances that can be achieved.

All this can be difficult to determine, particularly for the new designer. Refer to the *Routing Models* topic later in this chapter for examples of typical grid and primitive settings. Once a placement grid has been selected, do the following;

- Set the Snap grid to the value you have chosen for the placement grid.
- Select the Edit-Select-Off Grid Pads menu item to identify off grid pads. These off grid pads may belong to components which are position critical, such as connectors. These components should be locked in place. If the position of any off grid components are not critical they should be moved onto the placement grid.
- Select Tools-Align Components-Move To Grid to move the reference point of all un-locked off grid components onto the selected grid.

Setting up the Design Rules

The design rules that are observed by Route2 are described below. Refer to the *Design Rules* chapter for a detailed description of each rule.

Width Constraint

This design rule controls the width of any tracks (straight or curved) placed by the autorouter. Route2 will route with the Maximum width specified. This value should be set in conjunction with the grid size and clearances to enhance routing success.

Routing Priority

Add routing priority rules to define an order of routing. The routing priorities range from 0 (lowest) to 100 (highest). The routing priorities are relative values which are used to set the order that the nets will be autorouted.

Routing Via Style

This design rule specifies the via parameters. Route2 supports through-hole and Blind/Buried Layer Pair vias. Blind and buried vias improve the density of the board when four or more signal layers are used. Refer to the *Vias* topic in the *Design Objects* chapter for information about blind and buried vias.

Routing Layers

The Route2 autorouter in Advanced PCB can use all 16 signal layers plus power and ground planes. You should include at least one Routing Layers design rule, specifying which layers are to be used and the primary routing direction on each of these layers. If the board is to be multi-layer and you intend to use blind and buried vias, you must use the appropriate layers. For information on selecting layers for blind and buried vias refer to the *Vias* topic in the *Design Objects* chapter.

Route2 can use the following routing direction options that are available for each layer:

Not Used

The layer is disabled and will not be used for routing.

Horizontal

Tracks will be primarily placed in a horizontal orientation. Short vertical tracks will be placed on this layer to avoid planting a via.

Vertical

Tracks will be primarily placed in a vertical orientation. Short horizontal tracks will be placed on this layer to avoid planting a via.

Any

Router will place tracks either horizontally or vertically. If only one active routing layer is available, the router assumes a single sided board with no vias. Generally it is not advisable to use this option for multi-layer boards as the layers will quickly become choked.

Layer biasing

In Advanced PCB, you can assign the routing direction for each layer: Horizontal, Vertical or Any (for single layer routing). It is standard practice to alternate the primary routing direction used on each pair of layers. This is known as layer biasing.

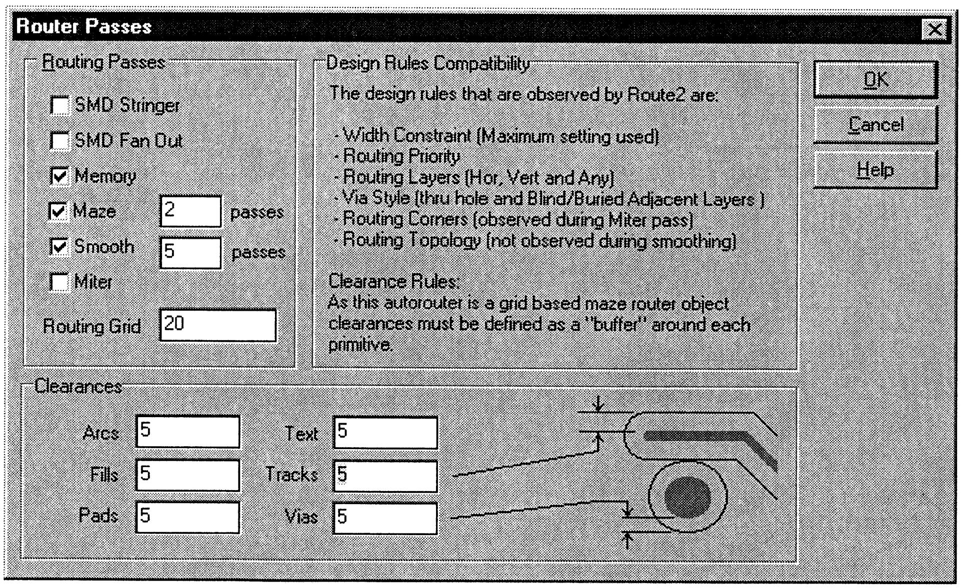
Multi-layer board manufacturing technology enables you to route connections across two or more layers, separated by a dielectric. Placing parallel tracks on adjacent layers will result in an increased capacitance between the layers, and will block track placement in the opposite direction across the board. For this reason, layer biasing is commonly used in designing boards with two or more layers. This does not mean that routing is restricted entirely to this direction (which would generate too many vias), but that the majority of tracks on a layer will travel in the same direction. Layer biasing also improves autorouter performance, because connections are less likely to be blocked by this arrangement of parallel tracks on each layer.

Checking the Routing Density

To help in the process of determining how the board should be routed; that is, the track/grid sizes, the via size, the number of layers and so on, Advanced PCB includes a "Density Map" feature. Select the Tools-Density Map menu item. After a few moments the board will be "painted" with a colored map. The green color represents "cool", or less dense regions, the red color represents "hot", or most dense regions. If there are large areas of red you may wish to analyze the current component placement and try to remove these "hot" zones. If this is not possible your design may require more routing layers.

Setting Up the Autorouter

Select the Tools-Setup Autorouter menu item to pop up the Router Passes dialog box, where you set up the routing passes, routing grid and the routing clearances.



Routing Passes

SMD Stringer

When your design includes SMD components which connect to a net on an internal plane, the SMD Stringer pass routes a short track segment with a through hole pad at the end, connecting to the plane. You assign a net to an internal plane in the Internal Planes dialog box (select the Design-Internal Planes menu item).

Add a power Plane Connect Style design rule to control how the net connects to the plane (thermal or direct). Refer to the *Design Rules* chapter for more information on the Plane Connect Style design rule.

SMD Fan Out

This pass routes a short track segment with a via at the end, out from each surface mount pad. If a particular pad can not be fanned out it will be fanned in. To force a particular component to fan in, place a temporary boundary around the component on the Keep Out layer and run the Fan Out pass on that component.

Memory

A fast heuristic pattern router to put in the classic wave structure for memory busses. As a general rule this pass can always be included.

Maze

The maze router is a gridded wave expansion router with rip-up and re-try capabilities. The maze router can take some time to complete on a complex board. Maze routing places a high priority on layer biasing. It will rip-up and reroute other connections which block its path. When the Maze router rips-up an obstacle, it doesn't necessarily rip up the entire connection. In most cases it will only rip-up and reroute a single track segment. Connections which are ripped up are rerouted immediately after the current connection is routed. Sometimes this reroute will result in some fairly obvious backtracking, so it is always best to run at least one smoothing pass after maze routing.

The Maze and Smoothing passes can be run multiple times in an attempt to improve the finished result. Maze passes will stop when all connections are complete, regardless of the number of passes specified.

Generally it is better to route the board as close to 100% as possible with Maze passes. If you find that the board will not route to 100% with Maze passes it may be that it has become choked with vias. In this situation, run a smoothing pass and then run more maze passes.

Smoothing

This pass checks the board for vias that can be removed by swapping tracks from one side of the board to the other and resolves loops into direct connections by using copper sharing. Smoothing can result in very significant improvements to the router result.

The smoother works by ripping up the entire net and rerouting it with different internal costs. Smoothing places high priority on reducing the via count and copper sharing. Smoothing is applied progressively with each pass. If multiple smoothing passes are enabled each pass will run with increasing thoroughness; the cost for undesirable conditions (e.g. extra via, etc.) increasing, and the cost for desirable conditions (copper sharing, wrong way tracks, etc.) decreasing with each subsequent pass. Use the router .LOG file to see the progress made during each pass.

➡ As the costing of each pass is based on the total number of smoothing passes specified, it is preferable to allow the smoother to complete all specified passes. The smoother will achieve its best results in the later of the specified passes.

If you do not want the smoother to be able to process a particular connection or net, lock the tracks and vias in place. Refer to the *Global Editing* chapter for an example of how to do this.

Miter

A miter is where a corner is changed from a 90 degree angle to either a 45 degree diagonal track, or an arc. The Miter pass uses the settings in the Routing Corners Rule.

Routing Grid

The routing grid sets the location of tracks, vias and arcs placed during autorouting. Decreasing the grid size will improve the completion rate, but slow the router and increase the memory usage during the Maze and Smooth router passes. The default value is 20 mils. The range is 5.000 – 100.000 mils. Fractional routing grids such as 8.333, 12.5 and 16.667 can be used.

Select a routing grid which suits the chosen routing model. There may be no advantage selecting a smaller routing grid if routing is constrained by primitive sizes and clearances. Remember - halving the routing grid increases the memory required and the number of routing cells to process by a factor of 4.

➡ Ideally the routing grid should fall exactly on the most common component pin-pitch. This limits off-grid pins that make autorouting more difficult. For example, components with a 50 mil pin pitch can easily be routed using a 50, 25, 10 or 5 mil routing grid.

Clearances

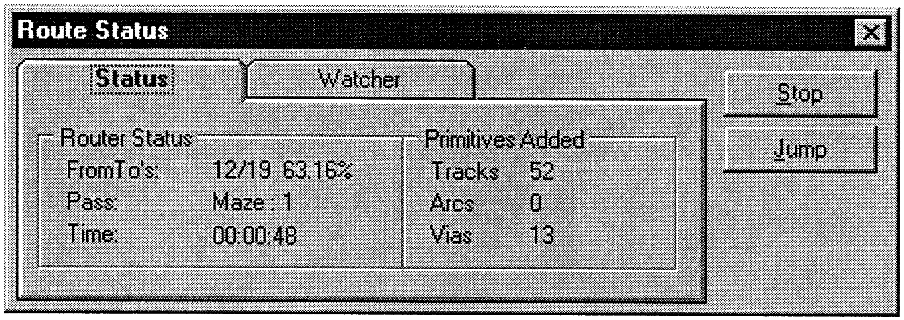
As Route2 is a grid-based maze router, the clearances must be specified as a "buffer" around each primitive. Remember, you add the two clearances to find the clearance *between* two primitives.

Routing Your Board

The Tools-Autoroute sub-menu is used to start the autorouter. This menu has a number of routing options. These include:

All

Attempts to route all netlist connections in the workspace using the current router setup specified in the Router Setup dialog box.



Once autorouting commences the Route Status dialog box will pop up. This allows you to keep track of; which router pass is currently being used, the number of connections routed out of the total, the number of primitives added to create the routes and the total time taken.

Net

Attempts to route all the pin-to-pin connections of the selected net using the maze router setup.

Connection

Attempts to route a single pin-to-pin connection. When prompted, select a connection. The Maze router will be used to route the connection.

Component

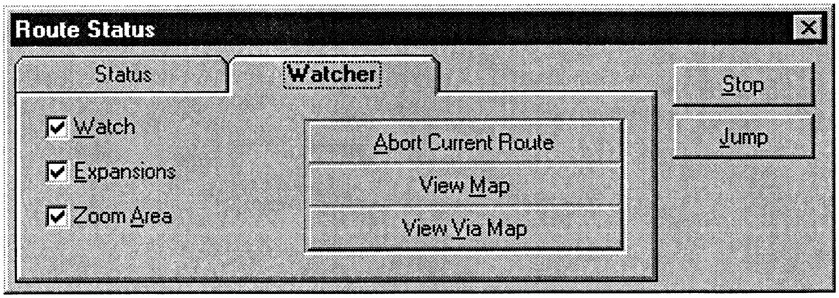
Attempts to route all connections to the selected component using the Maze router. After choosing this menu item you will be prompted to select a component. This option routes the connections to this component only, not the entire nets.

➡ You can un-route a component (Tools-Un-Route-Component), move the component to a new location (including changes in orientation) and then re-route all of the connections by selecting the Tools-Autoroute-Component menu item.

Selected Components

Attempts to route all connections to the selected components using the Maze router. Select the components prior to choosing this menu item. This option routes the connections to the selected components only, not the entire nets.

Watching the Autorouter



Advanced PCB includes a "Watcher," which allows you to monitor the progress of the routing passes. The Watcher is very useful if you are having trouble understanding why your board will not route. In order to route a connection, the maze router and the smoother build a grid or "cell" map of your board and then expand from one end of the connection to try to reach the other end. The Watcher lets you see the board being routed from the router's point of view. The Watcher options are selected in the Watcher Tab of the Router Status dialog box.

When the Watcher is on small arrows are displayed on the screen as the router tries to expand from one node, or pin, to another. These arrows indicate the direction of expansion. When the target node is hit, the router traces the successful route back to the source and then places actual tracks and vias on the PCB. Turn the Expansion option off to hide the expansion arrows. Enable the Zoom Area option to automatically zoom to the connection currently being routed.

Watcher Symbols

- Arrows show the direction of expansion.
- Plusses indicate moveable (i.e., rip-up-able) obstacles.
- Crosses indicate 45 degree obstacles.
- Green rectangles are places where the router is not allowed to place a via.
- Crosses during expansion are potential via sites.
- Green arrows are potential rip-up points.
- White crosses indicate areas where the expansion is sharing copper with the same net.

Un-Routing

The Tools-Un-Route menu options remove routed primitives (tracks and vias) and restore the connection lines. The un-route options are similar to the Tools-Autoroute options and include: All, Net, Connection, Component, Selected Components and Track (single track segment).

Routing Models

To get the most from the autorouter, carefully select a combination of clearances, track size, via size and routing grid that match the board technology. Here are some examples.

Single Density Through-hole (100 mils pad centers):

Track: 12 mils

Via: 50 mils (or 62 mils if preferred)

Clearances: 13 mils between primitives

Routing Grid: 25 mils

Placement Grid: 25, 50 or 100 mils

To use this model, the majority of your pads should be 62 mils or less in diameter. This will enable a single track to pass between (100 mil spaced) pads.

Double Density 1 (through-hole)

Track: 10 mils

Via: 50 mils

Clearances 10 mils between primitives

Routing Grid: 20 mils

Placement Grid: 10, 20 or 100 mils

To use this model, pads should be 50 mils or less in diameter. This will enable two tracks to pass between a pair of pads on 100 mils centers.

Double Density 2 (through-hole)

Track: 8 mils

Via: 50 mils

Clearances 8 mils between primitives

Routing Grid: 20 mils

Placement Grid: 10, 20 or 100 mils

To use this model, pads should be 56 mils or less in diameter. This will enable two tracks to pass between a pair of pads on 100 mils centers. This double density model allows for larger pads.

Triple Density 1 (through-hole)

Track: 8 mils

Via: 42 mils

Clearances: 8 mils between primitives

Routing Grid: 16.667 mils

Placement Grid: 33.333, 50 or 100 mils

To use this model, the majority of your pads should be 42 mils or less in diameter. This will enable three tracks to pass between a pair of pads on 100 mils centers.

Triple Density 2 (through-hole)

Track: 6 mils

Via: 30 mils (or larger)

Clearances: 6 mils between primitives

Routing Grid: 12.5 mils

Placement Grid: 25, 50 or 100 mils

To use this model, the majority of your pads should be 56 mils or less in diameter. This will enable three tracks, maintaining clearances, to pass between a pair of pads on 100 mils centers. This triple density model allows for larger pads.

Single Density SMD (50 mils pad centers)

Track: 8 mils

Via: 40 mils

Clearances: 8 mils net between primitives

Routing Grid: 25 mils

Placement Grid: 25, 50 or 100 mils

To use this model, the majority of your pads should be 26 mils or less wide (in one axis), to leave a gap of 24 mils between pads. This will enable one track, maintaining clearances, to pass between a pair of pads 50 mils apart. The via size can be varied as required. This a typical SMD board.

Double Density SMD (50 mils pad centers)

Track: 5 mils

Via: 30 mils

Clearances: 5 mils between primitives

Routing Grid: 10 mils

Placement Grid: 10, 50 or 100 mils

To use this model, the majority of your pads should be 24 mils or less wide (in one axis), to leave a gap of 26 mils between pads. This will enable two 5 mils tracks, maintaining clearances, to pass between a pair of pads on 50 mils centers. The via size can be varied as required. Check with you board manufacturer before working to these specifications, as this is a fairly demanding level of board technology.

Getting the Best Results from the Autorouter

As the designer you will nearly always be able to "improve" the results of autorouting. The trade-offs required to meet the primary objectives of a multi-purpose router will not always generate an "optimum" result for a given design. Not all designs are appropriate for autorouting. Autorouting should be viewed as another automation tool which, if used properly, improve your overall productivity.

While you can view the board as a whole, the router is only able to "see" one connection at a time. Understanding, and working within the limitations of the autorouter will help you get the best result in terms of overall productivity.

User-defined variables have a significant impact on the completion rate, quality and speed of the route. The most important factor is the routing grid selected. If you halve the grid, you quadruple the number of potential solutions for each route – however, routing time also quadruples. Off-grid pads, component layout, pre-routed connections, and clearance settings, will also profoundly impact the completion rate and route quality obtained. To improve Router performance:

1. Moving components (and therefore most of the pads) to the routing grid will significantly improve the performance of the router. Making sure that all pre-routed tracks are "on grid" will have even more effect. Because the router is grid based, tracks which are even slightly off grid can cause unexpected blockages.

   Select the Tools-Align Components-Move to Grid menu item to move the reference point of all components onto the routing grid. Make sure that you have nominated a grid that "fits" the maximum number of components used, including SMDs. The minimum routing grid is 5 mils (or .125 mm). Set up the router to route on a grid which is equal to, a fraction of, or a multiple of the placement grid.

2. Minimize the connection distances of discrete components, such as resistors. The connection lines and the Density display are good aids in positioning components to minimize congested routing areas. Use the PCB:ReportNetlistLength process to indicate changes in the overall connection length as you trial various placements. A lower total connection distance should improve routing.
3. Choose a routing model that 'fits' the density of your design. The ability to be able to choose the most appropriate model will improve with experience.
4. You may wish to run smoothing passes separately from the routing passes. This will allow you to conveniently revert to the "routed" results if you are not happy with the clean up.

Memory Requirements

Memory available for autorouting should be physical – not virtual – because virtual memory (accessing the hard disk) is impracticably slow due to the extreme number of computations required to complete each connection. If you find that routing is proceeding very slowly, you may need additional RAM to successfully autoroute.

Use this formula to estimate the memory requirements for maze routes:

(x size / grid size) x (y size / grid size) x (route layers) = memory used

Using this formula a 6" x 4" board, routed on a 25 mils grid, top & bottom only would yield:

(6000/25) x (4000/25) x (2) = 77 KB

For a 12" x 14" board, 20 mils grid, top bottom and 2 mid layers:

(12000/20) x (14000/20) x (4) = 1.7 MB

If you mix track widths for different nets it is usually much faster to route all the nets that have a particular size assignment together.

➡ The maze router will run much faster if the following two conditions are met: track size plus 2 times the track clearance setting is less than or equal to the routing grid; via size plus 2 times the via clearance setting is less than or equal to 3 times the routing grid.

Glossary

absolute origin The absolute workspace origin at the lower-left corner of the workspace. See also; current origin.

active document In the Windows environment, the Active Document is distinguished by its title bar being colored in the active color (default is blue) with inactive documents having their title bars colored in the inactive color (default is gray).

active Document Editor The active Document Editor is distinguished by its EDA Editor Tab being on the top and filled in gray. Documents of the active document editor type can now be edited.

active layer Any board layer which has been enabled in the Setup Preferences dialog box.

Advanced PCB Server which allows you to design printed circuit boards and generate output files from which phototools can be created.

Advanced Schematic Server which allows you to create and edit schematic sheets and the components used in those schematics.

annotation Component reference designators (or labels) that appear on the schematic sheet, in netlists or on a printed circuit board.

ANSI Refers to an international standard for technical drafting. See also *ISO*.

any angle Non-orthogonal tracks that can be placed at angles other than 45 or 90 degrees.

aperture file An ASCII text file which includes a description of each of the apertures used to generate a Gerber photoplot file. These descriptions are stored in aperture files, also called aperture tables.

application A program (Windows terminology). An application has the .EXE file extension.

arc Circular or semi-circular design element. Advanced PCB supports arcs of .001 degree resolution.

array Multiple instances of a single item, placed using the Paste Special option.

area fill See fill.

ASCII American Standard Code for Information Interchange. Standard seven bit code for representing alphanumeric data and computer instructions.

attributes The characteristics which define an object. Attributes can be edited, or changed. For example track attributes include width and layer.

automatic startup Start up state of a server where it is automatically started when EDA/Client is started.

auto place Option for automatically positioning components within the board outline.

autoroute Options for automatically routing the connections in a PCB.

back annotate Updating schematic information from changes made to the printed circuit board layout.

batch load Process of reading multiple Gerber files into Advanced PCB, translating the Gerber information into PCB design objects and placing them on the appropriate layers.

batch mode Option for outputting multiple layers to a single file or printout.

beep Sound used by the computer to signal or prompt the user for some action.

bias See layer bias

Bill of Materials Or BOM. A list of the components (including quantities) used in a PCB.

binary rule A Design Rule that tests for a condition between two design objects. An example is the clearance rule, which specifies the minimum clearance between one object and a second object.

blind via Via connecting one of the outer layers to an internal layer.

Bottom layer Signal layer for the bottom (or "solder side") of the PCB.

break Conversion of a single track segment into two connected segments.

broken net marker A dashed line which ties together two sub-nets in a broken net.

buried via Via connecting two internal layers. Buried vias are not visible on the outside of the board.

button Graphical icon used to perform process launching.

button editor Allows you to change the process launched by a button and assign the bitmap of a button.

check print A print or plot of multiple artwork layers, one on top of the other. Used to verify PCB artwork.

class A number (1-32,000) used to group nets together for routing by an autorouter.

Clear To remove an selection permanently from the workspace. Same effect as Delete. See also; Cut, Copy.

clearance The specified minimum air gap that separates each electrical primitive (signal layer pads, vias, tracks, fills, etc.) from other electrical primitives.

Client Short for EDA/Client. An application which runs under Windows, providing the user interface and network support for EDA servers.

Client Basic Programming language in which macro scripts can be written to run in the EDA/Client Server environment.

Client Menu Menu through which you control EDA/Client. This is where you install, remove and configure servers, customize and edit resources, setup user preferences and run script files.

Client Pascal Programming language in which macro scripts can be written to run in the EDA/Client Server environment.

Client Station A Personal Computer (PC) running EDA/Client.

Client Status Bar The Client Status Bar is the two line bar across the bottom (or top) of the screen. It includes the Status Bar and the Command Status Bar. The Status Bar is dynamic, providing information about the cursor position in the workspace and the current state of the process being executed. The Command Status Bar reports the current process being executed and gives a description of that process.

clipboard Reserved memory used to hold Cut or Copy selections.

command The term previously used to describe any process that is performed by choosing a menu item, e.g., Place Component or File-Save. A menu item is now known as a *Process Launcher,* which is used to launch a *Process*.

comment Optional component text field created when a component is placed. Normally used to hold a component value, description or part number.

component A collection of primitives stored as a single entity in a component footprint library. Components generally consist of one or more pads and tracks and/or arcs on the top overlay, which define the component shape, although anything can be stored as a library component.

component text Text that is part of a component. Component text is created at the time the component footprint is placed from a library. It can be moved (including rotate and flip) but cannot be deleted (only hidden). This text remains associated with the component until the component is deleted.

configure server Add and remove resources to the chosen Document Editor, from the pool currently available in EDA/Client. You can also alter the start up state of the server, get information on the .INS and .DLL files it uses, and a list of all the processes the server provides.

connection The logical or physical link between any two netlist nodes. Each logical connection is indicated by a thin connection line on the connection layer. Physical connection are completed routes. Advanced PCB allows connections to be partially logical and partially physical, i.e. partially routed.

Connection Layer The connection layer is a non-physical system layer that displays all the unrouted netlist connections.

connection line There are two types of connection lines in Advanced PCB. From-Tos, which indicate the pin-to-pin connections as they are exist in the netlist. These are drawn as thin solid lines on the connection layer. The second type is Broken Net Markers, which indicate that a net is only partially routed. This condition is also described as the net being broken into sub-nets, with the sub-nets connected by Broken Net Markers. These are drawn as thin dashed lines on the connection layer.

contention A contention exists when an object has the same rule kind with the same rule scope applied more than once. For example, a pad may be covered by two solder mask expansion rules which apply to different regions, but the regions overlap. The particular contention is resolved by applying the largest expansion rule to this pad.

copper Any non-etched (conductive) portion of any signal layer of a printed circuit board. The copper shape is defined in Advanced PCB by placing primitives on a signal layer.

copper sharing where two connections on the same net share tracks, vias, etc.

Copy To add a selection to the clipboard without removing it from the workspace. See also; Clear, Cut.

cross probe A technique of locating matching objects in different documents. Advanced PCB supports cross probing of most objects, including nets, pins and parts to and from Advanced Schematic. You can also cross probe to documents open in other EDA/Client Editors, such as the text editor. Cross probing is performed by the Client:CrossProbe process.

current layer Advanced PCB is a multi-layer design environment. The current layer is the layer on which a single layer object (track, string, arc, fill or single layer pad) will be placed. The current layer is identified by its Layer Tab at the bottom of the workspace being active.

current library list List of libraries whose components are currently available in the PCB Editor.

current origin User definable origin which can be set anywhere in the workspace. The Status Line coordinates indicate the current cursor position relative to this origin. Also referred to as the relative origin. Default position in a new document is the lower left of the workspace. See also; absolute origin.

cursor The graphic “pointer” or selection tool used to select or position objects in the workspace.

customize server Customize the resources for the active Document Editor. Customization includes; selecting another resource from the list of resources available to this Document Editor (perhaps choosing your own specialized menu), and toggling the display state of toolbars.

Cut To clear a selection from the workspace and copy it to the clipboard. See also; Clear, Copy.

default Program settings or options which remain selected until changed by the user. Environment type defaults (files open, size of windows, etc.) are stored in the CLIENT.INI file. Server relevant defaults are stored in the ADVPCB.INI file. Design relevant defaults (active layers, grid, etc.) are stored with each design.

de-select Releasing the selected condition of an item (or group) in the document window.

designator Also called component label. The unique identifier assigned to each component in a circuit.

Design Rule Check(DRC) Tool for checking the PCB database against the current set of design rules.

design rules The PCB design requirements are specified by defining a set of design rules. Advanced PCB incorporates 21 design rules, which allow you to monitor / specify requirements such as copper clearance, track widths, mask expansions, and so on. Each design rule identifies the set of objects it is to apply to by its rule scope.

document User-generated or auxiliary file.

documentation Information that explains how to use the software. It can be provided in an electronic format and a physical format.

double-sided Refers to a PCB with tracks on both sides of a single laminate layer.

drag Move an object along with any connected objects.

draft code A code used to identify each aperture in an aperture file or in a Gerber format photoplotter file. Aperture draft codes (also know as D codes) are typically in the format of a “D” followed by two or three digits. Certain draft codes are reserved for commands in the Gerber standard. As a general guide, number apertures with draft codes greater than 10.

draft mode The display, plotting or printing of primitives (tracks, pads, arcs, fills, etc.) in outline, rather than filled, form.

Drill Drawing A special plot that uses coded targets to indicate the position and size of each hole on a PCB.

Drill Guide A special plot, similar to a pad master, which marks the exact position of all holes on a PCB.

EDA An acronym for "Electronic Design Automation". EDA is used as a label for software tools used for product development in the electronics industry.

EDA/Client™ An application which runs under Windows, providing the user interface and network support for EDA servers.

EDA document The design or database produced by an EDA Document Editor.

EDA Document Editor The Document Editor is where the user performs the actual editing; typing in text, placing wires, moving tracks and so on. Each server will contain none, one or more document editors. An example is the Advanced PCB Server, which has two document editors, the PCB Document Editor and the PCB Library Document Editor. Another example is the Netlist Server which has no document editor, the Text Expert Document Editor being used to display its results.

EDA document type Each Document editor will produce a different document type. The file extension gives an indication of the document type, for example - .PCB, .NET. These extensions are not fixed, any extension can be used.

EDA document window The document window is the window frame through which you view your document. It includes the title bar with the maximize and minimize buttons and the window frame with the scroll bars (if required).

EDA Editor Short for EDA Document Editor.

EDA Editor Tabs Tabs located along one edge of EDA/Client, used to switch between Document Editors.

EDA Editor Panel The EDA Editor Panel appears down the left of the screen. A Document Editor may have a panel to provide easy access to some of the features available in that editor. For example, the PCB Document Editor has a panel which allows you to add and remove libraries, select components from the active library and also browse through objects placed in the workspace.

EDA Server EDA Servers provide the 'services' in the EDA/Client Server environment. These servers may include text editors, schematic servers, simulation servers, PCB servers, autorouter servers, and so on.

Edit Menu Menu through which you can perform editing functions on the document such as; cutting, pasting, placing, moving, changing, etc.

electrical grid Defines a range within which a moving electrical object (such as a track, pad or via) will attract to another electrical object. The object being moved will jump to a hot spot on the nearest electrical object within the electrical grid range. See also; hot spot.

Excellon Standard file format for numeric control (NC) drill equipment used to automatically drill PCB holes.

exception handling Process followed when the executable code encounters an invalid condition. These conditions are trapped, allowing for graceful recovery. If there is an error condition which causes the application to close, the exception handler will attempt to restore data when the application is re-started.

fabrication layer Artwork used as a reference in fabricating PCB layers. See also; Mechanical layer.

File Menu Menu through which you can perform file related operations such as creating, opening, saving and closing files.

fill A rectangular object which can be placed on any layer. On a copper layer fills are used for shielding and/or supply of high current. Fills can also be placed for "non electrical" uses, such as defining "no-go" areas on the Keep Out layer.

focus The state of an object when it is visually identified as "having the focus" or "being in focus". Only one object can be in focus at any time. The object that is in focus is displayed differently to other objects. If it can be graphically modified it will display editing handles (a fill has re-sizing handles at each corner and along each side, and a rotation handle near the center). If it cannot be graphically modified it will display a focus cross-hair. To focus a particular object position the cursor over it and click LEFT MOUSE. To graphically modify the focused object click on an editing handle and move the mouse to drag that handle. Click anywhere on the focused object to move the object. To de-focus (or focus on nothing) click in free space. Not all objects can be focused. See also *selection*, *handle.*

footprint The set of primitives placed in the PCB Library Editor to "hold" the component. When the footprint is placed on the board it is assigned a designator and then becomes a component.

forward annotation The process of bringing forward design changes from the schematic into the PCB. The design changes supported include: Adding a node, net or component; Removing a node, net or component; Changing a net name, component footprint, component designator or component comment.

free pad Any pad that does not belong to a library component. Free pads are identified by the default empty pad designator when placed.

free primitive Any primitive which is not part of a group object is a free primitive. Group objects include; components, dimensions, coordinates and polygons.

free text Any text which is not associated with placed components. See also; component text.

from-to When you load a netlist Advanced PCB displays the pin-to-pin connections in each net as a series of thin lines. The thin connection line that connects each pin in the net to another pin in the net is called a From-To, going From one pin in the net To another pin. The From-Tos are collectively referred to as the *Ratsnest.*

Gerber format The RS-274 format, a standard file format adopted for coding photoplot files in terms of draft codes and coordinates. The draft codes control the aperture to be used and the opening and closing of the shutter. Coordinates give the position of flashes and strokes on the plot. Command codes instruct the photoplotter when to turn the light source on and off.

Gerber plot A photoplot stored in or created from a Gerber format file, also used generically to refer to any photoplot.

global change A change which is to extend beyond the object currently being edited to other objects of the same type in the PCB.

grid A system of defining points in the workspace, relative to an origin. All objects placed in the workspace are placed on the current snap grid, relative to the current origin.

group object A group object is any set of primitives which has been defined to behave as an object. These may be user defined, such as components and polygons, or system defined, such as coordinates and dimensions. A group object can be manipulated as one object - they can be placed, selected, copied, changed, moved and deleted.

guide hole Hole used for manual drilling of (typically prototype) PCBs.

handle Small identifying markers displayed on the object that currently is in focus. Handles mark points where the object can be graphically modified. Square handles are for re-sizing, circular handles are for rotating. See also *focus.*

hardware arc When plotting, arcs which are created by the plotter, from coordinate, line width and radius information. Some plotters support this option, if they do not you must use software arc descriptions generated by Advanced PCB.

Help Menu Menu through which you can open the on-line help files.

hot spot A point on an electrical object which another electrical object will jump to if the electrical grid feature is enabled. The hot spots on each primitive are; track center line, pad center, via center, arc ends, fill corners.

imperial Inch-based measurement system - Advanced PCB uses the mils (.001inch) as its default unit. Measurements are stored in imperial format regardless of the display mode.

ISO International Standards Organization. See also *metric.*

Keep Out layer Special layer used to define a board perimeter and "no go" areas for placement and routing.

keyboard mapping Mapping of keystroke(s) to process launchers.

keyboard shortcut Key stroke(s) used to perform process launching.

Keyboard Shortcut Editor Allows you to map key stroke(s) to processes.

keyboard shortcut list List of shortcut key assignments.

lattice Polygon Plane filled with an open grid of crossed track segments.

layer Printed circuit boards are constructed from one or more layers. Photo-tools (or master artwork) used to fabricate these layers are generated as individual plot or print files, based on the contents of each workspace layer.

layer bias Practice of alternating the principal direction for track routing on PCB layer pairs.

layer pair Multi-layer boards are normally fabricated as a set of thin double sided boards which are then sandwiched together, each separated by an insulator. Each original thin double sided board is called a layer pair. This fabrication technology supports vias passing from top to bottom of each layer pair. These vias are called blind and buried vias. See also; blind vias and buried vias.

library Collection of component footprints stored in an Advanced PCB library format.

Library Editor Document Editor used for creating component footprints and managing Advanced PCB component libraries.

load pin A pin can be set to be a *source*, load or *terminator* pin. This setting is used when a net topology is applied to a net, affecting the place of this pin in the topology.

Local Area Network A network whose domain is local, typically within an office or building.

Macro Execution of a sequence of jobs. May include other processes.

Macro Server Server which interprets Macro script documents.

Macro Script Set of instructions and parameters written in a Macro Script Language.

Mechanical layer Any of four layers which can be used for displaying fabrication and assembly details. The contents of each mechanical layer can be added to all other layers when the output is generated.

Memory Monitor Monitors the availability of free memory and free resources in the Windows environment. When either falls below a preset level a Memory Monitor Warning pops up. The preset levels are user definable.

menu List of menu items.

Menu Editor Allows you to define menus and menu items and map them to process launchers.

menu item A label that appears in a menu, which when chosen will either launch a process or display a sub-menu.

merge To move components from one library to another.

metric Metric-based measurement system using mm (millimeter) as the base unit of measure for PCB design and fabrication. Advanced PCB stores all dimensions in imperial format, regardless of the display mode.

Microsoft Email Electronic mail that is handled by the Microsoft Email software.

Mid layer Any of fourteen internal layers which can be used for routing the connections on a multi-layer PCB.

mils Unit of imperial measure equal to .001 inch.

minimum X, Y The minimum X or Y coordinate of items in the Advanced PCB workspace. This describes the left-most and bottom-most coordinates used in the file or plot.

mm Millimeter - unit of metric measure.

move Move an object without regard to other objects which may be “connected” to it. See also; drag.

Multi layer Special display layer used for storing objects which are to appear on all copper layers. Typically used for pads and vias that are to appear on each layer of the PCB.

multi-layer A multi-layer board is one which is made up of two or more sheets of board laminate, which allows electrical connections to be made on a choice of several layers. See also; layer and layer pair.

net A net is a set of nodes that are to be electrically connected. Each node is a component pin. See also; connection, node.

net topology The pattern or arrangement of how the nodes in a net are connected to each other. If a specific topology has not been applied to a net Advanced PCB will apply the shortest topology to the net.

netlist A text file which lists all the connections of an electronic circuit and the components used. The standard Protel format separates the netlist into two sections, component information and nets. Netlists are used to transfer design information between EDA systems and to verify the contents of a design.

netlist macro When you select a netlist to load Advanced PCB searches for differences between the netlist and any PCB design data present in the workspace. Each difference is recorded as one of ten different types of Netlist Macros including: *Adding* a node, net or component; *Removing* a node, net or component; *Changing* a net name, component footprint, component designator or component comment.

netlist node A component pin that is part of a net.

Netlist Server Server which creates a netlist in the chosen format.

node A component pin that is part of a net.

object Any item that can be placed in the Advanced PCB workspace and manipulated as one item. Includes primitives (track, pad, via, arc, fill, string) and groups (component, dimension, coordinate, polygon).

On-Demand Start up state of server where it is available for use but not loaded into memory.

On-Line Help User documentation in the Hypertext On-Line Help format.

On-Line Manual User documentation which appears in the same format as a book, except that it is accessible on-line.

origin Location of the 0,0 coordinate in either a PCB file or a plot. Advanced PCB has two origins, the Absolute Origin which is the extreme lower left corner of the workspace, and the Current Origin which can be set anywhere in the workspace.

orthogonal Drafting standard where lines are constrained to either vertical or horizontal placement – a common practice in PCB design. See also; any angle.

Overlay Special layers of PCB artwork, also called the (Top or Bottom) silkscreen layers. Overlays are used to identify components on the top or bottom of a PCB, and are provided as an aid to assembly and maintenance of the PCB.

package The physical description, or "footprint" of a component, e.g. DIP16, defined by the number and location of pins, overlay, etc.

Pad master A special plot type that includes all the pads in the PCB.

pad A design element used to locate and connect tracks to component pins on a PCB, also called a land. Pads can be single layer (exist on one layer only) or multi-layer, existing on all copper layers.

pan The ability to move the viewing area of the screen as you work on a magnified area of the document window. Advanced PCB provides automatic panning whenever performing an edit type function, such as selecting, placing or moving.

Paste Mask Special plot automatically created from SMD pads, used to define a mask for applying solder paste for "hot re-flow" fabrication. Positive and negative expansion factors can be applied globally to all pads or locally for individual pads. Paste masks are created in the negative for plotting efficiency.

placement Position or arrangement of components in the workspace.

Polygon Plane Solid or lattice plane created by defining a polygon perimeter on any layer. On copper layers polygon planes can automatically connect to a designated net and will pour around all objects on other nets.

primitive object The most basic object available in the Advanced PCB design environment. The set of primitives includes; track, pad, via, fill, arc and string. Primitives are used to create group objects (components, dimensions, etc.) and the other entities that make up a printed circuit board, such as; the board outline and mechanical details, etc.

printed documentation User documentation which is presented in a book or set of books.

process Execution of a sequence of jobs. To perform any operation in EDA/Client or in a server, a process must be executed. All processes can be launched from menu items, tool bar buttons and keyboard short cut keys.

Process Identifier Each process is identified by its Process Identifier. An example of a Process Identifier is PCB:PlaceTrack. This Process identifier tell us that the Advanced PCB server can execute a process to place a track.

process launcher Instruction to execute a process. Any user initiated action that begins a process. Includes: menu items, tool bar buttons, mouse and keyboard shortcuts.

process long summary The process long summary provides a brief description of the process and is used for the tool tip and the Status Bar description.

process parameters Each process may require parameters to successfully execute. If a process requires parameters, it will prompt for these through dialog boxes and / or mouse actions. Process parameters can also be passed by process launchers and by macros.

Project Manager A panel that displays an icon to represent each open document and indicates any relationship between them. If the documents are related they will appear in a nested fashion with a line linking each child document to its parent. You can select another document to be the active document by clicking on it in the Project Manager.

raster device Device which creates an image as a series of small points. A raster device passes across the page or film, moves down slightly, passes across the page or film again, moves down slightly, and so on, building up the image.

ratsnest When a netlist is loaded into the workspace a pin-to-pin From-To is created between all the pins in each net. Each net is then a set of these pin-to-pin From-Tos, and these are collectively called the ratsnest.

Resource Menus, Toolbars and Keyboard Shortcut Lists are resources.

Resource Editor For each of the three sets of resources available in EDA/Client (Menus, Keyboard Shortcut Lists and Toolbars) the Resource Editor allows you to create, edit and delete resources. The Resource Editor has access to the entire pool of resources currently available in EDA/Client.

Resource File Each server has a Resource (RCS) file. It holds the default definition of all the resources provided by that server. These include the mapping of each menu item to the process identifier of the process it launches, the mapping of each toolbar button to its button image file and the process identifier of the process it launches and the mapping of shortcut key strokes to the process identifier of the process it launches.

Resource File, Client The resource information for each server that has been loaded is stored in the CLIENT.RCS file. Customization of Client or Server resources are stored in this file.

Resource File, INI INI files hold the environment defaults, such as open files, window configurations, editor preferences, printer setup, etc.

Router The Router handles the flow of information between EDA/Client and its servers and between EDA/Clients throughout the network.

Rule Scope The scope of a rule defines exactly what the rule is to apply to. The available scopes include; Whole Board, Layer, Object Kind, Component Class, Component, Net Class, Net, From-To Class, From-To, Pad and Region.

schematic capture Process of capturing the “schema” or circuit design in a diagram. The term is generally used when the process is carried out electronically, as this “capture” process allows the connectivity information to be extracted and transferred to other EDA environments.

selection A special display state that indicates items included in a selection. Selected items can be manipulated as a group. Items must be selected to be Cut or Copied to the Clipboard.

serial The process of transferring information in a one-bit-wide stream. Used as a general term to refer to the RS-232C and RS-422 standards for data terminal equipment (DTE) communications.

Server EDA Servers provide the 'services' in the EDA/Client Server environment. These servers may include text editors, schematic servers, simulation servers, PCB servers, autorouter servers, and so on.

Server Description File The *server*.INS file, also known as the server installation file. It lists; the server DLL file, the server resource file and the supported document editors. It also lists each process provided by the server and their tool tip description / Status Bar description.

Server Installation Servers are installed by loading their .INS file. Once installed, a server can then be started (unless it is configured with a start up state of Automatic) or left in the On Call state where it will be started the first time you open a document type supported by that server.

shape based router Also referred as gridless routers. Track and via placement is constrained by the shapes of the existing objects and the design rules, rather than the available points on a routing grid.

shortcut key Any key that can pop up a menu or launch a process. Many shortcut keys have default assignments, e.g., PGUP to zoom-in. All shortcut keys are user definable.

signal Any net. Generally used to refer to any non-power net.

signal layers Layers available for routing PCB connections in Advanced PCB, specifically the Top, Mid 1–14 and Bottom layers.

silkscreen See Overlay.

SMD Surface Mount Device. Also SMT (Surface Mount Technology). Components and special PCB assembly techniques for components which attach to either the top or bottom surface of the PCB without using holes, carriers or mounts.

snap grid An invisible array of regularly spaced points in the workspace. The snap grid defines the location points at which an object can be placed in the Advanced PCB workspace. It is calculated from the current origin.

snap to Special property of track placement in Advanced PCB where tracks will "snap to" pad centers, if the track is led to within 10 mils (.010) of the pad center when manually routed. Allows off grid pads to be manually routed.

software arc When plotting, arcs which are generated by Advanced PCB can be replaced by short straight line chord segments (software arcs). Used when the plotting device can not render a curved arc. See also; Hardware arcs.

Solder mask Special plot used to create a mask for the top and bottom layers of a PCB. The mask is a "resist" layer which has openings to expose the pads to the solder, while protecting any tracks, etc. The solder mask is automatically generated in the negative.

solder side Refers to the Bottom side layer of a PCB.

source pin A pin can be set to be a source, *load* or *terminator* pin. This setting is used when a net topology is applied to a net, affecting the place of this pin in the topology.

Start Server To load a server into memory so that it is ready to run a process.

Status Bar The bar along the bottom of the screen (can be repositioned at the top) which displays the current X and Y coordinates of the cursor, prompts for user action and the state of the current process.

string Individual element of free or component text.

Stop Server To remove a server from memory, returning it to the On-Demand state. A server that is stopped is still available and will be automatically re-started when a document requiring that server is opened.

sub-net When a net is only partially routed each part that is routed is referred to as a sub-net. For example, if you delete a single track segment from a net that is completely routed you break the net into two sub-nets.

syntax Valid arrangement of words, identifiers and expressions for a given language. In Text Expert the valid set of words, identifiers and expressions are; reserved word, symbol, string, number, comment and identifiers.

Syntax Highlighting A technique for document highlighting based on the syntax of the language, where different words types, symbols and identifiers are assigned different colors.

terminator pin A pin can be set to be a *source*, *load* or terminator pin. This setting is used when a net topology is applied to a net, affecting the place of this pin in the topology.

Text Expert Server which allows you to create and edit text documents. It includes Syntax Highlighting.

thermal relief Connection style where a pad is connected to a polygon or a power plane by strips of copper rather than a solid pad-to-plane (or polygon) connection. This technique is used to limit the flow of heat between the pad and the plane when soldering or de-soldering.

through hole PCB technology where component pins pass through all layers of the assembled PCB.

tool A process, e.g. PCB:PlaceTrack, that can be invoked by clicking a button on a Tool palette and allows you to place an object in the workspace.

toolbar Set of buttons.

Toolbar Editor Allows you to add and remove buttons and separators from a toolbar and assign process identifiers to buttons.

Tools Menu Menu which provides access to the Advanced PCB tools, such as design rule checking, autorouting, re-annotating the design and so on.

topology See net topology.

track Primitive object used to define lines in the PCB workspace. Used to create current or signal carrying traces on the PCB, and any other item requiring straight lines.

track placement mode Defines how tracks can be placed when changing direction during placement. There are seven possible modes - *Any Angle* (allows track to be placed at any angle); *90 Degree Horizontal Start and 90 Degree Horizontal End* (constrains track placement to a horizontal and vertical orientation); *45 Degree Start and 45 Degree End* (constrains track placement to a 45 degree line and a horizontal/vertical line); *Arc Start and Arc End* (constrains track placement to an arc and a horizontal/vertical line). Change the mode by pressing the SHIFT+SPACEBAR during placement. Press SPACEBAR to change between Start and End modes.

unary rule A Design Rule that tests for a condition around one design object. An example is the solder mask expansion rule, which specifies how much the solder mask is to be expanded around a pad or via.

User Guide Documentation that explains how to install and use the software and the theory behind it.

vector font Special fonts designed for vector output devices, such as pen plotters and photoplotters.

vector device A vector device creates the image by "drawing" a set of lines to create the image. Information is passed to the vector device as a series of coordinates and instructions about what to do at each coordinate (e.g. - pen down, light on, etc.). See also; raster device.

vertex Point where two connected track segments meet.

via Or through hole, a special purpose pad with a drilled (normally plated) hole, used to connect tracks on different layers. Advanced PCB vias are multi-layer, or connect two layers of a multi-layer board (blind or buried vias).

View Menu Menu through which you can change your view of the active document and also turn on and off other screen facilities such as status bars, the Panel, etc.

visible grid Two independent, user-definable display layers which provide a visual reference for positioning items accurately on-screen.

Window menu Menu through which you can re-arrange and re-order the open document windows.

workspace The document window area where items can be placed or moved.

X, Y size The difference between the minimum and maximum coordinates used on each axis of a PCB or plot, i.e. the height and width of the board.

Index

D

E

F

K

L

M

R

S

T