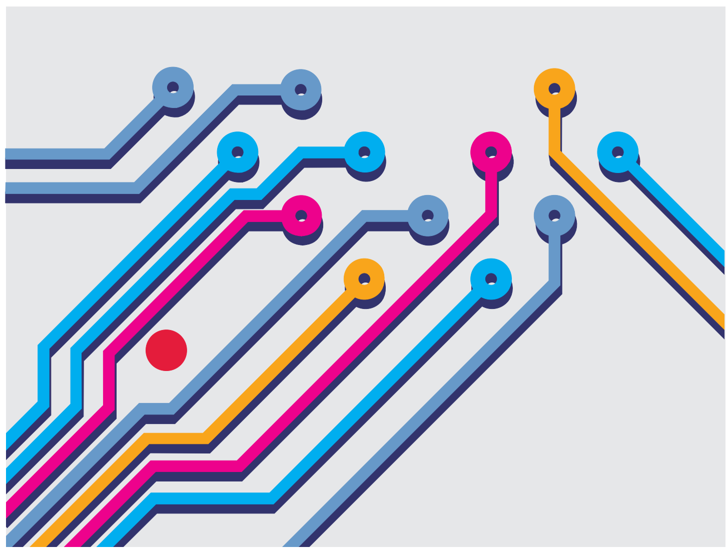
Advanced PCB™

Printed Circuit Board Design System for Windows



Professional 32-bit PCB design system for Windows

Options: **Advanced PCB, with design automation & productivity tools**
Advanced Place, intelligent component auto placement
Advanced Route, 16 layer rip-up / retry autorouting

Protel Technology

On-Line User Guide

6/95

Advanced PCB

Protel Technology Inc
4675 Stevens Creek Blvd. Suite 200
Santa Clara, California 95051 USA

Sales: (800) 544-4186
Business: (408) 243-8143
Facsimile: (408) 243-8544

Protel Technology Pty Ltd
Technopark, Dowsings Point, Tasmania 7010 Australia
Postal address: GPO Box 204, Hobart, Tasmania 7001

Sales: (008) 03 0949
Business: (002) 73 0100
Facsimile: (002) 73 0944

Contents

Change commands

Nets and netlists

Auto placement

Introduction

This section provides an overview of the Protel design environment, including many of the tools, features, key concepts and terminology used in the basic 32-bit PCB design system and in the Advanced PCB, Advanced Place and Advanced Route options.

Advanced PCB capabilities

The Protel Design System combines the ease-of-use associated with Windows applications and the advanced capabilities of a professional electronics design automation system. New users will find this system easy to understand and easy to master. Once you have mastered one Windows application, you already know a lot about other applications because of the standardized way that tasks are performed, such as loading files.

In 386 Enhanced mode, Windows 3 provides virtual memory capability, allowing users to design without restrictions on the total number of components, nets, tracks, etc. All Protel design tools support the multiple document interface (MDI) standard. You can load any number of files at the same time, use standard Windows routines like Cut and Paste to move information between files. You can even run multiple applications, using Windows multi-tasking capabilities.

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.

32-bit PCB design

The basic 32-bit PCB system 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 fab and assembly drawings for your design. Boards can be as big as 100 inches (or 254 cm) square. 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.

Protel Design System PCB options

This guide refers to the entire system generically as *Advanced PCB.* All options (except for Advanced SB Route) are documented in this guide and in the *Reference.* Options available for this system include:

PCB Design System

The basic 32-bit PCB layout system with 20 layer design capability, Gerber input and output plus N/C drill output capabilities.

This is Protel's basic PCB layout system, which can be upgraded with the following options:

Advanced PCB features

All the features of the standard 32-bit system, plus global change capability, array placement, Gerber batch file loading, global editing, auto placement and the line probe autorouter.

These features are dimmed in menus and dialog boxes, until the Advanced PCB module is activated during installation or when upgrading from the standard-level PCB system. Information on upgrading to Advanced PCB can be obtained from your Protel dealer.

Advanced Place features

High-performance global auto component placement tools for Advanced PCB. These features are dimmed in menus and dialog boxes, until the Advanced Route is activated. Information on upgrading Advanced PCB with Advanced Place can be obtained from your Protel dealer.

Advanced Route features

High-performance rip-up/retry maze autorouter for Advanced PCB. Information on upgrading Advanced PCB with Advanced Route can be obtained from your Protel dealer.

All of these options are documented together in the *User Guide*, *Reference* and On-line Help systems.

Advanced SB Route

State-of-the-art, shape-based (or gridless) autorouter for Advanced PCB. Information on upgrading Advanced PCB with Advanced SB Route can be obtained from your Protel dealer. Advanced SB Route is documented separately, in its own *User Guide* and *Reference.*

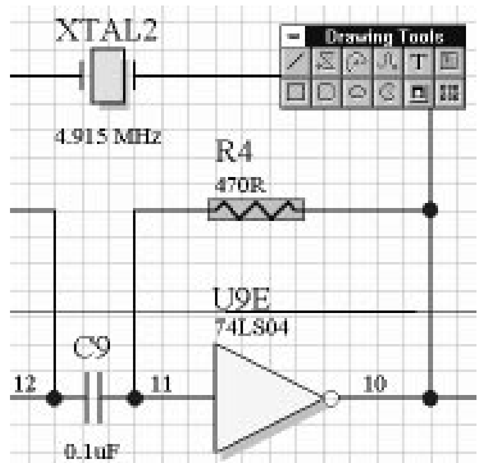
➡ Obtaining access codes for the Advanced PCB, Advanced Place and Advanced Route modules can be as simple as placing a phone call. Advanced SB Route is a separate application delivered with its own documentation. Contact your Protel representative for details.

Unlicensed options are included in menu commands and dialog boxes. This options are disabled until the appropriate access codes are entered in the Help-About (Set Access Codes) dialog box. See the *Environment Guide* for details.

Advanced PCB links to schematic capture

Full support is included for netlist-based design entry. Importing a schematic netlist allows you to take full advantage of Advanced PCB's auto component placement, autorouting, engineering change order and design rule checking facilities.

Advanced Schematic



Special links are provided for Protel's Advanced Schematic, including bi-directional cross-probing between sheet and board files as well as back-annotation, forward annotation and engineering change order (ECO) support.

Design system documentation

The printed documentation (*Environment Guide, User Guide* and *Reference*) and On-line Help system have been designed to guide the new user through the many features of Protel's PCB design system and to simplify the retrieval of specific information once you have a working knowledge of the package.

These manuals assume that you are familiar with Windows icons, menus, windows and using the mouse to make selections. We also assume that you have 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 User's Guide*.

Protel's PCB design system is similar in operation to other Windows applications. Once you have mastered a few Windows basics you'll be ready to learn the Protel design system.

Inside the manuals...

This system is delivered with an *Environment Guide*, *User Guide* and *Reference*. The *Environment Guide* features basic information about the Protel design system, including installation instructions. This *User Guide* provides a broad overview of PCB system concepts with detailed explanations of key procedures plus a glossary of Advanced PCB terminology. The *Reference* includes specifics for all Advanced PCB command and dialog box options. The reference also includes file format descriptions, command shortcuts and library listings.

Much of this information is also available in the On-line Help system, available from inside Advanced PCB. To access help, just press the F1 key at any time while using the software.

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 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*. Step-by-step instructions for performing an operation are generally numbered as in the following examples:

italic indicates anything to be typed. Always enter the italicized information exactly as it appears.

bold place-holder for information that you provide.

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 commands (e.g. File-Open), dialog box names (e.g. Set-up Layers) or option names (Auto Via). Command sequences are hyphenated – File-Open means choose the File menu and the Open command.

SHIFT+ALT The + sign means: hold down the SHIFT key then click 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.

This manual also includes some special terminology – words like *track* or *via* that are unique to PCB design or words like *browse* or *attribute*, that have some specific meaning within the Advanced PCB design environment.

Definitions for these words will be found in the Glossary, at the end of this guide. Your Windows documentation includes definitions for any special Windows terms.

Advanced PCB features

Protel’s PCB design system works 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 the File Manager to locate and organize your documents. The way the PCB design system uses tools, menu commands and shortcuts will also be familiar. For example, using combinations of ALT and other keys to execute menu commands.

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, etc. 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 series of layers which correspond to the individual "tools" 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.

The Advanced PCB design process includes:

1. Selecting the layers that will correspond to the PCB tooling required for your design. These can include a top (component side) layer, bottom (solder side) layer, up to 14 additional mid layers, up to four power plane layers, top and bottom silkscreen layers and up to four

mechanical drawing layers. Other special purpose layers are also supported.

2. Placing the required component patterns from a component library. Or, if working from a schematic netlist, using automatic component placement to load and position the components into a pre-defined board outline.
3. Connecting the component pads with tracks, manually, with the autorouter, or by combining manual and automatic track routing strategies.
4. Checking your work using built-in design verification tools and reports.
5. Printing or plotting the required manufacturing artwork using any Windows-supported output device or the Gerber and NC drill file output options.
6. Generating additional design documentation, such as Bill of Materials (BOM) files or Engineering Change Order (ECO) files.

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 assembled.

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.

➡ Connectivity works at two levels – with the logical (netlisted) connections and with the physical geometry of connected tracks, pads planes, etc.

This concept, connectivity, is the basis of automatic component placement, autorouting and design rule checking. Connectivity also extends to manual layout. For example, you can drag components and connected tracks will stretch (or *rubberband*), without being broken. The system stores this connection information which can be used to generate a netlist directly from a routed PCB. This netlist can be used to facilitate checking of the completed design.

Protel's PCB design system will also generate reports, such as a Bill of Materials (BOM) listings; Back Annotation files; NC Drill and Pick and Place reports for board fabrication and assembly; Engineering Change Order (ECO) reports and other design documentation.

Some key features and benefits of the PCB design system PCB design system are described below. These items will be discussed, in detail, in the relevant sections of this manual. The descriptions include features of Professional PCB, and additional features and capabilities that are part of Advanced PCB and the Advanced Place and Advanced Route options.

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. Selections can be manipulated using standard Windows Edit menu commands like Cut, Copy, Paste or Clear; moved; flipped on either axis; rotated in .001 degree increments; Imported or Exported as files; saved as library components.

Powerful editing options

Attributes can be edited by double-clicking directly on the item to open a dialog box. In Advanced PCB, changes 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 components and other primitives.

Library system

Multiple libraries can be opened simultaneously. 316 component patterns, including through-hole and SMD footprints are included with the standard PCB design system. Components can be placed directly from a Browse dialog box. Simultaneous multi-user library access is supported for network installations.

Special strings

Special purpose pre-defined strings allow the user to place date, layer or sheet name, filename, component count or other information to be interpreted at plot time. For example, a multi-layer string called ".LAYER_NAME," when plotted, places the correct layer name on each layer. Automatic dimensioning and coordinate markers are also provided.

Multiple fonts

Three improved display fonts (San serif, simple and Serif) support vector plotting and photoplotting. Protel's PCB design system also allows the user to substitute built-in printer fonts for the display fonts when printing. Component designators and comments can be pre-set in 18 horizontal and vertical orientations.

Fractional arcs and rotation

Arc placement resolution of .001 degree and .001 degree rotation of any selection. Arcs can be placed on any layer. Connectivity checking works with signal layer arcs.

On-line and batch Design Rule Check

On-line DRC signals clearance violations and net collisions (when netlist is loaded). High-speed "batch" DRC checks allow quick verification of board layouts to user-specified physical / logical properties.

Thermal relief control

Thermal relief pads where power pins are connected to special power plane layers, can be user-defined (both the conductor path width and air-gap), with 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 are automatically removed when printing or plotting artwork.

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 size.

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. Professional PCB loads and displays generated Gerber files. Advanced PCB batch loads Gerber files with each plot assigned to a PCB layer.

Windows support for printing and pen plotting

Dot matrix and laser printing, pen plotting and PostScript® output are all controlled from common Print Commands. Any device supported by Windows 3 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. A comprehensive set of print/plot/Gerber and N/C drill options provide complete control over PCB artwork generation.

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.

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 adaptors 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)

Blind and buried vias

Vias can be either through the whole board or through any single layer pair. Blind and buried vias can be placed by the autorouter or manually edited by layer pair. Vias are displayed to indicate layer status.

Density report

Advanced PCB generates a non-linear density map which accurately predicts routing difficulty for a placed board.

Intelligent polygon planes

Solid or lattice polygon planes can be placed on any signal layer with automatic connection to a specified net. Copper "pours" automatically, wrapping around all placed arcs and non-orthogonal primitives. Polygon shapes can be defined using line or arc perimeters and vertices can be moved, added or deleted after the polygon is generated. A Re-pour Polygon command allows regeneration around new obstacles and the user can redefine polygon design rules (grid, fill-type and net assignment) each time the polygon is re-poured.

Powerful repeat 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. Repeated item can be rotated around its own axis.

Undo and Redo commands

Multi-level Undo and Redo commands work for all physical changes to the board layout. User can make multiple changes, backtrack using Undo, then reinstate each “undo” change with the Redo command.

Multiple file formats

Protel’s PCB design system loads 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 three different formats: an efficient binary format for fast loads and saves, a text format which can be directly edited by the user, and Protel Autotrax (DOS) text format plus direct export of DXF (AutoCAD®) format files.

ECO system

An Engineering Change Order (ECO) system tracks physical changes made to the board during layout. These changes can be exported via an .ECO file to update a schematic. Schematic ECO files can be used to update the PCB (referred to as *forward annotation*). This system is compatible with the PADS .ECO file format.

Forward and back annotation

Advanced PCB updates a design file every time a new netlist is 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 passed back to Protel’s Advanced Schematic for *back annotation*.

Editable nets

Netlisted information can be updated by physically editing the PCB connections, using Add Node, Add Net and Delete Node commands.

On-line Help system

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

Getting started

Before installing or using Advanced PCB, we ask that you take a few moments to read the Protel Software License Agreement at the beginning of the *Environment Guide*.

1. 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 serial number is printed on the hardware lock and displayed when you use the Help-About command. This permanent identification number is used to generate all option licenses. Quote this number when making any product enquiries.

2. Back up your original install diskettes

We recommend that you make a backup copies of the original Advanced PCB install diskettes and store the originals in a safe place *before* you install the software.

3. Review the README document

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

Your Advanced PCB package includes the following materials:

Protel Design System Environment Guide

Advanced PCB User Guide (this book)

Advanced PCB Reference

Letter with the required access codes (this may be delivered separately).

Hardware lock (unless upgrading from an earlier version).

➡ Hardware locks are packed in a separate white corrugated insert, outside the sealed product box. Avoid accidentally discarding the key, when unpacking.

Protel Software Registration Card (unless purchased as an upgrade).

Advanced PCB - install diskettes.

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

Hardware and software requirements

Advanced PCB operates with Windows version 3.1 (or later) under two modes: *Standard* and *386 Enhanced*.

Standard Mode

Standard mode requires a 80286-based machine with a minimum of 1MB (640K standard memory and 256K extended memory). Advanced PCB requires at least 4MB of RAM (8MB or more is recommended) and a numeric co-processor.

386 Enhanced mode

386 Enhanced mode provides access to Windows virtual memory and multi-tasking capabilities. This mode requires a 80386 or 80386SX-based machine. Enhanced mode offers substantial performance improvements over standard mode and is highly-recommended.

➡ Advanced PCB requires a math co-processor (or software emulation of a floating point co-processor). A 486 DX system includes an integral co-processor and is the minimum configuration recommended by Protel.

Other systems (386 and 486 SX machines) will require an add-in processor for reasonable performance.

Graphics

Advanced PCB requires a minimum 800x600 display. As with other CAD applications, a large display coupled with a high-resolution graphics card will greatly enhance user productivity. Windows especially benefits from a high-performance graphics adaptors and many suitable choices are available.

Printing and plotting hardware requirements

For check prints or simple prototype artwork you can use any Windows-compatible printer, including dot matrix, laser and PostScript® printers or a pen plotter.

For basic production-quality artwork you will need access to:

A pen plotter supports either HP-GL® or DM-PL® plot languages or other plotters supplied with a Windows-compatible driver, or:

PostScript-compatible imagesetting equipment supported by Windows. This equipment is often found in typesetting or graphic arts bureaus.

➡ Newer model large format plotters that use *raster* rather than *vector* control provide the best result when using Windows plot drivers. Older vector-based plotters are poorly supported under Windows. For this reason, Protel has supplied separate internal Pen Plot options that drive HP-GL and DM-PL compatible devices, bypassing the Windows drivers for these devices.

For fine-line production tooling, you will need access to:

Gerber® (or compatible format) photoplotter.

Windows Hardware support

Windows is shipped with a wide variety of output device drivers for both graphics devices and for a wide range of specific printer, plotters and other output devices. Advanced PCB includes two special drivers for pen plotting: an HP-GL driver for Hewlett Packard compatible plotters and a DM-PL driver for Houston Instrument compatible plotters. These special driver bypass the standard Windows versions, which are not optimized for precision vector plots.

New drivers are continually being released for both new and existing equipment. Some of these drivers are currently supplied with Windows, some can be obtained from the Microsoft Windows product support. Microsoft supplies a Driver Library Disk, which is frequently updated with new or improved printer (and plotter) drivers. If you have a question about a particular device, see your *Microsoft Windows User's Guide* or contact the manufacturer for up-to-date information.

Installing Advanced PCB

Follow the simple step-by-step procedure outlined in the *Environment Guide*. Installation of the Advanced PCB program and accessory files will require approximately 5 MB of hard disk space, depending upon the options purchased.

Review the “README” document

Open and read the document README.TXT. This document contains updated information about the current version of Advanced PCB, prepared since this manual was printed.

➡ Notepad allows you to view and/or print this, and other, reference files supplied with Advanced PCB. These files will be identified by the extensions such as .LOG, .DMP, .BOM, .RPT, etc. The Notepad can open any of these text files, once the default extension .TXT has been replaced by *.* (or another mask) in the Notepad File-Open dialog box.

If the Notepad utility is not present, you can review the README document after installation is complete, using a text editor or word processor.

Install will create a new Windows Program Group, called Advanced PCB and inside this group will be installed the icon for Advanced PCB (the executable program). This item is automatically setup under Windows with its own path and a Protel icon. The destination path (supplied during installation) is automatically appended to the AUTOEXEC.BAT file.

Setting-up or upgrading program modules

The *Environment Guide* includes complete instruction for setting-up or upgrading Advanced PCB program modules. You must enter the correct Access Code for each module from the Help-About dialog box.

➡ If installing the Advanced PCB option, you will need to enter codes for the basic PCB module *and* the Advanced PCB module. If you are installing Advanced Place, Advanced Route or Advanced SB Route, enter the additional codes supplied for each of these four modules.

If all the modules are enabled, you can click OK to return to Advanced PCB. If any of the codes entered remain "disabled" try re-typing the code, making sure that you enter the code exactly as supplied. If all the modules remain disabled, and you are certain that the codes have been correctly entered, make sure that the hardware lock is correctly installed on a *parallel* port, not on a serial port.

If you still cannot enable the modules, contact your Protel representative for assistance.

Advanced PCB files

The Advanced PCB installation disk includes the following files:

SETUP.EXE	the installation utility.
INSTALL.EX$	reads the install script.
INSTALL.INS	the installation script file.
INSTALL.BMP	background file for Install.
PFW.EX$	the compressed application.

When PFW.EX$ is decompressed, the following files are written to the destination directory (default C:\PFW):

PFW.EXE	the Advanced PCB application.
PFW.HLP	source file for Protel on-line help.
PFW.LIB	component library file.
PFW.PAD	pad library file
PFW.XFR	OrCAD-to-Protel PCB footprint cross-reference. This file is used to load an OrCAD generated netlist. It links OrCAD part names with Advanced PCB component footprints.
README.EXT	A supplemental information text file (see above).

The following demonstration files are supplied:

DEMO1.PCB	This is a small through-hole layout, in placed but un-routed condition. You can use this board to demonstrate various auto place and router options.

DEMO1.NET	The netlist for DEMO1.PCB and DEMOSMD.PCB. You can use this netlist to demonstrate auto placement, autorouting and DRC features of the system.
DEMO2.PCB	This is a larger through-hole layout in placed but un-routed condition.
DEMO2.NET	The netlist for DEMO2.PCB.
DEMOSMD.PCB	This board shares the same netlist as DEMO1.PCB, but has been laid out with SMD rather than through-hole components
DEMO1AP.PCB	Auto placed version of DEMO1.
DEMO2AP.PCB	Auto placed version of DEMO2. components.

Network installations

Advanced PCB is designed to support Local Area Networks (or LANs) and you can share or exchange files and libraries with other users. File locking is provided to protect PCB and library files, while in use. See the *Protel Design System Environment Guide* and your LAN documentation for further details.

Before installing Advanced PCB on a network, you should be aware that the Protel Software License Agreement (at the front of the manual) restricts the use of this package to a single user, operating a single computer.

In other words, each Advanced PCB user must be in possession of his/her own licensed Advanced PCB package. Contact Protel Technology or your Protel dealer for details regarding multi-user licensing.

Windows memory management

Windows provides virtual memory, when operating in 386 Enhanced mode. Virtual memory allows you to open and save files that are larger than the available RAM. However, you may find that certain editing operations, such as cutting and pasting a large selection, will generate a Windows “Out of Memory” error message. This is because a certain minimum amount of system resources must be available, irrespective of the “swap file” size allocated to the hard disk.

To increase the available memory, try one or more of the strategies listed below:

1. Exit other applications you are running.
2. Choose File-Save All to clear the Undo stack.
3. Close all windows except the active window.
4. In the Options menu, turn off scroll bars, tool bar, and status bar.
5. Exit, and then restart Advanced PCB and Windows.
6. As a last resort, exit Windows, then restart your computer.

If these steps are not sufficient, it may be necessary to temporarily disable network drivers, or other TSRs (Terminate and Stay Resident) programs and utilities that run alongside Windows.

Other ways you can increase Windows memory:

Install DOS version 5.0 or later, which frees additional memory for Windows applications. DOS 5.0 or later is strongly recommended for best performance.

Install additional RAM in your machine. Additional memory will greatly enhance your system’s performance, particularly if using Advanced Place and Advanced Route.

Advanced SB Route requires a minimum of 16MB of RAM for reasonable performance.

Navigating windows

As with other Windows application, Advanced PCB allows you use any Windows-compatible mouse to make menu and toolbar selections and to position items in the document (workspace) window. For example, you click once to activate button commands and double-click to activate many icon or menu selections using the left mouse button.

The left mouse button functions as the enter key in many cases. Similarly, the right mouse button can often be used as the equivalent of ESC.

You can also use the keyboard, including cursor keys, for all Windows operations. For example, press TAB to toggle through dialog box check boxes, enter to accept the “OK” option or press ESC to cancel the dialog box.

For additional information about Windows mouse and keyboard equivalents, see your *Microsoft Windows Users Guide*.

Advanced PCB basics

Starting Advanced PCB

From the Windows Program Manager, you can start Advanced PCB by double-clicking on the application icon or by selecting the Advanced PCB icon and choosing the File-Open command.

When you first run Advanced PCB, a Load PCB File Name dialog box opens. Thereafter, the current workfile (if any), when you Exit, is re-opened automatically the next time you launch Advanced PCB. When opening a PCB file, just use the standard Windows method of changing the drive and directory (path) to display all files with the reserved extension ".PCB" (see "Opening a PCB file," below).

To start a new file, click Cancel (or press ESC) then go to the menu bar and choose File-New (shortcut: f, n).

The Advanced PCB application window

When you launch Advanced PCB, the application window is displayed. This is the main program window which holds any open *document* (or PCB file) windows. Features of the Advanced PCB application window include:

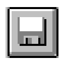
Menu bar

The menu bar displays the main menu commands for the application: File, Edit, Library, Netlist, Auto, Current, Options, Info, Zoom, Window and Help. These commands are all detailed in the *Advanced PCB Reference*.

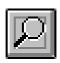
Tool bar

The tool bar, when displayed, appears below the menu bar. It contains buttons and tools to speed many commonly used operations. Tool buttons (from left-to-right) include:

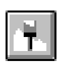
File-Open  Opens a PCB file in its own window.

File-Save  Saves the contents of the current PCB file window.

Print  Prints the contents of the current PCB file window.

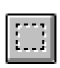
Zoom-All  Redraws the display to include all the current PCB window contents.

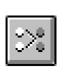
Zoom-Window  Prompts the user to define a new display window area, then redraws the display.

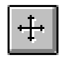
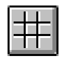
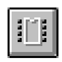
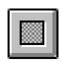
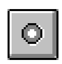
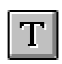
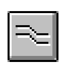
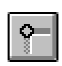
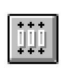
Zoom-Redraw  Updates the screen display without changing the zoom level or center location.

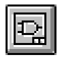
Cut  Removes all selected items from the document window; places selection into clipboard.

Paste  Adds the current clipboard contents to the current PCB window.

Select-Inside Area  Prompts user-defined selection rectangle; items that fall entirely inside rectangle are added to selection.

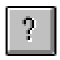
De-Select-All  Removes everything from the current selection.

Move-Selection		Allows the user to move the current selection to a new location in the workspace.
Snap Grid		Pops-up menu for assigning the current snap grid value.
Place-Arc		Initiates arc placement (by center) sequence; current track width value is used as the default arc width.
Place-Component		Initiates sequence to place a component footprint in the current PCB window.
Place-Fill		Initiates sequence to place a rectangular area fill on the current layer.
Place-Pad		Initiates free pad placement on current layer; uses the current pad type as the default pad.
Place-String		Initiates text string placement on the current layer using current font and string size as default.
Place-Track		Initiates sequence to place a track segment on the current layer. Current track width is the default track.
Place-Via		Initiates sequence to place a free via on the current layer. Current via type is the default via.
Place-Array		Opens the Place Array dialog box.
Library-Components		Opens the Library Components dialog box

Run Schematic  Launches Protel Advanced Schematic, if included in the current DOS PATH statement.

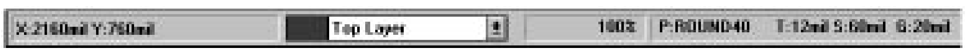
Undo  Removes the effect of the last edit.

Redo  Restores the effect of the last Undo.

Help  Launches the Advanced PCB On-line Help system.

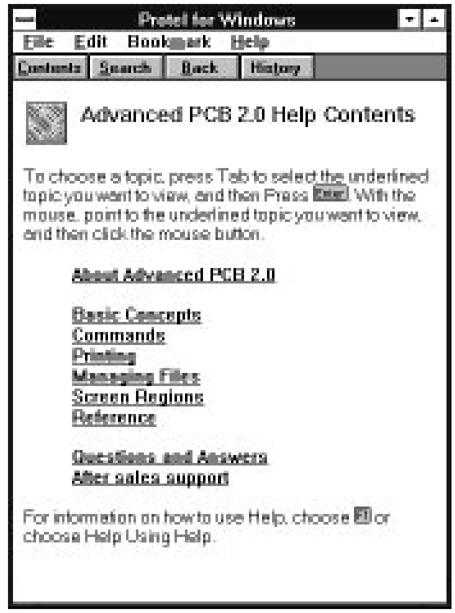
These features will be described, in detail, in the subsequent sections and in the *Advanced PCB Reference*.

Status line



The status line at the bottom of the screen displays (from left-to-right) the current cursor coordinates, the current layer color and name and information about the current activity or mode. This layer box shows the color assignment and name of the current PCB layer. To change the current layer, click the layer selector button. If you choose a layer that is not currently enabled in the Setup Layers dialog box (Options-Layers command) that layer will be activated. The status line can be toggled off or on (Options-Status command).

On-line Help



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

To use Help, press F1 or choose from the available Help menu commands:

Opening a PCB file

Advanced PCB is Windows Multiple Document Interface (MDI) compliant. You can run multiple instances of Advanced PCB and load any number of printed circuit board files, 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 these other file formats is transparent to the user. Translation is performed during the load 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. Details for loading these files are listed below.

The Advanced PCB binary format produces compact files that load and save faster than the text format. The text format allows users to directly manipulate PCB data and allows third party products to interface with Advanced PCB. The ASCII PCB format is documented in the *Advanced PCB Reference*.

When no open document (or *file*) is displayed, the menu bar displays three options: File, Info and Help. The toolbar buttons (if displayed) are dimmed. This dialog box will display the current directory and any files that include the extension .PCB which is reserved for Protel PCB workfiles. To open a previously created file:

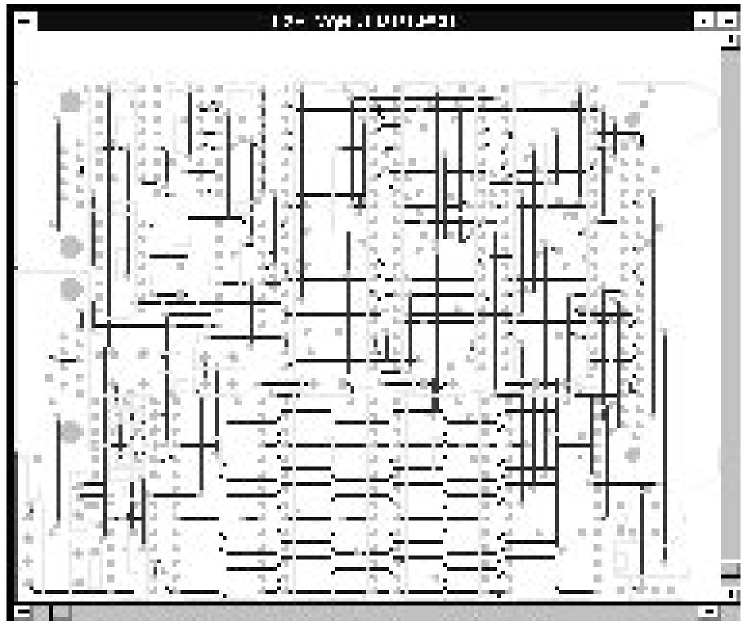
1. Type the filename (include the full path, if different from the path listed) or double-click on directory folders to locate the desired file, then double click the file name.

2. Click OK to open the files.

The selected file will load into the active PCB document window. It may take a few moments to draw the PCB layers in the window.

➡ Abort redraw of the document window, at any time, by pressing SPACEBAR. This allows you to move directly to another command or tool button without waiting for the entire screen redraw to be completed.

The PCB document window



When you choose the File-New command, an empty document window is displayed. This is where components, tracks, text, solid copper fills, etc. are placed. The workspace (document) window is titled with the current path and workfile name, if any. The maximum board area is 100 x 100 inches (254 cm square). The PCB workspace also includes the toolbar, normally displayed across the top of the workspace, just under the Advanced PCB menu bar.

Features of Advanced PCB document windows include:

Title bar

The bar across the top of a window that contains the window's name. You can move a window by choosing the Control Move command from the PCB window. Mouse users can move the active window by dragging the title bar. If a dialog box has a title bar, it can also be moved.

Screen redraw during scrolling

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 SPACEBAR anytime when the redraw is in progress. This saves time whenever you wish to immediately scroll or zoom again, without waiting for the redraw to complete. To make redraw faster, make sure that you are using solid (not transparent) display options for each layer. Working with the Display mode options set to Draft (Options-Display command) for all primitives will also slightly improve redraw speed.

The *Protel Design System Environment Guide* describes many Windows basics including application and document window controls, scroll bars, etc. beginning on page 25.

Learning more about Windows

Windows includes many features to help you navigate applications and documents and to customize the Windows environment for your personal preferences. For more information about these, and other Windows features, please refer to the *Microsoft Windows User's Guide*.

PCB layers

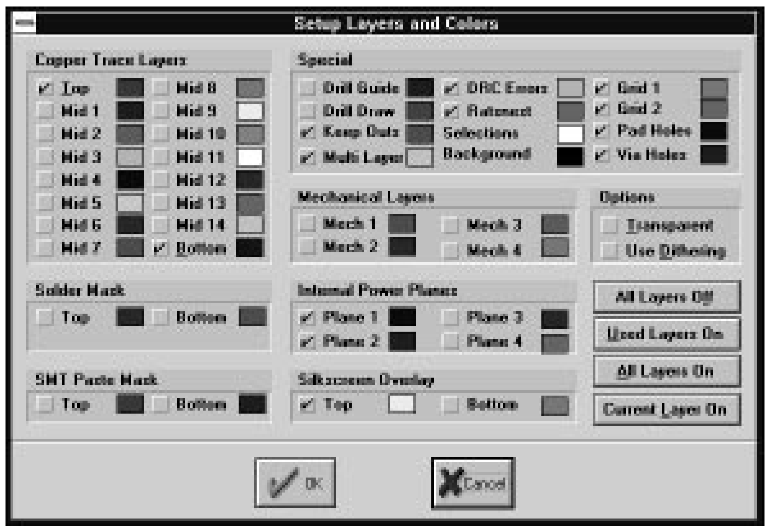
Protel's PCB design system stores and displays a PCB design as a series of layers. Each layer can be assigned an identifying color. Layers correspond (with some special exceptions) to the PCB tooling artwork.

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

This concept of multi-layered design distinguishes this design system 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 "belong" to as single layer, typically tracks, polygons, fills or free text strings.

The Advanced PCB design system includes layers which have a direct relationship to the finished PCB, such as the top, bottom and internal signal layers.

The system also uses special layers which do not directly correspond to finished PCB artwork. One example is the Background layer. This layer is part of the PCB design system display – you can change the background color assignment, but the background is not a PCB artwork layer that can be edited or printed.



The Options-Layers command opens the Setup Layers and Colors dialog box.

To simplify editing of your design, the system allows you to activate specific layers required for your design and to turn the display of these layers “on” or “off” as needed.

Copper trace layers

These are the “signal” layers (16 in all) used for track placement. Other primitives (area fills, text, polygon planes, etc.) can also be placed on these layers. Anything placed on these layers will be plotted as solid (copper) areas in PCB artwork.

Top “component” side signal layer.

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

Bottom “solder” side signal layer.

Internal power 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 reverse, for efficiency. In other words, placing any primitive on these layers will create a void in the artwork.

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 included in other layers when printing or plotting artwork.

Solder masks

Top and bottom masks are provided for photo or silkscreen solder masks. These layers are used to generate masks for wave soldering, usually covering everything except component pins. Vias can be included as an option. You can specify clearances for these masks when printing/plotting. These layers are plotted in reverse, for efficiency.

Paste masks

Top and bottom masks are provided for photo or silkscreen masks of solder paste locations. Used for surface mount device (SMD) boards. You can specify clearances for these masks when printing/plotting. These layers are plotted in reverse, for efficiency.

Other layers

These layers are used to generate board artwork or are used for reference while editing:

Drill Guide Plots of all holes in the layout – sometimes called *pad masters*. Individual layer pair plots are provided when blind/buried vias as specified. These plots include all pads and vias with holes greater than zero (0) size. You can specify a guide marker, which locates the hole center, from the Drill Plots dialog box.

Drill Drawing Coded plots of board hole locations, used to program numeric control (NC) drill equipment. Individual layer pair plots are provided when blind/buried vias as specified. Symbols are plotted at the hole locations, indicating the hole size. A table of symbols, metric and imperial hole sizes and hole counts is included in the plot.

Keep Out This layer is used to define the physical limits for both automatic component placement and autorouting. This is achieved by placing a rectangular perimeter of tracks or by placing rectangular fills, which define any “no-go” areas.

Multi layer Displays all through hole pads and vias – items which occupy all signal/plane layers.

DRC Errors A reference layer that marks the location of clearance errors flagged by the system's design rule check (DRC) system.

Rats Nest Displays un-routed netlist connections as straight line (pad-to-pad) elements. The ratsnest display provides user feedback during automatic and manual component placement and during interactive routing (Auto-Auto Route-Manual command)

These display layers cannot be printed or plotted – but allow color assignment only:

Visible Grids 2 visible grid layers options allow you to display/hide or assign colors to the two visible grids. The grid size is set from the Current menu commands.

Selections Used to display items included in the current selection.

Background Sets the color assignment for the workspace window background.

Pad Holes Used to display the hole defined for a pad.

Via Holes Used to display the hole defined for a via.

The current layer

One workspace layer is "current" at any given time. The color (or pattern) assigned to this layer is displayed in the Layer box near the center of the status line at the bottom of the screen. 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. Selection (for moving, deleting, etc.) is layer-independent – you can perform these operations on any primitives without changing the current layer.

Setting-up display layers

Before you can access any of these layers, the layer must be turned "On." Once a layer is active, it will be displayed or edited:

To activate and setup layers:

1. Choose the Options-Layers command (shortcut: o, l) to open the Setup Layers & Colors dialog box.

Note how the layout and display layers are grouped by layer type. Some layers correspond directly to the tooling of the board, like the Copper Trace layers. Other layers are provided for display or output purposes. For each of the layers on the printed circuit board there is a check box which you can click (LEFT MOUSE) to turn the layer on or off, just to the left of the layer name. An "x" in the box indicates that a layer is active.

➡ The default active layers will be updated when you exit Advanced PCB. Any layers that you have activated will be active the next time you start Advanced PCB. When you load a PCB file, all layers used in the design are automatically activated.

2. Click in the layer check boxes to activate the required layers.

To assign colors to activated layers:

1. Choose the Options-Layers command (shortcut: o, l) to open the Setup Layers & Colors dialog box.
2. Click in the color box, to the right of the layer name to open the Color Selector box.

The color currently assigned to the layer is displayed in the left pane of the Color/Solid box. The Solid color (right hand 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 type, you can choose from up to 224 solid colors or a mixture of solid and dithered colors.

4. Click in the Solid (right) pane to make the solid color equivalent the assignment.

5. Click OK to accept the assignment or CANCEL to exit the Color Selector without changing the layer color.

If you want all colors to display as solid, de-select the Dithered Colors option at the right side of the dialog box. You may wish to use all solid colors if you are displaying primitives in Draft display mode (Options-Display command). This is because the primitives will be rendered in lines of one pixel (screen picture element) width. Dithering will obscure the lines with many color combinations.

Customizing display colors

To customize one of the 224 available colors from the Color Selector dialog box:

1. Choose Custom Selections (shortcut: ALT, c) to expand the Color Selector.

To customize a color, set the Red/Green/Blue values between 0 and 255 units. You can write down the units as a convenient way of recording custom color values for later use.

2. Click OK to accept the assignment or CANCEL to exit the Color Selector.

As you make color assignments, it is important to make sure that your assignments don’t conflict in some way that will obscure vital details when you edit your layout. It is recommended that you start with the Background and Selections assignments (Special layers), if you wish to change a number of the defaults. Remember, some layers may only need to be displayed during printing or plotting (or plot file generation) For example, the Solder or Paste masks can be left “off” unless you are editing or plotting these layers. The more layers displayed, the slower the screen will redraw.

Remember that Although several layers can be “active” and displayed simultaneously, only one layer can be the “current” layer. Single layer items such as tracks, fills, arcs or text are always placed on the current layer. The current layer color (or pattern) is displayed in the Layer box at the left of the Tool bar and the layer name is displayed (along with the filename, if any) in the title bar.

More about color assignment

On the bottom of the Set-up Colors dialog box is a button for setting the specific color to the full 24 bit resolution of Windows. If your system supports 256 colors, then your palette on will be reset to display as many available colors as possible when you close the Setup Layers dialog box. Advanced PCB dynamically reprograms this palette as colors are assigned.

Unless the graphics Card/driver you are using is supplied with a Windows 24 bit driver, Windows will use a System Palette which allows multiple applications to share color assignments. Windows will take the first 20 colors in this palette for itself. These are used for its own graphics and also to try and simulate unavailable colors using dithering. All 20 defaults are defined by Microsoft and all drivers are expected to provide as close as possible to these.

When using applications that take advantage of 256 color palettes, use of more than the standard 20 Windows default colors may cause “stealing” of colors from the system or from other applications. For example, if you have a bit-map as your background for Windows which used 256 colors then the quality of the display of the bit-map will deteriorate as you select more colors for Advanced PCB.

For standard VGA and EGA there are only 16 available colors. Windows “takes” all 16 and color requests from applications are either dithered or matched to the nearest solid color. The application can request either dithered or solid colors.

Transparent colors

You can also make selected display layers Transparent (color and grayscale displays only) rather than solid color. This makes identification of overlapping objects easier when several layers are active at the same time. However, transparent colors will make screen redraw significantly slower. So you may wish to leave this option off, until needed – as during the final cleanup of the board.

For more information about setting up Advanced PCB display layers, see the Options-Layers command in the *Advanced PCB Reference*.

The cursor

The familiar “pointer” tool is used for standard Windows operations, such as choosing from menus and dialog boxes. Inside the document, a special cursor support accurate positioning. This workspace cursor is selected from the Options-Preferences command. Choose from the “Large Cross”, the (default) “Small Cross” cursor or “45 Cross.”

Coordinate system

Coordinates displayed on the status line, at the lower-left of the workspace, indicate the position of the cursor relative to the current workspace origin. The coordinates measure the distance from the origin in mils or millimeters depending on the display option selected.

The absolute origin (the default “0, 0” coordinate) is the extreme lower left corner of the workspace.

Advanced PCB allows you to temporarily set a new 0,0 coordinate anywhere in the workspace. A 0,0 coordinate other than at the absolute origin is referred to as a *relative origin*.

Setting a relative origin

The Edit-Set Origin command sets a relative origin at the current cursor position. To accomplish this, first move the cursor to the desired relative origin, then choose the Edit-Set Origin command (shortcut: ALT, e, o). The status line will now 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 restore the absolute origin (extreme lower left corner of the workspace) choose the Edit-Reset Origin command.

➡ Before using auto placement use the Edit-Reset Origin command to re-establish the default (or absolute) origin at the extreme lower-left corner of the workspace. This can be important because the auto placement routines use the 0,0 coordinate (or origin) as a reference point and because a new origin may move all component pads off-grid where they are more difficult to route.

Grid systems

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.

To allow the designer to fully exploit this built-in accuracy, Advanced PCB provides two independent user-definable grid systems which work independently of one another. The first is the *snap grid* which controls the placement of objects in the workspace. The second grid system is the *visible grid*, which provides visual reference as you move around the workspace.

Both grids can be set from the Current menu. Snap grid coordinates, displayed on the left side of the bottom status line, are relative to some point (the current origin), and can have a value of between 0.001–1000.0 mils (or 0.0025–25.0mm).

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 unless you are using the mouse to place or move a selection. When the cursor keys are used, the cursor always "snaps" to the grid.

➡ If you are using the cursor keys to move the cursor, you will find that Choosing SHIFT and a cursor key makes the cursor jump move 10 times the current setting of the snap grid.

This grid can be changed at any time from the Current-Snap Grid command. The default snap grid is 25 mils.

Setting the snap grid (Shortcut: CTRL+g) 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 measurement unit controls the measurements displayed on the status line. If it is set to imperial measure then measurements will display as mils. If it is metric then measurements will display as mm. The unit of measure changes when you use the Options-Toggle Units command or press Q. See also Metric and imperial units.

Visible grids

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

The visible grid displays a system of coordinate lines (or dots) in the workspace background. The spacing of these lines is determined by the setting under Current-Visible Grid 1 and 2 commands (shortcut: ALT, g). The display of the visible grid will be constrained by the current zoom level. The default visible grid is 100 mils. The two visible grids are entirely independent from each other.

You can assign colors to each of the two visible grids from the Options-Layers dialog box. The Options-Preferences dialog box allows you to select line or dot grid types.

Measurement system

Advanced PCB supports both imperial (mils) and metric (mm) measurement units and dimensions. The unit of measure changes when you use the Options-Toggle Units command or press Q. When a metric snap grid value is designated (Current-Snap Grid command), Advanced PCB displays workspace coordinates and other dimensional information in millimeters (indicated mm on the status line). When you activate this option, all measurements and inputs can me made in mm rather than mils (.001 in). This allows accurate dimensioning of boards in millimeters or creation of new library components with metric pin spacings. Measurement units can be switched at any time by choosing a snap grid of the appropriate value.

Components and primitives

Your design is created by placing components and individual primitives (tracks, vias, text etc.) in the workspace. The total number of components and individual primitives in your design is limited only by the available memory of your system. Items which can be placed include:

Components

Components are any collection of primitives (tracks, pads, arcs, etc.) that have been selected as a group, then added to a component library. The size of a component is restricted only by the memory available to select and add primitives to a library file. Advanced PCB libraries can contain 500 components and any number of libraries (limited only by available memory) can be opened at the same time.

Both through-hole and SMD components can be placed on either side (Top layer or Bottom layer) of the board. The number of components is limited by memory only. The component designator can be up to 12 characters and can be placed anywhere relative to the component on either copper or overlay. The comment field, which is used to store the type of the component can be up to 32 characters and can be placed anywhere relative to the component on copper or overlay. Both comment and designator can be hidden from view and set to any size and font.

Arcs

Arcs are essentially circular curved track segments. They can be placed on any layer with any radius and width from 1 to 9999.999 mils wide. The angular resolution is 1 degree. Arcs can be placed using the Arc button, Place Arc commands or as tracks using the Place Track command (see the Options-Track Mode command). The Replace Arcs pass of the autorouter will convert 90 degree corners into arcs, wherever (Design Rules) clearances allow. Arcs are also used when generating polygon fills when the Place Polygon Plane command is used.

Fills

Rectangular fills are placed to indicate solid copper areas of any size, on any signal layer. When placed on Power plane, Solder or Paste mask layers, fills are used to designate voids.

Pads

Pads can be either multi-layer or placed on any individual layer. For example, for surface mount components and edge connectors single layer pads are placed on the Top and/or Bottom layers. Pads shapes include circular, rectangular, rounded rectangular or octagonal with X and Y size definable by the user from 1 to 9999.999 mils. Hole size can range from 0 (SMD) to 9999.999 mils. Pads can be identified with a 4 character designator. Pads that are part of component descriptions can be individually or globally edited after the components are place. On multi-layer pads, the Top layer, Mid layers and Bottom layer pad shape and size can be independently assigned, to define “ pad stacks.” Pad can be used individually, as free pads or they can be incorporated (with other primitives) into components.

Text strings

Text strings can be placed on any layer with any height from 0.05–9999.999 mils. Strings can placed using the Place Strings command.

Tracks

Tracks are placed on any layer using any width from 1 to 9999.999 mils wide. Tracks can be placed using the Track button, Place Track command or by the autorouter. Tracks are also used to generate polygon fills, when the Place Polygon plane command is used.

Vias

Vias are either multi-layer, blind or buried and can be any diameter from 1 to 9999.999 mils wide. Vias can be placed using the Via button, Place Vias command or by the autorouter. The hole size can be set from 1 to 9999.999 mils.

Polygon Planes

Polygons are special entities formed when you use the Place-Polygon Plane command. Although polygons consist of tracks, arcs and (optionally) area fills, polygons can be manipulated as a unit. Polygon vertices can be moved to a new location, and vertices can be added or deleted graphically. Polygon grids, fill elements and obstacles can also be redefined after placement. A special Edit-Change-Re-Pour Polygon command will regenerate a polygon around new obstacles, while providing the user a chance to define a new fill track width and grid rules. The Edit-Change-Convert

Selections To Fills command can be used to simplify the polygon fill geometry for more efficienct artwork generation.

Dimensions

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

Coordinates

Coordinate markers are similar to dimensions, consisting of text and track segments, automatically generated when you select a position in the workspace.

Fonts

The Advanced PCB system is delivered with three fonts that are specially designed to support vector plotting, including both pen plotters and Gerber plots: Default (plain); San Serif and Serif. Fonts for Free String placement are selected from the Current-Free Text Font command. Component text fonts are selected from the Current-Component Text dialog boxes.

All fonts, being line vector fonts can be plotted and photoplotted 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.

The default fonts supplied with the system are built into the software and cannot be unloaded.

The size of the text is controlled from the Current-Free Text Height command and Current-Component Text dialog boxes. Height can be set in either mils (.001 in) or in millimeters. Text can be placed on any layer. Once placed, Free Text strings can be moved or edited like other primitives. Component text can be moved independently of the component (Move String command). If the component is moved, component text will move relative to its current position.

➡ Strings in converted Protel Autotrax files will be converted to the Default font type. When performing a Design Rule Check, checks are made for collisions with text strings. The size of the string is calculated using the default font, even if the some other font is used in the string. This only affects the length of the string as the user controls the height and this is independent of the font.

Component libraries

A comprehensive library of 317 predefined through hole and SMD components is included with the standard PCB design system. This system also includes a complete set of commands for creating, editing and using library components. Custom libraries can be created and any number of component libraries can be opened at the same time, limited only by available memory. Each library can hold a maximum of 500 components.

The standard PFW.LIB (library) file is normally loaded automatically whenever Advanced PCB is opened providing the directory containing the library is in the current path when the application is launched.

Components are created on-the-fly, directly in the Advanced PCB workspace. A component is defined using selection, then written to a library using the New option in the Library-Components dialog box.

Component primitives generally include one or more pads (corresponding to component pins and numbered to reflect the actual pin numbers) plus track and/or arc segments on the overlay (or silkscreen) layer to define the component body. Once created, components can be un-grouped, then edited.

Pad types

Pads are organized into libraries, similar to components, with each Pad Type having a predefined size, shape, layer assignment, hole size and power/ground plane status. One Pad Type from the open library is always the "current" type, selected using the Current-Pad Type command (shortcut: c, p).

Pad definition and pad library maintenance is performed from the Library-Pad Types dialog box. Select the Library-Pad command to open the Pad Types dialog box, listing the currently available pad types. A maximum of 200 pad types can be included in a single pad library. Using this dialog box you can manipulate the list of currently available pad types. You can create new pad types, delete pad types, and load and save lists to and from files. The Edit command on the bottom right hand side of the pad types dialog box can be used to change current pad types. These changes can then be applied globally in the same way as editing pads on the printed circuit board.

Mouse and keyboard shortcuts

As you read through this guide or browse the *Reference*, you will notice several mouse and keyboard shortcuts that are used to speed-up or simplify frequently performed operations, commands and options. For example, pressing c, p allows you to choose a new Pad Type without having to go to the Current menu and choosing the Pad Type command. 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. Some keyboard commands provide the only practical way of performing an operation when you don't wish to move the mouse in the workspace such as setting a new grid or 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. Double-clicking may take slightly longer than using the Edit-Change command because the system must search the current coordinate position for each primitive type.

To move an item, hold down the CTRL key, click (left mouse) on the item you wish to move. If you wish to delete an item from the printed circuit board then hold down the DELETE key and click on the item you wish to delete.

Many shortcuts are part of the standard Windows interface, such as pressing ALT, F4 to close the application window. Other "hot keys" have been added that are specific to Advanced PCB. Some shortcuts that were part of the Protel Autotrax default key macro set have been retained in Advanced PCB, for the convenience of experienced Protel DOS system users.

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

There are also a number of third party Windows-compatible macro editors available, which allow users to define keystrokes.

Keyboard shortcuts

Two sets of keystroke shortcuts are available: standard Windows combinations of the ALT key and a command key (underscored in the menu). For example, press ALT, F to open the File menu. A second set of Protel default shortcuts augments these options, for example, to open the File menu, just press F once. Other shortcuts include:

ALT, F or F	File menu
ALT, E	Edit menu
ALT, L or L	Library menu
ALT, M or M	NetList menu
ALT, A or A	Auto menu
ALT, C or C	Current menu

ALT, O or O	Options menu
ALT, Z or Z	Zoom menu
ALT, I or I	Info menu
ALT, W or W	Window menu
ALT, H or H	Help menu
ALT+BACKSPACE	Undo
E	Edit-Change menu
U	Auto-Placement Tools menu
T	Edit-Toggle Selection menu
X	Edit-DeSelect menu
CTRL+A	Edit-Place Arc (Center)
CTRL+G	Current-Snap Grid
CTRL+H	Edit-Select-Physical Net
CTRL+L	Current-Layer
CTRL+M	Info-Measure Distance
CTRL+P	Library-Pad
CTRL+S	Current-Free Text-Height
CTRL+T	Current-Track (width)
CTRL+U	Un-delete one item
CTRL+V	Current Via Size
CTRL+Z	Zoom-Window
PGUP	Zoom-Expand
PGDN	Zoom-Contract
CTRL+PGUP/PGDN	Zoom maximum / minimum
SHIFT+PGUP/PGDN	Zoom at 0.1 step rate
HOME	Zoom-Pan
END	Zoom-Redraw
CTRL+HOME	Jump Absolute Origin
CTRL+END	Jump Relative Origin
CTRL+INS	Edit-Copy
CTRL+DEL	Edit-Clear
SHIFT+INS	Edit-Paste
SHIFT+DEL	Edit-Cut
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

SPACEBAR	Toggle track placement modes; rotates item during selection or move (uses preset step value in Options-Preference dialog box); aborts screen redraw.
TAB	Opens current menu to assign new value when placing arcs, pads, strings, tracks or vias.
SHIFT	When placing or moving items, pans at 5x normal rate.

Preferences and defaults

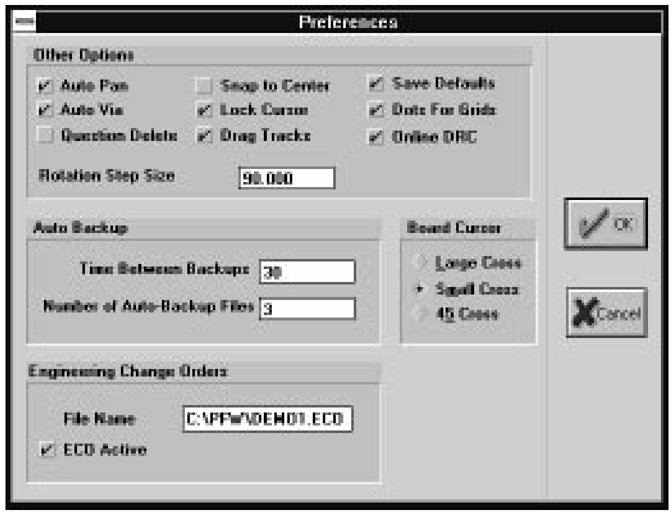
Advanced PCB stores many user settings, such as printer/ plotter setups, display colors, grid settings and many other options in a special file called PFW.INI which is automatically added to the Windows directory, the first time that you Exit the application. Thereafter, PFW.INI is updated each time you exit the application. When you start Advanced PCB, the program looks for this file and your preferred settings are restored.

Other settings belong to a specific workfile, such as the layers, grids and design rules used when the PCB file was created. When you re-load any PCB file, these settings are restored. If you select File-New, the normal system defaults or (where applicable) user preferences are restored.

See the *Advanced PCB Reference* for a listing of saved defaults and preferences.

Managing preferences

You can restore all original (system) default settings by deleting the file PFW.INI from your Windows directory. You can keep more than one .INI file (with different settings for different types of projects) by temporarily re-naming the file, or by moving it into another directory. For example, you might wish to create a directory that includes all documents for a project with a special .INI file.



The Preferences dialog box sets-up many Advanced PCB features.

Options-Preferences command

The Options-Preferences command (shortcut: o, p) sets-up a number of environmental conditions for Advanced PCB.

For more information about setting up Advanced PCB system options, see the Options-Preferences command, beginning on page 179 of the *Advanced PCB Reference*.

Display options

Choose Options-Display to set the display characteristics of six primitive types: arcs, fills, pads, strings, tracks and vias.

Display Options dialog box

There are three options for each: Final, Draft or Hidden. If you choose Final then the item will be drawn in solid color. Draft will display the outline of the item. Hidden will remove the item from the display, Although it will still be included in the PCB file.

When Hidden is chosen, the item cannot be selected, but will be included when generating artwork. Draft or Final mode are available for pen plotting and printing. Draft mode is not available when photoplotting.

The Draft Track Threshold option sets a minimum track width for fully rendered track outlines (when Draft display mode is selected for tracks). Under the threshold size, draft tracks will be rendered as single pixel lines, rather than as full outlines. This option is useful for simplifying dense displays and will also make screen redraw faster.

The two Pad Information options allow you to either display or hide the pad designator (Show Numbers) and associated net name (Show Nets).

For more information about setting up Advanced PCB display options, see the Options-Display command, beginning on page 183 of the *Advanced PCB Reference*.

Managing files

One of the key advantages of the Windows environment is the excellent support provided for file management. Whether you use the File Manager or a third party Windows file handling utility, Windows makes it easy for you to find, open, close, hide, move or delete files. The term files refers generally to all the individually labeled entities that can be stored on hard or floppy disks. These can include the main program files, auxiliary files, such as device drivers or libraries and user files.

Under Windows, program files are generally referred to as the “application.” User files are usually called “documents.” For example, when you start Advanced PCB, you are opening the application window that includes the menu bar and Advanced PCB title bar. Document windows display PCB file(s) and have the filename in the title bar.

Advanced PCB file operations are performed similarly to other Windows applications. All of the standard file commands are found under the File menu. Some file operations use a specific extension to identify the type of file involved. For example, the File-Open command looks for files with the default extension .PCB that is reserved for workfiles. When this extension is displayed in the Filename box, only files with the extension .PCB are displayed for the current directory. If the extension is changed, Advanced PCB will list files that match the new value. The wildcard “?” is also supported. If you change the Filename to include the extension “.???,” all files in the current directory will be listed in the Files box.

Changing the current drive or directory (path)

To change the drive or directory (path):

1. Choose File-Open to open the Load PCB Filename dialog box.
2. In the Filename box, type the name of the drive or directory or both followed by *.* .

You can also click in the Directories window to select from the available drives/directories. Choose [..] to go up one directory level.

3. Click OK or press ENTER.
4. To close the dialog box, click Cancel.

To change to a subdirectory of the current directory:

1. Choose File-Open.
2. In the Directories box, select a subdirectory.
3. Click OK or press ENTER.
4. To close the dialog box, click Cancel.

To change to the parent directory of the current directory:

1. Choose File-Open.
2. In the Directories box, select [..].
3. Click OK or press ENTER.
4. To close the dialog box, click Cancel.

To close the active PCB window:

1. Choose File-Close.

If there are no changes, the PCB window is closed immediately. If there are unsaved changes, Advanced PCB asks if you want to save changes.

2. Click Yes to save changes, No to discard changes, or Cancel to cancel the command. If the file hasn't been saved before, the File-Save As dialog box is displayed.

3. In the Save As box, type a name for the file or use the proposed name.

4. Click OK or press ENTER.

If the specified file already exists then you will be asked to confirm overwrite

5. Click Yes to continue, choose No to enter another name, or click Cancel to abort the command and return to the PCB window.

To view a list of files in a directory:

1. Choose File-Open.

2. In the Filename box, type a file specification.

It can include drive, directory, and the wildcard character "?" (to match a single character) and "*" (to match any number of characters). You must include a combination of three characters (including wildcards) to list the files.

3. Click OK. The Files box displays the names of all files matching the file specification you type.

4. When you have finished viewing the list, click Cancel.

To View all PCB files in a directory:

1. Choose File-Open.

2. In the Filename box, type *.PCB (asterisk, period, PCB).

3. Click OK or press ENTER.

4. When you have finished viewing the list, click Cancel.

To View all files in a directory:

1. Choose File-Open.
2. In the Filename box, type the name of the drive or directory or both followed by *.*.
3. Click OK or press ENTER.
4. To close the dialog box, click Cancel.

Saving files

There are three Save options on the file menu:

Save

The standard file save command saves the PCB file in the active window. The file is always saved in the Advanced PCB binary format. The reserved extension .PCB is always appended to the filename (shortcut: f, s).

Save As

To save the active file with a new name or format use the File-Save As command (shortcut: f, a). The Save PCB File As dialog box opens when you choose Save As, or when you choose Save the first time for a new file. The reserved extension .PCB is used, regardless of format. There are three Save As file format options:

Protel Binary

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

Protel Text

Protel's published (ASCII) text format for PCB files. The format of this file is described in the appendices at the back of this manual. ASCII files are less efficient than the binary version, but allow the user direct access to the design database for translation into other formats or other manipulation. If you open a Protel Text format file in Advanced PCB, it will be automatically saved in the default binary format.

Autotrax

This format allows you to export a Advanced PCB file back into Protel Autotrax format, which is very similar to the Protel Text format described above. When you load an Autotrax file into Advanced PCB, it will be automatically saved in the default Advanced PCB binary file format.

➡ If you save your files in Autotrax format you should be aware that you may lose some information. For example, Advanced PCB supports many more layers than Autotrax – any information on these extra layers will not be included in the Autotrax format files. Advanced PCB also includes special attributes not supported by Autotrax, such as 1 degree arc resolution and blind/buried vias. Arcs will be truncated to 90 degrees in Autotrax and all vias will be converted to multi-layer.

For more information see page 12 in the *Advanced PCB Reference*.

Save All

The Save All command (shortcut: f, l) can be used to save all of the printed circuit boards in all currently opened windows. All files will be saved in binary format.

Importing PADS® files

Advanced PCB directly loads PADS-PCB and PADS2000 (.ASC) files. To load these files, simply:

1. Choose File-Open.

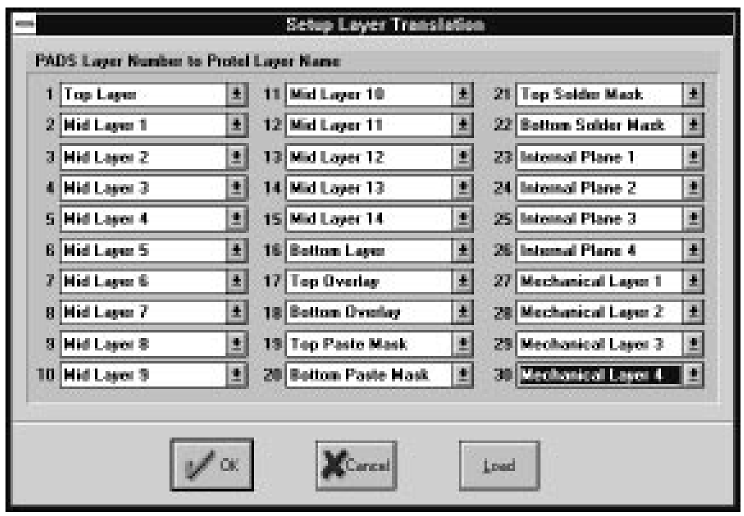
The Load PCB File dialog box opens.

2. In the Filename box type *.ASC to list PADS-PCB files in the current directory.

3. Select the required file by double-clicking on the filename, or by selecting the file and clicking Open.

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 the user 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 Multilayer 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.

Importing Tango-PCB® files

Tango PCB Series II files will be transparently loaded 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 and Protel-PCB version 3.12 (as produced by Protel) files directly. These files can also be loaded if they were previously converted to Autotrax format using the Protel Autotrax PCB3CON utility.

File integrity checking

Advanced PCB performs a file integrity check when loading all ASCII text format files, irrespective of the source. File validation is also performed the first time you load Advanced PCB binary files that were created by earlier versions. This check screens the file for any out-of-range errors in the file data and will attempt to repair any problems that prevent the file from loading, if possible. File validation is important because files from other sources (i.e. files from another design system) may have values that fall outside legal Protel for Windows ranges. It is also possible for the user to introduce out-of-range values when editing ASCII (text) file formats directly.

For example, the validation routine checks for primitives with coordinates outside of the range 0-99999.999 mils and moves any offending item into the legal workspace. Similarly, out of range layer numbers are re-assigned to a legal value. The validation routine will check the netlist and connection data in the file for errors.

If a problem is encountered, a warning message will be displayed and the netlist will be stripped from the PCB file and be written into a separate .NET file. In the file, the string **ERROR** will replace any bad node.

Any discovered problems will be reported as the file is loaded. A separate report file <filename>.VLD will be generated with a detailed listing the problems encountered in the loaded file. For more information about ASCII file data, see the PCB File format description at the end of the *Advanced PCB Reference*.

Design topics

This section includes a number of articles that address both the general process of board layout in Advanced PCB and provide a detailed look at some of the specialized tools and features provided by the software. In many cases, step-by-step procedures are described in detail in these articles. Specific information about each command and dialog box option can be found in the *Advanced PCB Reference*.

Details of specific procedures, such as setting-up display preferences, working with netlists, autorouting or generating PCB artwork will be found in other articles within these Design Topics. A summary of all commands will be found in the Command Reference.

Defining a board

One of the first operations performed, would normally be to define the physical outline of the PCB. There are several ways that this can be accomplished in the Advanced PCB system.

If you are using auto component placement and autorouting (from a schematic netlist), you must use the special Keep Outs layer to define the perimeter of the usable board area and any “no go” zones where components and/or tracks are to be excluded when these automated operations are performed.

The Mechanical Layers are also provided for the purpose of documenting the physical details of the board. For example, you may wish to start by placing dimension lines or coordinate markers on one or more of these layers. See the Place Dimension and Place Coordinate String commands.

In either of these cases, you place the same tracks, fills and arcs on these layers that you use on signal layers to define the signal/current carrying portions of the board.

PCB primitives

Using tool buttons to place primitives

The Tool bar provides direct placement of primitives: arcs, components, fills, pads, strings, tracks and vias. For example, pressing the Arc tool button is equivalent to choosing the Edit-Place-Arc command. Other Place command options are available under the Edit-Place command: polygon planes, coordinate strings, dimensions or arrays of primitives.

The Tool bar can be displayed or hidden using the Options-Tool Bar command (shortcut: o, o).

Step-by-step procedure for placing, moving and changing all PCB primitives are found in the *Advanced PCB Reference*.

About arcs

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 the mechanical layer to indicate the board outline, holes, etc. Advanced PCB arc resolution is 0.001 degree, which allows very large arcs to be placed with extreme angular precision.

Arcs can also be placed on signal layers as tracks. “Track” arcs are generated “on-the-fly” while placing tracks if the 90 Arc/Line is selected under the Options-Track Mode command (shortcut: o, t, r).

Advanced PCB allows you to define arcs of any angle (0.001 degree increments), using the same line widths used for tracks (0.001–9999.999 mils). The current default arc line width is set using the Current-Track command (shortcut: c, t).

Place-Arc (Center) command

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

1. Choose Edit-Place-Arc (Center) (shortcut: p, a) or the Arc tool button.

The prompt “Select Arc Center” is displayed on the Status line. 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 where the center of the arc is to be placed and press ENTER or LEFT MOUSE once.

The Status line will now display the Radius and Start Angle of the arc. As you move the mouse or use the cursor key array, a highlighted arc will be displayed.

3. Position the cursor to establish the desired radius and starting point, then press ENTER or LEFT MOUSE once.

The Status line now includes an End Angle value. Note that the possible location of the arc center and start point are constrained by the current snap grid.

4. Position the cursor to define the end point of the arc, then press ENTER or LEFT MOUSE again.

If you are drawing a 360 degree arc, click without moving the cursor (Start and End Angle values will be the same). As you define the end point, note that the radius does not change, regardless of the distance of the cursor from the arc center.

5. Start a new arc or press ESC or RIGHT MOUSE to exit this command.

Place Arc (Edge) command

To place an arc on the current layer using the arc edge to set the position, Choose Edit-Place-Arc (Edge) (shortcut: p, r). This command works identically to the Edit-Place-Arc (Center) command.

➡ When using the Arc tool button, the default placement method (Center or Edge) is determined by the option last used under the Edit-Place-Arc command.

For more information see page 90 of the *Advanced PCB Reference*.

About components

The Library-Components command (shortcut: l, c) is used to open the Browse Libraries dialog box. This feature allows you to open any required library files preparatory to placing components and to perform library maintenance tasks.

Components are always initially placed on the top side of the PCB. The component outline is displayed on the Top overlay (or silkscreen) layer. Component text (designator or comment) can be assigned to either the overlay or "copper" layers. Top and Bottom Overlay (or silkscreen) layers can be plotted separately for printing onto the fabricated board surfaces.

While each component is stored as a single entity, it is actually a collection (or "group") of primitives (normally tracks, pads, arcs and text). These component primitives belong to various layers – initially assigned when the component is created. Components can be swapped from the Top (component side) to the Bottom (solder side) layer using the Change Component command. When components are moved to the Bottom layer, Top Overlay primitives are automatically swapped to the Bottom Overlay and the component orientation is flipped along the X axis.

It is a simple matter to create new components by selecting a group of items which can then be added to the current library.

Placing components

To place a component in the workspace:

1. Choose Place Component (shortcut: press p, c or click the Component tool button).

The Name in Library dialog box opens, displaying a “?” if no components have been previously placed. If components have been placed during this session, the package description of the last placed component will be displayed.

2. If you know the component package description (DIP14, for example), type it into the window and Click OK or press ENTER. If you are not sure of the package description, type *?* to list the components in all currently opened libraries.
3. Move the selection bar through the available components using the mouse or cursor keys.
4. Press ENTER or LEFT MOUSE when the correct component name is highlighted.

The Component Designator dialog box opens, displaying a “?” if no components of this pattern have been previously placed. If you just type *?* then the system will use the lowest letter (starting at A) with the lowest number (starting at 1). If components have been previously placed, the next available designator will be displayed. The File-Re-Annotate command can be used to re-designate all board components based upon component placement position, at any time.

5. Press ENTER or click OK to accept the default designator or type in a new designator then click OK or press ENTER.

The Comment dialog box opens. Include a comment, such as a part number or name, if desired and click OK or press ENTER.

Components will always be assigned a designator. Comments – such as a component value or part number, are optional. Designators are limited to 8 alphanumeric characters – spaces are not allowed. Comments can be up to 32 alphanumeric characters, including spaces.

A highlighted component, with the cursor at the center of pin 1 (or A1), is displayed in the workspace. The prompt "Moving Component (designator)" is displayed on the Status line.

When placing an SMD component, the current layer should be either the Top or Bottom layer. Press + or – to toggle through all active layers. All conventional components will be placed on the top of the board unless the Bottom layer is the current layer.

6. Move the component into the desired position.

You can change Zoom and Snap Grid settings to facilitate this task. Press SPACEBAR to rotate the component 90 degrees counter-clockwise. Press x or y to flip the component along either axis. You can free rotate the component in 0.001 degree increments, after placement, using the Move Rotate Selection command.

7. Press ENTER or LEFT MOUSE to complete the placement.

The component will be redrawn with the highlight removed. The Top Overlay layer color will be used to display the outline and text and the appropriate layer color to display the pads – assuming that these layers are activated.

The Name In Library dialog box re-opens, displaying the component type which was just placed.

8. Click OK or press ENTER or LEFT MOUSE to place another component.

If you place another component, you will note that the Designator has automatically incremented. You can accept this default value or type in a new designator.

9. Type a new component type or a *?* to browse the available component types in the current libraries. Click CANCEL or press ESC to exit component placement.

You can also place components while browsing the current libraries by clicking on the Place button.

Component text defaults

Advanced PCB allows you to pre-define the defaults for component text independently of the defaults used for free text strings. These options include rules for positioning component designator and comment text relative the component footprint on the (silkscreen) Overlay layers.

➡ Component text defaults are defined prior to placing components – either manually or automatically using the Auto Place features. Changes to component text after placement are performed using the Move String and Change String commands (and shortcuts).

To set-up component text defaults choose Current-Component Text (shortcut: press c, c) and choose either the Comments (c) or Designators (d) option. Options include:

Layer

Places component text on either the Top (component) side or Bottom (solder side) layer (according to the component layer status), or on the (silkscreen) Overlay. Silkscreen Overlay is the default setting.

Rotation

Positions component text horizontally or vertically. Horizontal is the default setting.

Height

Assigns a default text size in mils (.001 in) or mm. Default setting is 60 mils.

Font

Assigns a default font, either Default, Sans Serif or Serif.

Horizontal Alignment
Vertical Alignment

These two sets of options define the default position of the text.

Show Text on PCB

Displays or hides the text. Hidden text is retained in the PCB data and can be toggled between show/hide at any time using the Change Component options.

➡ Default settings for designator strings and comment strings are independent of one another – different defaults can be defined for each string type.

After placement, you can edit the layer, size, font and display/hide status of component text. You cannot globally post-edit the position of designators and comments on the components.

For more information see page 93 of the *Advanced PCB Reference*.

Creating a new component

Each component pattern (or footprint) is made up of a collection of standard PCB primitives: tracks, arcs, pads, fills etc. The process of creating a component is a simple process of arranging these primitives, selecting the arrangement as a group, then adding the selection to an existing component library.

Two basic component creation strategies are available. You can start from "scratch" placing and numbering each pad, creating an outline on the overlay, etc., then add the component to a library. The other approach is to first place an existing component in the workspace, then convert the component back into primitives (Library-Un-group command). You can then edit the primitives, and finally add the selection into a library. Global editing options can be productively applied to either method, particularly if you are creating a number of similar components. A library can also be automatically generated that includes *every* component placed in the current document window.

To add a new component to an existing library:

First place all the items you want in the component using the Top or Bottom overlay, Top or Bottom layer for surface mount pads or Multi layer for through-hole pads. Be sure to add pin designators to all pads for later reference. When your component is designed, use Edit-Select-Inside Area (shortcut: s, i) to group the primitives inside a selection rectangle.

➡ Before placing primitives for a new component, choose a snap grid that matches the component pin pitch. For example, the default 25 mils grid will work fine with standard through hole components with 100 mils pin spacings. Choose Edit-De-Select-All (shortcut: x, a) to clear the current selection.

1. Choose Library-Components (shortcut: l, c).

The Library Components dialog box will open.

2. Make sure the correct library is selected under the Libraries menu, then click New.

You will be prompted to type a New Library Component Name and to provide a reference point by clicking anywhere in the workspace. This reference point will be on the snap grid and will typically be the center of pin 1.

3. Type a name of up to 10 characters, then click OK. No spaces are allowed in component names.

If the designated name is already used, the message “Library Name is Already Used” will be displayed. Click OK, then provide a unique name.

4. Click OK at the bottom of the Library Components dialog box.

The next time you place a component the added item will be included in the Components menu.

Adding all placed components to a library

The Add All Placed Components To Library option allows you to add all placed components in the current design to a component library. This supports users who are importing files from PADS or Tango and wish to add components created in these systems to an Advanced PCB library. To use this feature:

1. Open the PCB file which contains the components you wish to add to the component library;

If this is the first time an ASCII, PADS-PCB or Tango-PCB file is being loaded, the conversion will take a few moments. When saved, the file will be re-formatted in the Advanced PCB binary format.

2. Choose the Library-Components command.
3. In the Browse Libraries dialog box, choose Add All Components to Library.

All placed components will be added to the current library. Component pattern fields will be truncated to the first 12 letters, if the pattern fields are longer. This can occur if the components originated in PADS files.

➡ If any of the new components names match components in the current library, these new components will not be added. You must first remove all duplicates from the library.

To check for duplicate names, use the Report option to list the current library components in a file named PFW.REP. You can then create a new library, import all the components and generate another .REP file (making sure you give it another, unique name). You can then check the two report file listings for duplicate names, etc.

Report components

The Report button will generate a list (default name PFW.REP) of components in all open component libraries in the Libraries list box (Browse Libraries dialog box).

Library-Un-Group

This command is used to convert a placed component back into its constituent primitive parts. When you choose the command (shortcut: l, u) 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 display – these text fields are assigned only when a component is displayed.

The primitives in the component can now be edited as free primitives. This command is particularly useful if you wish to design a new component, based on an existing pattern.

➡ This command has no effect on the component pattern stored in the library – only on the individual instance of the component “placed” in the document window.

About fills

Fills (or *area fills*) are solid copper rectangles placed on signal layers to provide shielding or to carry large current supplies. Fills of varying size can be combined to cover irregularly shaped areas or fills can be combined with track or arc segments and be recognized as electrically connected when using features like Select Physical Net or Select Physical Connection or when running the design rule check (DRC) feature. Fills are also placed on non-electrical layers. For example, placing fills on the Keep Out layer is used to designate “no-go” areas for both autorouting and auto component placement. Fills can be moved or re-sized after placement, using the Edit-Move-Fill command and shortcuts.

For more information see page 97 of the *Advanced PCB Reference*.

About pads

Advanced PCB is delivered with a standard pad library that includes a wide variety of both through hole multi-layer pads and SMD pads. The Pad Types options allow you to manipulate pad libraries and to edit the attributes of existing pads.

Only one pad library can be loaded at a time but custom pad libraries can be created for different design tasks – such as a library of SMD pads, for use in designing surface mount boards. The number of possible pad types in the current library is limited only by available memory.

Each pad type is stored as a library item, with predefined attributes: name, size, hole size, layer assignment and shape. Storing attributes allows pads to be placed without having to specify this information at each placement. Each pad library, identified by the extension .PAD, is a list of these predefined pad types – each identified by name (Round50, etc.). The current pad library is stored in memory.

➡ When you start Advanced PCB, the application attempts to load a pad library called PFW.PAD. This contains a list of default pad types. To change the default list, temporarily re-name the standard pad file and use the PFW.PAD name for your preferred default library.

Managing pads and pad libraries

Library-Pad Types command (shortcut: l, p) options are used to set-up, load, edit, create and modify pad libraries, including changes to the attributes of individual library pads. When you choose Library-Pad Types a list of all currently loaded pad types is displayed.

Type the pad library filename or use the default mask, changing directories to display all .PAD files. Pad libraries always use the reserved extension .PAD.

➡ If you have made any changes to current library, save it before loading another or your changes will be lost.

Delete

Click to remove the selected pad type from the current library. The message “Confirm Delete Item” will be displayed. Click YES (or press y) to delete the pad; NO (or press n) to abort the deletion and return to the Pad Types dialog box.

Save

To make the changes in the current library permanent, click Save. The specified filename will be forced to the reserved extension PAD. If the file already exists you will be given the option to overwrite the existing name. To create a new pad library, use Save to create a duplicate of the current library, then use Clear to remove all predefined pads, if preferred.

Plotting multi-layer pads

Multi-layer pads will be removed from Mid layer (1–14) prints/plots if the pad is unconnected on the layer being printed.

Placing pads

Free pads (pads that are not grouped in a library component) can be placed manually anywhere in your design. Through-hole pads (and vias) are Multi layer objects which occupy each copper (signal) layer of the PCB. Multi-layer pad size and shape attributes can be independently assigned to the Top layer, Mid 1–14 and Bottom layer, allowing the user to create “pad stacks.” Single layer pads, like pads in SMD components (or edge connectors), can be placed on any layer. You can place a through-hole (multi-layer) pad or a via without regard to the current layer selection. The default pad type can be changed from the Current menu.

If placing SMD 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 position defines the pad center.

Pads can be labeled with a designator (usually representing a component pin number) of up to four alphanumeric characters (spaces are not allowed). The designator of placed free pads is initially empty. The ability to leave pads un-numbered extends the flexibility of the global pad editing options – for example, you can constrain global changes to un-designated pads. Use the Edit-Change-Pad command to change the designator.

Pad designators can also be auto-incremented during placement. Incrementing can be numeric (1, 2, 3), alphabetic (A, B, C) or a combination of alpha and numeric (A1, A2, or 1A, 1B, or A1, B1 or 1A, 2A, etc.).

You can also define the step value for incrementing. For example, an increment of “B” will generate designators: A (initial pad name), C, E etc. An increment of “3” will generate: 1, 4, 7, etc. Any value within the legal range (A–ZZZZ or 0–9999.999) is allowed (pad names can also be empty or null (0).

When incrementing reaches a limit, the system intelligently wraps to the next legal value or combination of characters to form a legal value (“E” would yield: A, E, J, O, T, Y, AD, etc.). Incrementing always works with like — alphas for alphas, numbers for numbers.

To automatically place multiple pads with incremental numbers (as when building a new component pattern), see the Edit-Place-Array command.

For more information see page 98 of the *Advanced PCB Reference*.

About strings

The Place String command is used to place short "free" text strings (up to 32 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. You can define the text height in .001 mils increments from 0.05 mils up to 9999.999 mils. The width of text character is automatically proportioned to height. The line width used to draw the text is also automatically proportioned to the text height.

Component text, (designators and comments generated when placing components) have the same font options and height range as free text and can be moved (Move String) and edited (Change Component). Free text can be placed on any layer. Component text is automatically assigned to the Top or Bottom Overlay layers or Top and Bottom ("component" or "solder-side") layers. The default free text height and/or font can be changed from Current menu, prior to placing text. These parameters can be edited after placement by using the Change String command.

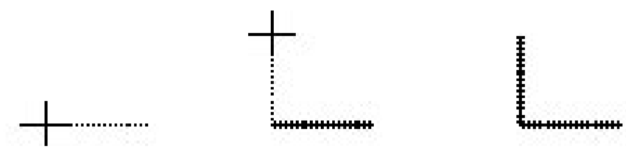
For more information see page 99 of the *Advanced PCB Reference*.

Special strings can be placed, which are interpreted when printing to automatically generate legends on artwork layers. A listing of these special strings is provided on page 233 of the *Advanced PCB Reference*.

About tracks

Track placement in Advanced PCB works differently from draw/paint applications where you "click-and-drag" to "stroke" a line. A single track can have many segments. You can precisely "mark" the starting and break (corner) and end points, providing the placement control required for accurate PCB layout.

Tracks (also called *traces*) may be routed on the Top layer, any of fourteen Mid layers or on the Bottom layer also called the solder side. Where tracks on paired layers meet vias are placed to carry current between these layers. These vias can be Multi-layer (passing from the Top layer to the Bottom layer through all other layers) or confined to any fabricated layer pairs as blind or buried vias (see Placing vias, below).



Tracks are placed with a series of clicks which define each segment.

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

The Top (component side) layer.

Any of fourteen Mid-layers (number Mid Layer 1, etc.).

The Bottom (solder side) layer.

These layers are known collectively as the signal layers.

“Non-electrical” tracks can also be placed on:

The two Silkscreen overlays (normally used to draw component package outlines).

Any of four Internal (power) Plane layers (to create voids in these solid copper planes).

The Keep Out layer to define the board perimeter for autorouting and auto component placement:

Any of four Mechanical layers (as alignment or trim marks, or other “mechanical” details).

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. If you move placed objects, you will notice that they snap from grid point – to grid point as they are dragged. If you change the grid, placed items can be left temporarily off-grid, until moved to a new position.

Placing tracks

Tracks can be drawn in line widths from 0.001–9999.999 mils. The Current-Track command allows you to choose the default width of the track.

To place tracks on the current layer:

1. Choose Edit-Place-Track (shortcut: p, t or click the Track tool button).

The prompt “Select Track Start Point” is displayed on the Status line. You can toggle to the desired “active” signal layer by pressing * or toggle through all active layers using the + and – keys. The selected layer will become active, regardless of whether or not it was previously activated in the Option Layers dialog box. Press TAB to change the default track width during placement.

2. Click LEFT MOUSE (or press ENTER) once to define a start point for the track.

The prompt “Place Track” is displayed on the Status line. As you move the cursor, the track length will be displayed to the right of the prompt.

3. Drag the highlighted track segment in any direction. Click LEFT MOUSE (or press ENTER) to end this first segment of the track – which removes the highlight.

Note that the prompt “Place Track“ is still displayed on the Status line and that the Length display re-sets to “0.”

4. Move the cursor to continue with a new highlighted 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, you can press BACKSPACE to remove the last track segment. You can also press ESC or RIGHT MOUSE to "cancel" the current segment currently being placed.

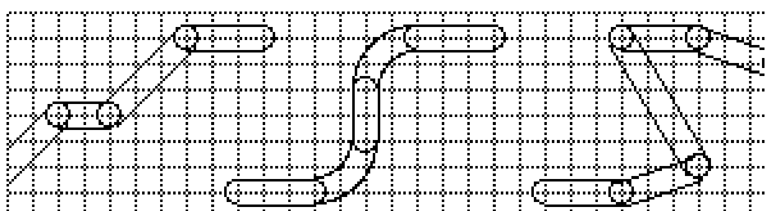
5. Click LEFT MOUSE again to end a series of connected tracks.

Note that "Place Track" is still displayed on the Status line. 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 the Edit-Place-Track command again.

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

Track placement mode

Advanced PCB provides four track placement modes (Options-Track Mode). Options include:



Orthogonal (with 45s), 90/Curved and Any Angle placement (Draft display mode)

Any Angle

Allows track to be placed at any angle.

90/90 Line

Constrains track placement to horizontal or vertical orientation.

45/90 Line

Constrains track placement to 0, 45, 90, 135, 180, 225, 270 or 315 degree orientation.

90 Arc/Line

Constrains track placement to horizontal, vertical or 90 degree arc orientation.

The 90/45 track segment option is generally referred to as “orthogonal” track placement. Curved tracks can also be automatically added to a design using Advanced Route.

➡ Toggle through the Track Mode options, while placing tracks, by pressing SPACEBAR.

For more information see page 101 of the *Advanced PCB Reference*.

About 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.

Advanced PCB Vias can be Multi-layer (passing from the Top layer to the Bottom layer through all other layers) or confined to any two (fabricated) layer pairs – known as blind or buried vias. Possible via layer assignments include:

Top layer - to - Bottom layer (multilayer)
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)

➡ Because managing the layer pairing of blind/buried vias can be a complex process, most designers will allow the layer assignment to be handled automatically by the autorouter or, when routing tracks manually, by using the Auto Via feature (enabled under the Options-Preferences command).

However, it is possible to edit the layer assignments (and all other attributes) of vias manually, after placement, if required. Layer pair colors are displayed inside the hole of all blind/buried vias. The default via diameter and via hole diameter can be set using the Current-Via command.

For more information see page 103 of the *Advanced PCB Reference*.

About polygon planes

Polygon planes (or copper pours) are similar to area fills, except that they can fill irregularly shaped areas of a board and have the additional feature of being assignable to specified nets. These planes can be either solid (copper) areas or, if preferred, a cross-hatch "lattice" made of user-definable track width and spacing.

When placed in occupied board space, polygon planes "pour" copper around any tracks, pads, vias, fills or text while maintaining predefined clearance between the fill and placed items. If you are working with a netlist-based layout, the plane will automatically "connect" with any adjacent component pads on the specified net.

➡ Solid or lattice polygon? The rules used to generate the polygon plane are derived from the current Router Default Variables: Routing Grid and Track Width.

If the track and grid settings are the same, a solid pour is generated. If the grid is greater than the track size, a lattice will be generated. To set-up these parameters, choose Auto-Setup Auto Route (shortcut: a, t)

Placing polygon planes

To place an polygon plane on the Top or Bottom layer:

1. Choose Edit-Place-Polygon plane (shortcut: p, e).

The message “Select Start Point On Polygon” will be displayed on the Status line.

2. Click at the starting point of the polygon.

“Line Length: 0” is displayed on the Status line. As you move the cursor, the perimeter of the plane is indicated by a highlighted line and the length of each segment is updated on the status line.

3. Click at each vertex of the polygon until the area of the polygon plane is defined.
4. Press ESC or RIGHT MOUSE to “close” the perimeter of the plane from the first vertex to the current cursor position.

Any remaining opening in the polygon will automatically close and a Net Name dialog box opens allowing you to assign the polygon to any net that is attached to the current workfile.

For example, if you specify he ground net, all pins (within the polygon) that are assigned to the ground net will be physically connected to the plane. You can use this facility to generate true split power planes. Just place “external” planes on any un-used mid-layers. All pins on the specified net will be connected to the plane on the mid-layer.

This process can be repeated on any un-used mid-layers – which can then be plotted like any other signal layer. If you are using a printer, pen-plotter or vector photoplotter, it will take a very long time to plot these fills, which will be generated as positive rather than negative artwork. Standard Internal (solid copper) plane layers are plotted in the negative, which is much more efficient.

5. Specify a net to be assigned to the plane or press ESC if no net assignment is required. Click OK or press ENTER or LEFT MOUSE to select from the loaded nets.

"Preparing Data" will be displayed momentarily on the Status line, and the plane will be generated. This may take a few-moments, depending upon the complexity, size of the plane and number of arcs enclosed within the polygon, etc. The wrap-around clearance is determined by the Clearance setting selected from the Netlist-Design Rules dialog box.

Once generated, polygon plane tracks can be individually or globally edited, like other tracks. The Edit-Select command options can be used to move or delete polygon planes. The Edit-Change-Convert Selections To Fills command can be used to convert polygon fill tracks (once selected) into rectangular area fills, which make the file smaller and which plot more efficiently.

For more information see pages 80 and 104 of the *Advanced PCB Reference*.

Outline Selected Items command

This command places an outline of continuous track and arc segments around any selected primitives on a signal layer. This feature uses the current track size, snap grid and Netlist-Clearances settings to define the outline. Outlines can be places for a variety of purposes, such as shielding individual clock lines by tying the outline to a ground source. Outlines can be placed over non-selected items on the same layer, and must always be checked for clearance violations.

About coordinates

Coordinate strings are special position markers that can be placed on any layer. These markers include a point marker (small cross of two tracks) and the x, y coordinates of the position. When you place coordinate strings, the default font type and height are used (Current-Free Text command).

Coordinate units, imperial (mils) or metric, are determined by the current Snap grid setting. The current Free Text height sets the coordinate string size and the current track width sets the marker track width. The size of the marker is one snap grid increment from the coordinate. Once placed, the marker and coordinate becomes track and text primitives which can be edited either locally or globally.

For more information see page 107 of the *Advanced PCB Reference*.

About dimensions

One of the first tasks many designers will want to perform, when starting a design, will be to layout dimensional details for the board. Advanced PCB provides a convenient auto-dimensioning feature to make the process both highly-accurate and easy.

The default string font and height (Current-Free Text command) is used for dimensions and the size of the arrow is 1 x 2 snap grid points. The thickness of the track used to make the line and arrow points is the current track width. Imperial or metric units will be calculated, depending upon the current Snap grid setting. Once placed the dimension becomes track and text primitives which can be either edited locally or globally.

For more information see page 108 of the *Advanced PCB Reference*.

About arrays

When you use the Edit-Cut (or Copy) command, you are placing a copy of the current selection in the clipboard. The Place Array command provides a powerful way to place multiple duplicates of any selection into the workspace. To use this feature:

1. Select the item(s) that you wish to repeat place as an array.

2. Choose Edit-Cut if you wish to clear the selection from the workspace prior to placing the array, or choose Edit-Copy if you wish to retain the selection in the workspace.

You will always be prompted “Select Reference Point” when cutting or copying to the clipboard. This allows you to pre-designate the cursor position when pasting the selection back into the workspace from the clipboard.

➡ It is recommended that you use the cursor keys (not the mouse) when selecting a reference point. This ensures that the reference point and the selection are on the same snap grid. If you designate an off-grid reference point, you will be unable to paste the selection onto the snap grid.

3. Being careful to keep the cursor on the snap grid, press ENTER or LEFT MOUSE to designate a reference point.

The reference point can be at any position relative the selection and will be used in positioning the items in your array.

4. Choose Edit-Place-Array (shortcut: p, y)

The Array dialog box options define the array. Array parameters include:

Placement Variables

Item Count

The number of repeat placements to be performed. For example, typing *4* will place 4 of the current selection, when Place Array is used.

Text Increment

This option is used for designators on pads and components. Setting this to 1 (default) will assign array designators in-series, for example U1, U2, U3 and so on. This feature follows the same rules as the automatic designator incrementing when placing components (or selections that contain components).

Array Type

Circular

Repeat placements are made in a circular array using the rotation and spacing values specified under Circular Arrays are by the full Advanced PCB arc resolution of 0.001 degrees.

Linear

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

Circular Array

Rotate Item to Match

When this option is enabled, the array items are rotated as they are placed.

Spacing

Specifies the angular spacing of each placed item.

Linear Array

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

5. Enter the desired values and click Place Array. Click OK to store the current parameters for later use or CANCEL to return the parameters to their previous settings.

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.

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

If placing a circular array you will be prompted “Select Starting Point For Array.” Use the arrow keys (rather than the mouse) to keep the starting point (and thereby the selection) on the snap grid. Before the array is placed a Confirm dialog box will open.

7. Click YES (or press ENTER or LEFT MOUSE or y) to proceed with the placement. Click NO (or press n) to cancel the placement.

The selection will be placed as an array in the board window. If a portion of the array will be placed outside the workspace boundaries the warning message “Selection Is Outside Board Window”

Example: Array placement

To place a vertical row of pads for one half of DIP16 component, place the first pad manually. Edit the pad (Change Pad) and set its designator to 1. Select the pad and use the Edit-Cut command (SHIFT+DEL) to copy it to the clipboard, providing a reference point, as prompted.

Choose Edit-Place-Array (shortcut: p, y). Set Y-offset to –100 mils X-offset to 0 and Text Increment to 1. Set Item Count to 7 Now you will be prompted “Select Starting Point for Array” Click LEFT MOUSE in the workspace. A row of pads will be placed, descending down from the manually placed pad. There designators will be 1, 2, 3, 4, 5, 6, 7, 8 from top to bottom.

For more information see page 109 of the *Advanced PCB Reference*.

Changing the board layout

Advanced PCB provides a graphical editing environment that supports direct manipulation of the placed primitives that constitute a PCB layout.

Navigating the workspace

A number of commands and features assist the designer in navigating PCB document windows to either place objects or to find previously placed items.

Jump To

The Edit-Jump To commands (shortcut : ALT, e, f) allow 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. These options are particularly useful when manually routing a board or checking a laid out board. Search For commands include:

Component

1. Type the designator in the Search For dialog box 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.

2. Choose from the Components Placed dialog box and click OK.

The cursor will jump to the center of the reference pin (usually pin 1 or A1) of the selected component.

Net

1. Type the net name in the Search For 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.

2. Choose from the Nets Loaded dialog box and click OK.

The cursor will jump to the nearest pin that belongs to the selected net.

Pad

1. Choose a placed component to open the Jump to Pin Number dialog box.
2. Type the pin number and click OK.

The cursor will jump to the center of the named pin.

String

The cursor will jump to the named string. This option works with free text strings only – not component text strings. 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 and length but ignoring case.

Finally – for a string with same characters in it but perhaps having more characters (and ignoring case).

For example, typing “Component” would find the string “Component” first. If no match is found it would next find the string “CompONENT” and finally “CompONENTs.” When the string is found, the cursor will be relocated to the specified string.

With all of these options, Advanced PCB will only redraw 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.

Absolute Origin

Jumps to the absolute (0,0) coordinate. In the Advanced PCB system, this is the lower-left corner of the workspace.

Current Origin

Jumps to the current (or relative) 0,0 origin. This is an origin generated by the Edit-Set Origin command.

New Location

This command allows you to type in the desired coordinates for the jump.

Type an X coordinate (a distance from the left hand side of the work space), the default is the current X position. Type the desired location and press ENTER or LEFT MOUSE.

Type a Y coordinate (a distance from the bottom of the work space), the default is the current Y position. Type the desired location and press ENTER or LEFT MOUSE. The cursor will now be moved to the specified location.

DRC Error

This command jumps to the first DRC error marker. Repeating this command will jump to a second error marker, etc. Removing the violation will clear the error from the "Jump to" list. Otherwise, repeating the command will continue to cycle through all errors in the current document window.

For more information, see page 114 in the *Advanced PCB Reference.*

Setting a new origin

The absolute origin ("0,0" coordinate) is the extreme lower left corner of the workspace. Advanced PCB allows you to set a new 0,0 coordinate anywhere in the workspace.

To set a relative origin at the current cursor position:

1. Move the cursor to the desired coordinates.
2. Choose Edit-Set Origin command (shortcut: ALT, e, o).

The status line will now display X:0mils Y:0mils (or X :0mm Y:0mm if a metric snap grid is used) at the current cursor position.

To restore the absolute origin (extreme lower left corner of the workspace) choose the Edit-Reset Origin command to return the absolute origin.

For more information, see page 115 in the *Advanced PCB Reference.*

Displaying pad designators and nets

The Options-Preferences command allows you to show either or both pad numbers and pad net names for all pads in your layout. Free pads (those not belonging to library components) are displayed with the default pin number “0” (zero) until re-numbered. You can edit these identifiers for both free pads and component pads. Double-click on any pad to open the Change Pad dialog box. Net names can be a maximum of 20 characters. Pad names can be up to four alphanumeric characters.

Cross probing Advanced Schematic

Among several features that provide close links between Protel’s Advanced PCB and Advanced Schematic are three cross probe commands. A complimentary set of cross probe commands is provided by both applications.

These commands allow the user to move easily between the schematic and physical layout elements of a project.

From Advanced PCB, users can:

Use the Edit-Cross Probe Part On Schematic command to jump to the schematic part(s) that correspond to a PCB component.

Use the Edit-Cross Probe Pin On Schematic command to jump to a schematic pin that corresponds to a PCB component pad.

Use the Cross Probe Net On Schematic command to display a schematic net that corresponds to a physical net in the PCB.

See page 116 of the *Advanced PCB Reference* for additional details.

Changing items on the PCB

Once primitives (including components and other "collections" of primitives have been "placed" or added to the layout, they can then be modified to accommodate design changes. Changes can be as simple as moving a single item to a new location or complex. Advanced PCB includes a number of features which automate many board editing processes. Before using these features, it is useful to learn some of the underlying principles that apply to Advanced PCB editing. The concept of selection – choosing one or more items to be modified – is fundamental to all "change" processes.

Selection

The Advanced PCB system allows you to change the elements of your design in two ways:

Choose a command, such as Delete, then select an item to be removed, or.

Select one or more items, then choose a command which acts on the selection such as Cut, Copy, Move, Clear, etc.

Selection is simply the process of choosing one or more items, upon which some action is to be performed. Under Advanced PCB, selection is a highly flexible process. You can select multiple items, either as a group, or one-at-a-time. Similarly, you can remove items from the current selection individually or in groups.

Protel Autotrax users:

In Advanced PCB the functions of the Autotrax Block and Highlight commands (such as Block Define or Highlight Net) have been merged into a single concept: Selection. If you wish to perform an operation on a group of primitives on the printed circuit board, you first select the items, then use one of the Edit menu commands. Operations performed on single primitives can be performed using the action-target model similar to Autotrax. For example, choosing the Edit-Move-Pad command (the "action"), prompts you to Select a Pad (the "target").

Selection and highlighting

When you select an item, the color changes from its normal layer color to the Selections color assigned from the Options-Layers dialog box. The item remains selected until you remove it from the selection using either the De-select command or SHIFT+LEFT MOUSE.

Highlighting is related to selection but works within an operation, such as re-routing a track or when generating a netlist. You can see tracks highlight during both of these processes. As primitives highlight, you will notice that highlighted item is not displayed in the Selections color but in black.

Both selection and highlighting are based on the geometry of primitives in the workspace – not on the current netlist. In other words, items that are physically connected are included in the selection. This allows you to use selection to trace the connectivity of your design – like a "continuity" check as you manually route connections and to perform other selection-based operations, such as Cut, Copy, Clear or Paste.

When you select connections or nets, other previously selected items stay selected until de-selected.

Selections are generally made in four ways:

- By direct selection, using SHIFT+LEFT MOUSE to add (or remove) individual items to the current selection.
- By using the Edit-Select and De-select commands to define a selection group.

- By choosing an action, like the Move command, then *selecting* a single target item for that action.
- By using the Selection field in Change dialog boxes. This options includes the ability to use Advanced PCB's global editing feature to apply selection status changes to other primitives in the current board window.

Making selections

Direct selection using the mouse and keyboard

Direct selection is the most flexible way to designate one or more items to be moved, copied, cut and pasted into a new location or deleted. To select one item at a time:

1. Hold down SHIFT and click LEFT MOUSE with the cursor position over an item.

The item will be redrawn in the Selection color (Options-Layers command). 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. Components, especially complex components can take a moment to select.

➡ The Snap to Center option (Options-Preferences command) will cause the cursor to "snap" to the nearest primitive, when making the selection. This can make it easier to select items from a dense layout, or when zoomed out from the board.

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 any selected item.

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

You can use the Zoom commands (and hot key shortcuts PGUP and PGDN) at any time when making or releasing selections.

The selection hierarchy

When making a direct selection using SHIFT+LEFT MOUSE, the selection routine looks first for vias, then tracks, then pads, then area fills, then strings and then components. If you wish to select a component using this method you simply position the cursor inside the component, being careful to avoid any free primitives that are placed inside the component. The same selection hierarchy operates when you de-select items individually, using SHIFT+LEFT MOUSE.

Edit-Select and Edit-De-select

The Edit-Select commands allows you to select all items inside or outside of an area, all items on one layer, or all free primitives (items other than components, dimensions or coordinate markers). You can also select by physical net or physical connection. These options allow you to extend the selection (or de-selection) beyond a few items.

Edit-De-Select provides the same options, less the Physical Net / Connection and Hole Size commands. Using these two sets of commands you can define complex grouping which can then be moved, copied or deleted. The Edit-Toggle Selection command allows you to turn the selection state of individual primitives “off” or “on” duplicating “direct” selection performed using SHIFT+LEFT MOUSE, described above. Shortcut: press x to choose from the De-Select options.

Select and De-select commands include:

Inside Area

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

The newly selected primitives will be highlighted using the selection color. Any previously selected items will remain selected until de-selected.

Outside Area

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

All

This command selects everything placed in the Protel document window.

Free Primitives

This option selects all tracks, vias, free pads in the document window. Components are not selected. This feature is useful for stripping a routed, or partially routed board back to its “placed” condition.

All On Layer

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

Physical Net

This command selects all free primitives (tracks, vias, fills) that connect a physical net. Component pads within the net are not selected.

To use this feature:

1. Choose Edit-Select Physical Net (shortcut: CTRL+h).

You will be prompted "Select Pad."

2. Position the cursor over a component pad within the desired net and press ENTER or LEFT MOUSE.

The continuous physical net extending from the selected pad will highlight with the selection color.

Physical Connection

Choose Edit-Select-Physical Connection (shortcut: s, c) to select the free primitives (tracks, vias, fills) that connect any two component pads. The component pads are not included in the selection.

Hole Size

Choose Edit-Select-Hole Size (shortcut: s, c) to select pads or vias with holes of a nominated diameter.

For more information, see page 48 in the *Advanced PCB Reference.*

Toggling the selection status

The Toggle Selection command allows you to select or de-select individual primitives, by type. Using this option, rather than the SHIFT+LEFT MOUSE shortcut, can be helpful when you are working in a densely populated area of your design where components and other primitives are overlapped or when you wish to quickly add and remove a number of items from the selection. To use this feature:

1. Choose the Edit-Toggle Selection command (shortcut: t or ALT, e, n) then the primitive type: Arcs (a), Components (c), Fills (f), Pads (p), Strings (s), Tracks (t) or Vias (v).

You will be prompted "Select (primitive type)"

2. Click on the selected primitive type to add or remove the item from the current selection.

The prompt “Select (primitive type)” will be continuously displayed.

4. Press ESC to leave the Toggle Selection command.

Saving selections as a PCB file

Any selection can be saved as a Advanced PCB file using the File-Export Selection command. When you use this option you are creating a binary file which can be opened in a new Protel document window at any time. This provides a convenient way to store modular design elements for future use. You can use the Copy and Paste commands to move selections between open PCB document windows. Before using this command, make sure that the current selection includes only those items you wish to save.

If a netlist is attached to the current file, the netlist information is not attached to the exported selection.

Importing selections from another PCB

Any part of a PCB file can be imported into another PCB file. This is done by:

1. Opening a new file document window (File-Open command).
2. Using the Edit-Copy command to copy the file (or a selection of the file) into the Protel clipboard.
3. Using the Edit-Paste command to add the file (or selection) to the current PCB.

The file is then scanned and its size determined. The imported selection will be displayed as an outline with the cursor positioned at the predefined reference point. The selection can be moved, rotated or flipped during placement. The following section, “Moving a selection,” describes the process of positioning a selection in the workspace.

When a selection has been added to the current layout, any components that have been added are automatically re-designated to avoid duplication of existing designators.

If the imported file has an attached netlist, the netlist will not be loaded when you import the file.

➡ 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. Windows Metafile (.WMF) graphics format is not supported in the Advanced PCB clipboard.

For more information, see page 42 in the *Advanced PCB Reference.*

Selection and other Windows commands

Standard Windows commands, such as Cut and Paste, can be used to manipulate a selection. These commands work with the internal Advanced PCB clipboard which operates like the standard Windows clipboard.

For example, the Edit-Cut command (shortcut: SHIFT+DEL) will copy a selection to the clipboard and remove the selection from the workspace, and the Clear command simply deletes the current selection and no copy of it is retained in the clipboard.

Undo and Redo commands

Advanced PCB includes a full multi level Undo and Redo facility. Each procedure is stored in a stack-like arrangement. When the Undo command is called, the last operation is undone. Choosing the Undo command again will undo the next-to-last operation, and so on. Operations are grouped together in Undo for convenience. For example, if you choose Delete Track and then delete two tracks, undoing this operation will return both of the tracks. A new Undo set is started every time you select a command.

The Redo facility will reverse previous “Undos.” 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 and then the next-to-last and so on.

If either Undo or Redo are available (when an operation which can be undone has been performed), the options in the edit menu will be available – otherwise they will be dimmed. Undo and Redo operations are very memory intensive, as these store every operation performed from the start to the end of your editing session. The File-Save As command clears the Undo memory stack.

➡ If Advanced PCB performance slows during an editing session, try clearing the Undo/Redo memory stack by using the File-Save command.

Moving a selection

Once selected, an individual item or a complex selection containing many items can be moved as a single entity. When you select a Move command the current cursor location becomes the reference point for moving. This operation is performed using the Edit-Move-options:

Move Selection

This option allows you to select a new location for the selection, which will move as a block (shortcut: m, m).

➡ Moving selections within a routed PCB can sometimes produce unexpected results. For example, if a component is partially within a selection and some tracks connected to it are inside the selection, then the tracks will be disconnected and “dragged” off the component during the move, damaging the connective integrity of the layout. You can use the Undo and Redo commands to recover from such an unintended change.

Flip Selection

This option flips the orientation along the vertical axis. Pressing the x or y keys during the move accomplishes this action.

Rotate Selection

The selection becomes free to rotate in .001 degree increments (shortcut: press SPACEBAR to rotate the selection clockwise). The amount of rotation is pre-set under Options-Preferences. Components, tracks, text strings and polygon planes can be rotated to any angle. Non-symmetrical pads in selections will rotate have restricted rotation properties, described below.

When you choose the Edit-Move-Rotate Selection command, Advanced PCB will prompt for a rotation value (degrees). This command has a .001 degree resolution. As with other Move commands, the current cursor position becomes the reference point for the selection, in this case the reference point serves as the axis for the rotation.

➡ When you rotate selections there are a number of limitations that you should be aware of. For example, rectangular pads and rectangular fills will have their position rotated but will maintain their sides and top and bottom parallel to the x and y axis of the printed circuit board.

For Gerber plot final artwork, the target photoplotter must allow rotated primitives. You may need to globally change non-round component pad shapes to round before rotating in other than 90 degree increments.

For more information, see page 82–83 in the *Advanced PCB Reference.*

What gets moved?

If one end of a track is inside the selection rectangle, only that end will be moved. If the center of pads and vias fall within the selection, the whole item will be selected with the selection. Fills or components are only moved if they lie entirely within the selection. If a component with tracks is partially within a selection and some tracks connected to it are within the selection, then the tracks will be dragged off the component damaging the connective integrity of the PCB layout.

Moving individual items

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

To move any item

1. Position the cursor over the item to be moved.
2. Hold down CTRL and click LEFT MOUSE once.

It may take a moment before the item becomes highlighted. If the system "beeps" you may not be exactly over the item to be moved. Use the Zoom options (or press PGUP) to enlarge your view of the board and try again.

3. Click LEFT MOUSE again and move the item to the desired location.

Moving or deleting one or more items can leave a "hole" in the display under the moved/deleted primitives. This is because Advanced Schematic 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.

Components and text strings can be rotated in clockwise steps during moves by pressing SPACEBAR. The degree of rotation is pre-set under the Options-Preferences command. Non-symmetric pads can be rotated in 90 degree increments without distortion.

Arc

Arcs can be moved to a new position after placement, without restriction.

Component

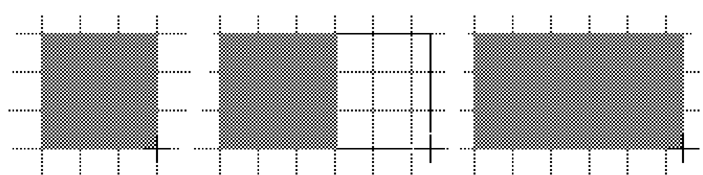
When you move a component, all of its associated primitives move as a group, including the pads, outline and component text (on the Overlay). Tracks connected to the component pads will be dragged as you move the component, if the Drag Tracks option is enabled in the Options-Preferences dialog box. If this option is disabled, the track/pad connections will be broken when the component is moved.

While moving a component, you can rotate it around the cursor and flip it along its x or y axis.

If you wish to move component text without moving the footprint, you can use the Edit-Move-String command (or CTRL+LEFT MOUSE shortcut). Zoom-in, if needed, to position the cursor directly over the designator or comment string only. Moved text remains associated with the component, so if you move the component later, the text will move relative to the component, as before.

Fill

Free rectangular area fills can be either moved to a new location or re-sized.



Fills are re-sized by clicking at the starting or ending corners, then dragging.

Once placed, you can re-size a fill by dragging the starting or ending corner to a new position.

Pad

Free pads (pads that are not part of a component) can be repositioned, like other primitives. While moving an asymmetric pad, you can rotate it around the cursor with the SPACEBAR. Each time you press SPACEBAR the pad is rotated 90 degrees clockwise.

String (text)

Both free text strings and component text strings are moved using the same procedures. While moving a text string, you can rotate it around the cursor and flip it along its x or y axis.

Moving Tracks

Four special commands are used to control the type of move applied to tracks: Break track; Drag End; Whole Track and Re-Route.

➡ When breaking, dragging ends or re-routing tracks, you can toggle through the active layers by using the *, + or – keys. Pressing * toggled between all active signal layers. Pressing + or – toggles through all active layers. As you change layers, a via will be automatically placed at the junction of the two segments. The Auto Via option (Options-Preferences command) enables/ disables this feature.

Break Track

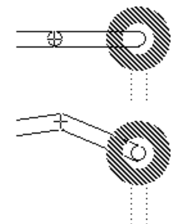
This command converts a single track segment into two connected segments. To break a track:

1. Choose the Edit-Move-Break Track command (shortcut: m, b).

When you choose this command, you are prompted “Select track.”

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

The track will be highlighted and the prompt “Dragging broken track” will be displayed on the Status line.



Move-Break Track command

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.

The highlight will disappear and you will be prompted “Select 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 cancel the move.

Hot key shortcut: press SHIFT+CTRL, position the cursor at the break point then click LEFT MOUSE once.

Drag End

This command is used to move the end of a track (and any other track ends or vias that are connected to it at that point). The other end of the track segment is left undisturbed by the move. To drag the end of a track:

1. Choose Edit-Move-Drag End (shortcut: m, d).

You will be prompted “Select track.”

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

The cursor will jump to the nearest end of the track segment. The selected segment and any intersecting segments connected to the end will be highlighted. The prompt "Dragging track end" will be displayed on the Status line.

3. Drag the track end(s) to the new location and press ENTER or LEFT MOUSE again.

"Select track" will be displayed on the Status line. If a via is part of the highlighted intersection, it will move along with the intersecting tracks.

4. Select another track or press ESC or RIGHT MOUSE to cancel the move.

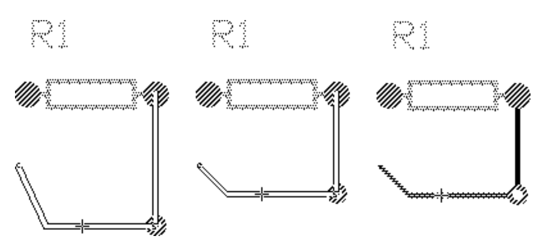
Whole Track

To move both the complete track segment (and all connecting track ends and vias):

1. Choose Edit-Move-Whole Track.

You will be prompted "Select track."

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



Move-Whole Track command

The selection will highlight, including all connected track segments and vias. The prompt changes to "Moving track."

3. Move the cursor to drag the track(s) to the new location and press ENTER or click LEFT MOUSE again.

"Select track" will be displayed on the Status line.

4. Select another track or press ESC or RIGHT MOUSE to cancel the move.

Re-Route

This feature is similar to the Break Track command. When re-routing, a new vertex is formed at the current cursor position. Each time you press ENTER or LEFT MOUSE, the highlighted track segment is terminated and a new segment is started. To re-route a track:

1. Choose Edit-Move-Reroute (shortcut: m, r).

You will be prompted "Select Track."

2. Position the cursor over the track segment and press ENTER or LEFT MOUSE to select a track.

The track will break at the cursor position and the two created track sections will rubber band with the cursor.

3. Drag the highlighted segments to a new location and press ENTER or LEFT MOUSE again.

One end becomes a "placed" track segment (and the highlight is removed). Two highlighted segments are now displayed from the end of the newly placed segment.

4. Continue to move and press ENTER or LEFT MOUSE to reroute the track segment-by-segment.
5. When finished, press ESC or RIGHT MOUSE and a track segment will be placed between the end of the last segment and the end of the originally selected track.

You will then be prompted "Select Track."

To finish Re-routing tracks, press ESC or RIGHT MOUSE.

You can change layers at any time during this operation with the *, + or - keys. If you specify two consecutive track segments, and change layers between, then a via will be placed at their intersection. (This Auto Via feature can be disabled).

Move track shortcuts

To select and move a track:

> Hold down CTRL and click LEFT MOUSE with the cursor directly over a track segment.

One of two different move commands will be executed. If you are within 25% of the track end, then the end will be moved as though you had chosen the Drag End command. If you select the middle 50% of the track then the whole track will be moved together with any tracks or vias that are attached to the ends of the track.

To “break” a track (convert it into two joined segments).

> Hold down the SHIFT and CTRL keys together and click on the track.

The track will be broken and you can drag the segments to a new location.

For more information, see page 84 in the *Advanced PCB Reference.*

Via

Vias can be moved, like other primitives, using the Edit-Move-Move Selection command or move shortcuts (CTRL + LEFT MOUSE).

Moving Coordinate Strings and Dimensions

These automatically generated items are placed as simple primitives: track segments and free text strings. You can move, edit or delete these primitives using the standard procedures for strings and text.

Cutting a selection

The Edit-Cut command clears the current selection from the workspace and copies it to the (internal) Advanced PCB clipboard (not the standard Windows clipboard). The Edit-Paste command can be used to place the selection back into any open Protel document window.

➡ The clipboard holds the last Cut or Copy selection contents only, each time you use the Cut or Copy command, 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.” A reference point is a coordinate relative to the selected item(s). When the selection is Pasted, the reference point will locate the cursor at this same relative position, for accuracy.

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 the Zoom-Redraw command (shortcut: press END) to restore the display under the selected items.

➡ When using a mouse, the cursor is not “tied” to the current snap grid. However, when you designate a reference point during Cut or Copy, the grid point nearest to the cursor will be used.

See page 47 in the *Advanced PCB Reference* for additional information about this command.

Copying a selection

The Edit-Copy command copies the current selection to the (internal) Advanced PCB clipboard (not the standard Windows clipboard). The Edit-Paste command 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 the Cut or Copy command, 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.” A reference point is a coordinate relative to the selected item(s). When the selection is Pasted, the reference point will locate the cursor at this same relative position, allowing accurate positioning.

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

The selection will be copied to the clipboard.

➡ When using a mouse, the cursor is not “tied” to the the current snap grid. However, when you designate a reference point during Cut or Copy, the grid point nearest to the cursor will be used.

See page 47 in the *Advanced PCB Reference* for additional information about this command.

Pasting a selection

The Edit-Paste command can be used to place the current clipboard contents into any open Protel document window. Advanced PCB has its own clipboard format. The standard Windows clipboard is not used.

To copy 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 cursor position relative the selection is determined by the Reference Point designated 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 command to duplicate the selection.

When a selection has been added to the current layout, any components that have added will be automatically re-designated to avoid duplication of existing designators.

See page 48 in the *Advanced PCB Reference* for additional information about this command.

Clearing a selection

The Edit-Clear command 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 the Zoom-Redraw command (shortcut: press END) to restore the display under the selected items. You can use the Edit-Undo command (shortcut: ALT+BACKSPACE) to restore the cleared selection.

See page 48 in the *Advanced PCB Reference* for additional information about this command.

Delete

The Edit-Delete commands are similar in many ways to using the powerful combination of selection and the Cut or Clear command described in the previous section. However, this command reverses the selection-then-action sequence used with Cut or Clear.

With Delete, you choose the type of item to be deleted (track, free pad, component, etc.), then select the individual item with the cursor. If you “miss” or position the cursor over the wrong kind of primitive, the system will simply “beep” allowing you to try again. This provides an efficient way to clear several primitives of one type from the layout.

Delete is independent of selection. In other words, the current selection is unaffected as you use the Delete Component command. For example, when deleting individual tracks, tracks that are part of the current selection will be left undisturbed.

➡ Move and delete commands will sometimes leave “holes” in the display, rather than immediately redraw the screen in each instance, which would slow down system response. Use the Redraw button on the Tool bar or press END to refresh the display during or following these operations.

If the Question Delete option is enabled, the warning message “Confirm Delete Item” will be displayed as you select each deletion. You can enable/disable this message in the Options-Preference dialog box.

To Delete any individual primitive types:

1. Choose Edit-Delete (shortcut: d).
2. Choose the target primitive type (shortcuts: Arc (a), Component (c), Fill (f), Pad (p), String (s), Track (t) or Via (v).

You will be prompted "Select (primitive)."

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

The selected item will be cleared from the workspace and the prompt "Select (item)" will be displayed on the status line.

4. Continue deleting additional items or press ESC or RIGHT MOUSE to leave this command.

If you want to delete a component by specifying the designator, select some empty space and press ENTER or LEFT MOUSE, then type in the designator (or *?* if you are not sure of the designator). All placed components will be listed. You can scroll the selection bar through the windows to make your selection.

All deletions can be restored by using the Edit-Undo command (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. The Edit-Redo command uses the same first-in/last-out logic. Redo reverses the Undo operations, one-at-a-time.

See page 56 in the *Advanced PCB Reference* for additional information about these commands.

Change commands

The Change commands are used to modify specific attributes of placed items. Each item or primitive has its own range of editable attributes. You can change one item or extend changes across your entire design using powerful global editing options.

To change any placed item, move the cursor over the item and double-click the LEFT MOUSE button. This shortcut will open the Change (item) dialog box for the selected item. For tracks, pads, vias fill or arcs, the associated net name (if any) will be included in the dialog box.

You can also use the Change command from the Edit menu:

1. Choose Edit-Change (shortcut: e, c).
2. Choose the type of item to be changed: Arc (a), Component (c), Fill (f), Pad (p), Track (t) or Via (v).

The prompt "Select (item)" will appear on the status line.

3. Position the cursor over the target item and press ENTER or LEFT MOUSE.

A dialog box opens, displaying the editable attributes for the item. It is now possible to change any or all attributes of the selected item, such as a track width or layer. These options are described in the next section, Global Changes.

4. Press ENTER or click OK to accept your changes. To cancel the change and close the dialog box press ESC.

The status line will prompt "Select (item).

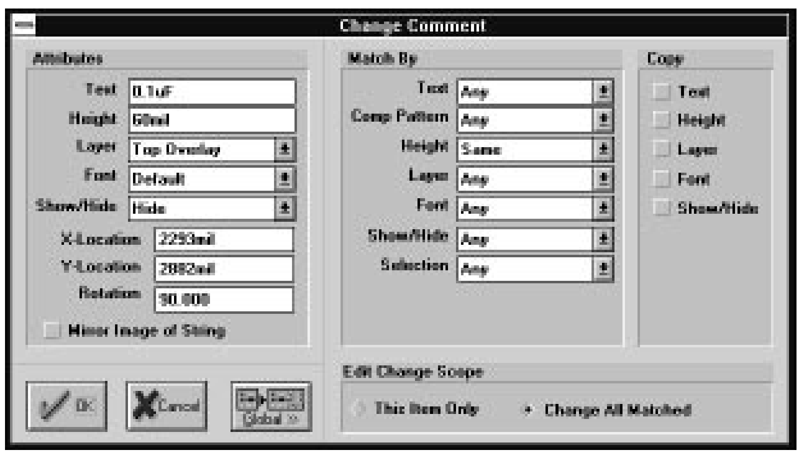
5. Select another item to be changed or press ESC or LEFT MOUSE to leave the Change command.

➡ The system will not prevent changes that violate specified design rules. A Design Rule Check (DRC) should always be performed before final artwork is generated. For more details See Netlist-Design Rule Check.

See page 58 in the *Advanced PCB Reference* for additional information about these commands.

Global changes

Changes can be made to a single selected item or they can be applied globally across the entire board using flexible, powerful global editing options. Most editable attributes can be globally applied to other placed primitives of the same type. A simple example would be changing the hole size assigned to a single pad. Typically, the designer would want this new hole size applied to other pads as well. However, it might be important that the new hole size be applied only to pads with the same (original) hole size as the selected pad, or perhaps pads of a specific shape or size only, etc.



Advanced PCB Global edit options allow the designer to extend changes to other primitives in a single operation.

These options (and more) are supported by the Change commands described below. 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. Working through the two examples at the end of this section will illustrate some potential uses.

Matching attributes for global changes can be assigned by clicking the Options button in any of the Change (item) dialog boxes. When you click Options, the dialog box expands to display the parameters for global matches.

Each Change (item) dialog box may contain different options since every type of item has a unique set of attributes. For example, tracks have three globally editable attributes (width, layer assignment and selection status), while pads have eight.

There are four groups in the expanded dialog box:

Attributes

These are the changeable parameters of the item. For a Track, the changeable attributes are width, layer and selection status. Any or all of the changeable attributes may be changed at each opening of the dialog box. Some attributes, such as coordinate points are not globally editable.

Match By

These attributes define matching criteria for other items. There are three settings possible for each copyable attribute:

Same

Change all items which match the original value of the item being changed.

Different

Change all items which do not match the original value of the item being changed.

Any

Change whether or not the attribute matches the item being edited.

Copy

Any number of the editable attributes can be copied to other items. Click in the box corresponding with the desired attribute to assign the changes to the target items.

Matching can be performed using mixed attributes. For example, track matching attributes include width, layer, selection status and net assignment. It is possible to make a global changes to currently selected tracks of any width, on the same layer, etc. This feature provides comprehensive control over global changes. Options for each editable item are described below.

Edit Change Scope

This option applies changes locally (to the current primitive only) or globally (to all matching items).

➡ Global change options are also provided for editing the nets in a loaded netlist. See the section Nets and Netlists, for further details.

Changing arcs

Arc attributes which can be edited include:

Line Width

Arc line width can be any value between 0.001–9999.999 mils. The current default track width setting is used as the default arc line width.

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.

Selection

Arcs can be globally selected or de-selected.

Net

Arcs can be matched by association with a particular net. Net assignment is not editable at the individual primitive level.

Coordinates and dimensions

Arc center coordinates, radius (to 0.001 degree), start angel and end angle can be edited. These attributes cannot be used to globally edit other placed arcs.

See page 58 of the *Advanced PCB Reference* for details.

Changing components and component text

Component attributes which can be edited include:

Designator/Comment

Component designators and comments have a number of editable attributes:

Text

The component label, U1 etc. This attribute is not globally editable as each component must have a unique designator. Designators can be a maximum of 8 characters in length (spaces are not allowed). It is standard practice to use a single letter prefix and numeric suffix.

➡ Advanced PCB stores the designators used each time you place a component and will automatically supply and incremental designator when you place any previously placed pattern. For example, if you designate a DB9F connector “J1” – a second DB9F will be automatically assigned a default designator of “J2,” and so on.

Height

The size of the text in mils (.001 inch) or mm.

Layer

Designators 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.

Font

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

Show/Hide

Component text can be displayed or hidden. Hidden text will not be printed. Click the Show/Hide button to toggle the display status.

Comment

Component comments share the same editable attributes as component designators. However, unlike designator text, comment text changes can be globally applied to other components.

Pattern

The current physical pattern (or footprint) of the component can be changed to any other available pattern in an open component library. If you type a new string in the Pattern Edit box, and the Edit Component dialogue box, when you exit the dialogue box Advanced PCB will search the current libraries to try and locate the new pattern that you have specified

Component patterns can be changed freely; however, if there are netlist connections to the pads, the new footprint must have the same pin numbers available as the previous one or a warning message will be displayed and the substitution will be aborted. For example, changing a DIP16 to an SMD16A would update the internal connection list. Changing a DIP16 to a TO-3 would generate a warning and the change would be aborted. The ratsnest display would also be updated to reflect these un-routed nodes. A DIP16 can also be changed to a DIP14, providing there are no connections to pins 15 and 16 on the DIP16. These rules also apply to global pattern changes.

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.

Determining whether a component is on the top layer or the bottom layer is based upon the designator and comment strings. When a component is first loaded from the library, the component is placed on the top layer and the strings are either placed on the top layer or the top overlay, depending upon how they have been set up using the Component Text dialogue box. When you enter the Change Component dialogue box Advanced PCB looks at these strings and if it finds either of the designator string or the comment string on the bottom layer or the bottom overlay then it assumes that this component is on the bottom layer otherwise it assumes the component is on the top layer. When the component is swapped to the other layer both of the strings, that is the designator and the comment, are swapped also and mirrored. There is no PCB database attribute for component layer assignment.

Place Status

Component placement status has two possible states. Free To Move indicates that the component position can be changed by the Auto Place options. Locked In Place indicates that the component will not be moved by Auto Place. When a component is first placed by Auto Place its status remains Free To Move until its status is changed by the user. This allows you to lock seed components into place and then re-run Auto Place.

Selection

Components can be selected or de-selected globally.

X, Y Location

Component reference coordinates can be edited. These attributes cannot be used to globally edit other placed coordinates.

Global Options

Click Global to view the global edit options for components. Remember, component designators and comments have their own global options, described above.

The Wildcard field can be used to match placed components, using command designator strings, e.g. *U?* to apply changes to all components with a designator prefix of “U”.

See page 61 of the *Advanced PCB Reference* for more information about this command.

Changing fills

Fills can be assigned to any layer of the PCB or have their Selection status changed. Additional attributes that can be changed include:

Net

Fills can be matched by association with a particular net. Net assignment is not editable at the individual primitive level.

Coordinates

Fill corner coordinates can be edited. These attributes cannot be used to globally edit other placed fills.

See page 67 of the *Advanced PCB Reference* for more information about this command.

Changing pads

Both "free" pads and component pads can be individually and globally edited. Attributes which can be edited include:

Designator

Pad designators are used to label pads with pin numbers when creating a new component. Free pads can include a optional designator or this value can be left empty.

X Size

This field describes the size of the pad on the horizontal (x) axis. Range is 1.000–500.000 mils.

Y Size

This field describes the size of the pad on the vertical (y) axis. Range is 1.000–500.000 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.

Shape (and size) attributes can be independently assigned to the Top layer, Mid 1–14 layers and Bottom layer. This allows the creation of "pad stacks."

Hole Size

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

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. Multi-layer pads can have size and shape attributes individually assigned for the Top layer, Mid 1–14 layers and Bottom layer to create “pad stacks.” Click the Layer button to scroll the selection bar through the available layers to choose a new layer.

Pwr/Gnd

This attribute is used to manually assign a through-hole (Multi-layer) pad to one of four internal power planes. Pads can be connected to a power plane by either a Direct or (thermal) Relief connection. SMD pads can be Tagged To Plane for automatic connection during automatic or interactive routing.

Selection

Pads can be selected or de-selected.

Global Options

Click Global to view the global Match By edit options for pads: designator, x size, y size, hole size, component pattern, layer, shape, internal power plane layer assignment, selection status, group and net assignment.

Group refers to pads that are part of a component. This option allows you to restrict changes to component pads, free pads or to allow changes to be made to any pad. Group or net assignment are not editable at the individual primitive level.

For more information, see page 69 of the *Advanced PCB Reference.*

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 32 characters long and any alphanumeric character, including spaces. Default string font and height is set using Current menu commands.

Height

Text size can be set in mils (.001 in) or mm. Range is 0.500–9999.999 mils. The stroke width and character width used to display/print the text is automatically proportioned to the height assigned. A minimum text height of 36.000 mils will be allow strings to be legibly photoplotted.

Layer

Text strings can be assigned to any layer.

Font

Three special vector fonts are available: Default, Sans Serif and Serif. These fonts support vector plotting.

Selection

Text can be selected or de-selected.

Coordinates, Rotation and Mirror Image

String coordinates and rotation can be edited but these changes cannot be globally assigned. The Mirror Image of String option “flips” the string for plotting in the correct orientation on bottom-side artwork layers.

Global Options

Click Global to view the global edit options for strings. Copyable attributes include Text (contents of the string), Height, Layer, Font and Selection status.

See page 73 of the *Advanced PCB Reference* for additional information regarding this command.

Changing tracks

Tracks can be changed both individually and globally. Editable text track attributes include:

Width

Track width can be set in mils (.001 in) or mm. Range is 0.001–9999.999 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.

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.

Global Options

Click Global to view the global edit options for tracks. Copyable attributes include width, layer assignment and selection status. The Include Arcs in Global Edit option allows curved tracks in connections to be included in global width, layer or selection changes. Net assignment can be used to match existing tracks, but the assignment is not an editable attribute at the individual primitive level.

See page 75 of the *Advanced PCB Reference* for more information about this command.

Changing vias

Vias are special purpose pads used to connect tracks between different layers. Vias can be placed through all layers (placed on the Multi layer) or on any layer pair (placed as blind/ buried vias). Vias are always circular in shape. If a shape other than circular is required a free Pad can be used. Editable via attributes include:

Diameter

Vias can be assigned any size in the range of 0.002–500.000 mils.

Hole Size

This option assigns the hole diameter to be drilled through each via. Via hole size can be assigned independently of the via size. The range is 0.002–500.000 mils.

➡ Via hole assignments should be determined in consultation with your board fabrication source. The diameter required may vary with the plating process used in fabricating the board.

Layer Pair

Via layer assignments are made automatically when the vias are placed, either by the autorouter or when manually routing tracks (with the Auto Via option enabled under the Options Preferences command).

➡ It is possible to change layer pair assignments individually, but this must only be approached with great care, due to the complexity of layer pair management. See the section on Autorouting for more information.

Selection

Vias can be selected or de-selected.

X, Y Location

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

See page 77 of the *Advanced PCB Reference* for more information about this command.

Global Options

Click Global to view the global edit options for vias. Copyable attributes include diameter, hole Size, layer pair assignment and selection status. Net assignment can be used to match but is not editable at the individual primitive level.

Examples of global changes

The following examples will give you a limited idea of the potential scope of global changes to 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 and click Options to display the global edit parameters.
2. Under Attributes, choose Bottom layer, (Track Width can be left unchanged).
3. Under Attributes to Copy, Layer is activated automatically.

Be certain that Width is not checked so that Tracks of any width will be changed.

4. Under Attributes to Match By, set Width to Any; Layer to Same and Selection to Any.
5. Under Edit Change Scope choose the Change All Matched and click OK or press ENTER.

The initial track selected will be swapped to and redrawn in the color assigned to the Bottom layer. The Confirm Global Change dialog box opens.

6. 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 restore all items in the display.

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.

If you have problems selecting a via, press PGUP to zoom-in on the board.

2. Type *40* in the Diameter field under Attributes.
3. Click Options and enable the Diameter option under Attributes To Copy.

Leave the Hole Size and Layer Pair disabled.

4. Under Attributes To Match By, choose Any for each of the four options (e.g. any placed via will be changed).
5. Under Edit Change Scope, choose the Change All Matched then 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.

Global editing summary

The two examples above show the most basic application of the global change options. With care and planning the designer can 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 create a fresh selection in these cases. Be sure to maintain adequate backup versions of your workfiles. Don’t simply rely on the auto-backup feature but archive various versions of your design, particularly if you intend complex changes. Finally, remember that the Undo/Redo features can allow you to recover several operations, if required.

Placement is one of the fundamental design processes in Advanced PCB. A number of different methods are provided for placing the components and primitives (tracks, vias, etc.) in the document window. For example, you can place individual components at any time using the Place Component command or Component tool button. Components are automatically loaded into the workspace, when you load a netlist. Similarly, tracks can be placed manually or automatically when using autorouter options.

Repour Polygon

This change command allows the user to change polygon plane (copper pour) attributes without having to re-place the polygon. When the Edit-Change-Repour Polygon command is chosen, the Place Polygon Plane dialog box opens.

All polygon attributes can be re-defined: net assignment, dead copper removal, tracks (horizontal and/or vertical), fill or tracks, octagon or arc pad surrounds, grid size and track width.

See pages 80 and 104 of the *Advanced PCB Reference*.

Edit Polygon Vertices

Polygon plane (or copper pour) vertices can be graphically edited without re-defining other polygon attributes. Vertices (or "corners") can be moved, added or deleted after the polygon is placed.

When the vertices are moved, added or deleted, the Place Polygon Plane dialog box opens. Other polygon attributes can be changed at this time, if desired. The polygon will then be regenerated.

See page 80 of the *Advanced PCB Reference* for additional details.

Converting selections to area fills

The Edit-Change-Convert Selection to Fills command recalculates the area of selected tracks and/or fills and converts that selection to rectangular fills. Although this command can simplify large polygons constructed from tracks, small or irregularly shaped areas may convert less efficiently, resulting in an array of small overlapping fills.

This command can be used at any time and can be applied to any selection. All tracks will be converted to fills, but only "touching" primitives will be converted into larger solid fills.

Nets and netlists

A key element of electronics design automation is the ability of schematic capture and PCB layout systems to recognize the connections within a circuit. This concept – referred to as *connectivity* – is used at several levels during the Advanced PCB design process. Two types of connectivity are employed under Advanced PCB:

Logical connectivity

Logical connectivity is the connection information derived from netlisted information. For example, the pin-to-pin connections in your schematic are transferable, via a netlist, to the PCB editor. In Advanced PCB, these connections are stored and used during the layout process.

Physical connectivity

Advanced PCB uses the physical geometry of your layout to perform connectivity-based operations. One example is the ability to select a connection or net, whether or not a netlist is loaded. Physical connectivity can be used to generate a netlist from a routed board. It also allows track-to-pad connections to be maintained as components are moved, etc.

Some operations use both logical and physical connectivity. When you perform a Design Rule Check, the system compares the physical connections in the layout with the node information from the netlist. DRC also uses the layout geometry to check clearances.

About netlists

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

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

Some netlist formats combine both sets of data in a single description, Others, including Protel, separate the two data into separate netlist sections. The Protel 2.0 version netlist, described below, carries additional information, used to define design rules for routing specific nets.

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.

➡ Save manually edited netlists in “un-formatted” or “text only” form, as hidden “control characters” can render the netlist unreadable by Advanced PCB.

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 (pattern). An identical description (or Type) will be required in the PCB library.
74LS138	Part Type, (or value).
(blank)	These 3 lines are not used.
(blank)	
(blank)	
]	Marks the end of the component description.

The component descriptions are followed by 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 components must be an exact match.
J21-1	Indicates the second node in the net.
U5-5	Another 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 supply design rules for autorouting. 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

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.

➡ Package description (type or footprint) names and pin numbers must have exact matches in the Advanced PCB library for all components and connections in the netlist. Advanced PCB accepts either a dash (-) or comma (,) delimiter between the designator and pin number (U1-16 or U1,16).

Loading netlists

To load a Protel format netlist, choose the Netlist Load command. If you have a netlist already loaded, you will be prompted “Netlist Already Loaded, Confirm Load.”

Netlists are loaded in a single step operation. As the netlist loads, component names are displayed on the Status line, followed by all net names. Components in the netlist will be extracted from the current libraries, assigned designators and placed directly into the active document window. The placed components are stacked, with the component reference points directly under the current cursor position. All connections will be displayed as straight lines on the Ratsnest layer.

During loading, the active window is checked for existing nets and components. If a netlist component designator already exists in the workspace, then that component will not be loaded.

After the components are loaded all nets are loaded and optimized using the (current default) strategy – i.e. shortest overall connection distance. Optimization orders connections in the net for various connection strategies. Nets can be individually or globally re-optimized using a variety of strategies. For an explanation of netlist optimization, see the section Optimizing connections, featured below.

Once loaded, a Netlist Load Status dialog box is displayed, reporting.

Number of components loaded.

Missing patterns – netlist components which cannot be matched to the PCB patterns in the current libraries.

Existing components – previously placed components whose designators match one or more netlist components.

Nets loaded – the total number of nets in the netlist.

Errors, such as missing pins or missing components in the loaded netlist.

Click Make Report File to generate a text file, listing all of the information regarding the missing pins, components etc. or click OK, or press ENTER to close the dialog box and return to the workspace.

Missing Components or Pins

Advanced PCB reports missing component patterns or missing pins when two factors are present in the netlist:

1. One or more Package Descriptions (or Type) is missing from schematic component information in the netlist, or the package in the schematic does not match any Advanced PCB library component. The names of missing components and pins will be listed in a Netlist Report file, if desired.

It may be necessary to re-edit the schematic or netlist to include the Type information, or additional Advanced PCB library components may be created to match any unique descriptions in the netlist.

2. If all components are present but pins are reported missing, the cause is usually that the schematic package's pin numbering differs from Advanced PCB.

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," as illustrated – each of which must be assigned to the correct pin number in

Advanced PCB. Capacitors are a similar case, with pins often named “A” and “K” in the schematic.

One solution is to leave the Schematic pin designations as “E,” “B” and “C” and then place components on the PCB. Use Edit Pad to change the pad designator to “E,” “B” and “C.” If you have a lot of pin outs in the same orientation, you may want to make a special version of the component in the library using the correct pin identifiers.

OrCAD® netlists

OrCAD SDT allows you to generate a “Tango” format netlist. This format is fully compatible with Advanced PCB. Translation of OrCAD (component) package descriptions to the Advanced PCB equivalents is also supported. Over 6000 standard OrCAD component type names are stored in a file called PFW.XRF, which lists the OrCAD name and Protel package matches. This simple text format file is user editable.

See page 215 of the *Advanced PCB Reference* for details.

Forward annotation

When an updated netlist is loaded over a current design, the new netlist and current (attached) netlist are compared. Advanced PCB will update the physical layout to reflect any changes. Changes that are supported include:

Adding or deleting a net.

Adding or deleting a node to an existing net.

Renaming a net.

Adding or deleting a component.

Changing a component footprint pattern.

Renaming a component

Joining two or more nets into a single net.

Splitting a net into two or more new nets.

If new components have been added to the netlist, these will be loaded into the workspace, if each footprint (or pattern) is available in the current library. Otherwise new components will be reported as missing pattern.

Clearing a loaded netlist

The Netlist-Clear command can be used to free memory and reduce the PCB file size when you have finished routing and design rule checking. Any PCB file saved when a netlist is loaded, will be saved with the netlist attached to the PCB data. Clearing the netlist allows you to save a version of the PCB without netlist information attached. If you intend to make future alterations to the design, either retain the attached netlist or save a copy of the netlist (in its final form) as a separate (.NET) file.

Optimizing connections

When using auto placement and autorouting, each net can be assigned its own optimization strategy. Optimization is the method used to determine the way connections in a given net are ordered. The default method is “Shortest ” (connection), but can be any of the other four methods available. Optimization can be performed on a single net, all loaded nets or, all nets connected to a selected component. Three options are available:

Net

The current default optimization will be applied to the net and you will be returned to the workspace.

On Component

The current default optimization will be applied to all nets connected to this component and you will be returned to the workspace.

All

The current default optimization will be applied to all loaded nets and you will be returned to the workspace.

Optimization methods are assigned individually for each net (Netlist-Edit Net command). These include:

Shortest

This method looks for the set of connections (with any topology) that connects all nodes using the shortest overall connection distance.

X Bias

This method looks for the set of connections (with any topology) that connects all the nodes together, preferring horizontal shortness to vertical shortness by a factor of 5:1. Use this method to force routing strictly in the horizontal direction.

Y Bias

This method looks for the set of connections (with any topology) that connects all the nodes together, preferring vertical shortness to horizontal shortness by a factor of 5:1. Use this method to force routing strictly in the vertical direction.

Daisy Chain

This method uses the order of the nodes in the net (from the netlist file) and connects them as one wire with the nodes strung along it, listed in same order as the nodes in the netlist. The netlist can be manually edited to change the node order within the net. Each node will have two connections to it, except the first and last which will have only one. No connection length minimization is performed. This option may be used for highly critical ECL designs. Nodes labeled Source and Terminal are used to define the end of the “chain.”

Minimum Daisy Chain

Using this method, the connections in the net will appear as nodes strung along one wire which threads its way around the board. Each node will have two connections to it, except the first and last which will have only one.

Minimization is performed, within the limits of the daisy chain topology, to reduce the total connection length. Nodes labeled Source and Terminal are used to define the end of the "chain."

Start End Daisy Chain

Using this method, the connections in the net will appear as nodes strung along one wire which threads its around the board. Each node will have two connections to it, except the first and last which will have only one. Minimization is performed, within the limits of the daisy chain topology, to reduce the total connection length. The first and last nodes in the net (from the order of nodes in the original netlist file) will be the two end nodes in the daisy chain. Use this option for nets which have termination components on the ends of the nets, high speed ECL boards for example. Nodes labeled Source and Terminal are used to define the end of the "chain."

Star Point

Using this method, each node is connected directly to the node labeled Source (Netlist-Edit-Net command).

Global options can be used to extend optimization method changes to other nets in the layout.

Showing connections

Un-routed connections are displayed as straight-line segments collectively referred to as the "ratsnest." You can display or hide a single connection, all of the connections associated with a net, all of the connections on a component or all of the connections on the board. A special Rats Nest layer is used for display color assignment for connections.

➡ The Options Layers dialogue box can be used to turn the Connection layer display "on" or "off." If "off," then no connections will be shown, independently of any attributes stored for each connection.

Net

Displays the connections associated with a net. When you choose this option, a Net Name dialog box opens. Type in the net name or *?* and click OK or press ENTER to list all loaded nets. Use the cursor or arrow keys to select a net, then double-click to display the connections.

On Component

Displays the connections associated with a selected component When you choose this option, you will be prompted "Select Component." Position the cursor and press ENTER or LEFT MOUSE to display the component connection. If components are overlapped, a Select Component dialog box will open listing those components.

All

Displays all currently loaded (un-routed) connections.

Hiding connections

With all connections showing, it may be difficult to see individual nets or connections. The Netlist Hide Connections (shortcut: n, h) options provide the same options as Netlist Show Connections (above) allowing you to selectively remove connections from the display. One extra option is provided:

Connection

Hides a single connection. When you choose this option, all board connections are listed in a Connections dialog box.

➡ When you are moving components during initial placement, the ratsnest is a good guide of the quality of the placement. If you are not using power and ground planes then hide the power and ground nets. Since they go to all components, they have no value in deciding on the placement.

Editing a net

The Edit Net command allows you to change the editable attributes of any net that is currently loaded. Net attributes include:

Net Name

Net name strings can be any combination of up to 20 alphanumeric characters. Spaces are not allowed in net names. If you supply a net name of more than 20 characters, the overflow will be ignored.

Routing Track Width

Sets the routing width of the tracks used to autoroute the net. If “0” (default), then the track width setting in the Router Setup dialogue box will be used instead. If set to anything other than “0” then this value will be used for autorouting the net.

Routing Via Diameter

Sets the routing via diameter used to route the net. If “0” (default), then the via size in the Router Setup dialogue box will be used instead. If set to anything other than “0” then this value will be used for routing the net.

Optimize Method

The optimization methods described above can be changed for individual nets. See Optimizing Nets (above).

Routing Priority

Chooses one of five priority levels for autorouting the net. The highest priority is routed first, lowest is routed last.

Class number

Nets can be assigned to a “class” for routing by Advanced SB Route. These nets will be routed together, using a single strategy.

Source node

Designates a component-pin name as the source node for the net. This information is used to order the connections when optimizing Daisy Chain and Star Point nets.

Terminal node

Designates the terminal component-pin name for the net. This information is used to order the connections when optimizing Daisy Chain and Star Point nets.

Globally editing loaded nets

Net attributes (other than Net Name) 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.

Indentifying nets

Choose Netlist-Identify (shortcut: n, i) to identify the net associated with a physical connection. Position the cursor over the ratsnest connection and press ENTER or LEFT MOUSE.

The name of the net will be displayed.

Finding the net length

The Netlist-Length command can be used to find the total Manhattan (x + y) distance of all connections on the printed circuit board. This feature provides feedback, when moving components around, to find the optimum placement for minimizing routing distance.

If you have changed the layout since the last optimization, re-optimize the nets before checking the length – to update the internal connection list used to calculate the connection distance.

Changing net connections

Net connections can be interactively changed from inside Advanced PCB. The user can graphically add net nodes, delete net nodes and add new nets to an existing netlist.

Adding netlist nodes

The Netlist-Add Nodes command (see page 138 of the *Advanced PCB Reference*) is used to graphically add nodes to an existing net by first choosing a net, then clicking on component (or free) pads in the workspace to add new nodes. Any existing connections to the pad will be removed (and the internal netlist will be dynamically updated) as nodes are added. Each new node is automatically re-optimized, based on the method assigned to its new net.

Removing netlist nodes

The Netlist-Delete Nodes command (see page 139 of the *Advanced PCB Reference*) is used to graphically remove existing nodes from a designated net. Once a net is chosen, the user can delete nodes by clicking on individual pads. The internal netlist will be dynamically updated as nodes are removed.

Adding new nets to the current netlist

The Netlist-Add Nets command (see page 140 of the *Advanced PCB Reference*) is used to add a net to the current loaded netlist. Once a new net is added, the user can then assign pads to the net with the Netlist-Add Nodes command, described above.

Engineering Change Orders

As you make changes to the PCB, these are written to a special text file <filename>.ECO (if enabled under the Options-Preferences command). 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.

The Netlist-Run ECO File command uses an .ECO file to update the layout and the internal netlist is also updated to reflect connectivity changes.

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

See page 140 of the *Advanced PCB Reference* for additional details about this command.

Generating a netlist

To generate a netlist using the physical geometry of a routed board, independent of the netlist loaded, choose the Netlist Generate command (shortcut: n, g).

This netlist file will then contain a complete netlist (components and connections) for the current layout. This netlist will be generated independently of any loaded netlist. The generated netlist represents the physical connections that have been made using tracks, vias, fills, arcs etc. Nets will be automatically assigned names: Net1, Net2, Net3, etc.

You can use this method to get a netlist from a manually routed file (for later use with an autorouter, for example).

Design Rule Checks

Design rule checking validates both physical and logical layout integrity prior to artwork generation.

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, such as missing connections. Because the Advanced PCB design system allows you to make physical changes to the layout at any time and without reference to a netlist or design rule settings – when performing a “cleanup” for example – it is particularly imported that you always perform a design rule check prior to generating final artwork.

Component Clearance design rules are set in the Auto Placement dialog box. Components that violate these rules will be reported, as will any overlapping components.

The DRC report

An extract from a DRC report follows, showing some of the possible error and warning messages:

Net C9.2_U9.11 Broken Into 2 Sub-Nets

Sub-Net 1

C9-2

Sub-Net 2

U9-11 R4-1

Component Missing From PCB : C10

Extra Pin On Net CPUCLK : U3-22

Extra Pin On Net CPUCLK : U3-26

Clearance Error On Net : CPUCLK

Track (4225,3325 4299,3576) Bottom Layer

Track (4325,3425 4325,3625) Bottom Layer

Note that as much information as possible is provided in the interest of helping the user locate and correct possible problems in the layout. For example, warnings pertaining to the netlist include the net name. Clearance errors include the x and y coordinates and layer where the violation is detected.

➡ Disable the Broken Net Details option when checking a partially routed board. This will speed up the checking process and greatly shorten the report.

See page 142 of the *Advanced PCB Reference* for additional details about running DRC options.

Setting clearances

Note that clearances are additive for any two primitives. For example, the settings above will result in a 10 mil "air gap" between any two items.

The Netlist-Clearances command is used to specify the minimum legal clearance or "air gap" for six primitive types: arcs, fills, pads, text strings, tracks and vias. Clearances have two purposes. They define the minimum distance maintained between primitives when running the autorouter or placing polygon planes and they provide the clearance variables for design rule checking the finished layout (as described above).

The clearances in this system are additive – in other words, the settings for two primitives are combined to define their air gap. For example, if you set the track clearance to five mils and the pad clearance to eight mils, then the design rule checker will look for a clearance of thirteen mils between a track and pad. This way you do not have to specify particular pairs of clearances such as track to track clearances, pad to pad clearance, etc., but you can control all possible combinations using these six primitive settings.

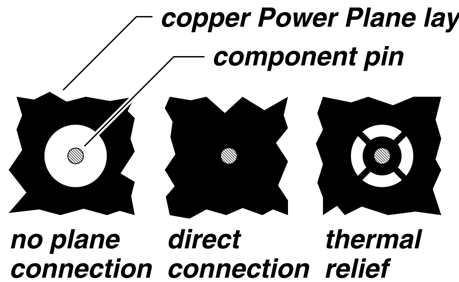
The default clearance for each primitive is 5 mils. The range of legal values is 1.000-1000 mils (or .025–25.0 mm).

Effect of clearances on autorouting

Clearances must be carefully assigned for autorouting, because the ability of the router to run tracks between pads or to replace 90 degree corners with arcs is directly dependent on the interaction of clearance and grid settings. Primitives are placed using the snap grid, so placement during routing is dependent upon the router finding a “legal” grid position which keeps the placed item outside the minimum clearances. It is possible, to have sufficient air gap, yet not to be able to place the primitive, because no grid position falls in the middle of the gap (except for Advanced SB Route, which is not grid-dependent).

Assigning Power Planes

Power planes are special “solid” copper mid-layers. Advanced PCB supports four of these internal power planes. If your design is netlist-based, you can assign a net to each of these layers. As the netlist is loaded all of the pads on the particular net will be connected to the assigned power plane. These pads are flagged to indicate their power plane status.



Effect of pin assignments on solid 'copper' mid planes, when plotted.

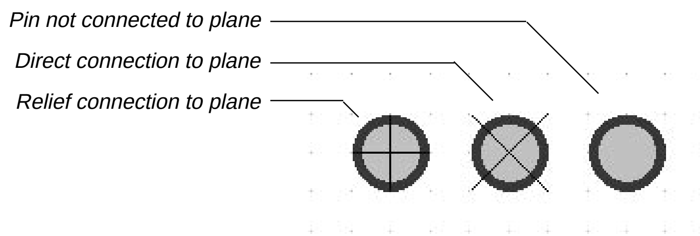
➡ Nets assigned to power planes will not be available as editable connections. If you wish to clear the connections that have been made through these pads, use the Netlist Power Planes command to remove the net's power plane assignment.

Two methods are used to connect the component power pins to the appropriate plane: either a direct connection or a "thermal relief" connection. Thermal reliefs, assigned by default, 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 when plotting the power plane layers.

Pins can be individually assigned to power planes using the Change Pad command at any time. Whenever nets (or individual pins) are disconnected from a plane layer, the ratsnest for that connection is restored.

Pads are marked to indicate connections (direct or by thermal relief) to power planes.

Pin connection status display



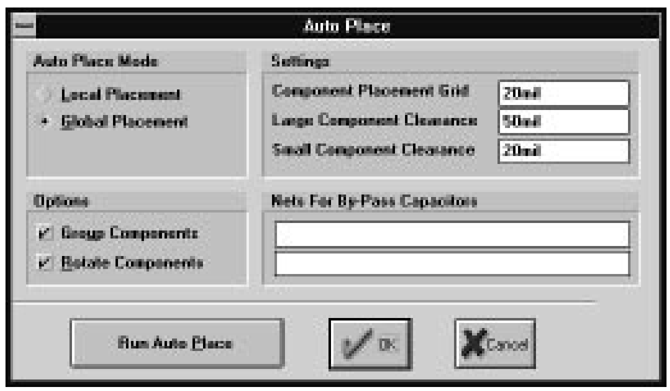
Special support is also provided for connecting SMD power pins to power plane layers. SMD pads that appear on these nets will be automatically “tagged” as connected to the appropriate plane. The autorouter will complete the actual physical connection for these pads by placing a SMD “stringer” – a short track and multilayer pad connection which is relieved-to or direct-connected to the plane layer.

Exporting a new netlist

The Netlist-Export command is used to save the current netlist back out as a new netlist (.NET) file. Any logical changes, such as deleted components, will be reflected in the new version, as will changes made using the Edit Net command. Note that this is different from the Netlist Generate command, in that it doesn’t take account of any of the physical attributes of the printed circuit board, but only the logical connections in the current (loaded) netlist.

Auto placement

Auto Place tools can be used on several levels. When you load a netlist, all components are automatically loaded into the workspace and placed in a stack. This keeps the designer from having to go into the library and move components, one-at-a-time into the document window. Auto Place speeds up the process for finalizing the component positions on the board. This is achieved by moving components around, aligning them, and spreading them out to make routing easier. All of the commands in this menu work on the components that have been selected. In other words, you first select one or more components, then use one of the Auto Place commands to modify its placement.



Options for both "local" and Advanced Place ("global") auto placement are included in the Auto Place dialog box.

The Advanced PCB option includes a simple "local" auto placement system which places netlisted components inside a predefined (or Keep Out) area. This placement/routing board area must be created (on the Keep Out layer) prior to running any of the Auto-Auto Place command options.

Advanced Place is an optional "global" placement system which optimizes placement of large digital layouts.

Local Placement

This mode provides quick, simplified “pre-placement” of your components. Local placement allows you to quickly spread the components in the available space. Local placement can be used prior to locking down connectors or seed components. You can then change over to Global placement to take advantage of the more powerful global strategies and interactive placement tools.

Global Placement

The global placement tools in Advanced Place apply a more sophisticated set of placement strategies to the layout. Where Local placement positions components one-at-a-time, working to simple rules, Global placement considers the total positioning problem as a whole and works to a “best” result. Because the strategy is more detailed, this option will run much more slowly than Local Placement. Global placement will significantly improve autorouter results for large, digital designs. The best results will be generated for boards around the 100 equivalent IC size. Advanced Place can generate placements for these boards which out perform hand placed boards in autorouting benchmarks.

Defining the Keep Out area

A Keep Out area defines the perimeter and “no go” areas for both auto placement and autorouting. This area is defined by placing tracks and fills on a special design layer, the Keep Out layer. Auto placement (and the autorouters) will recognize the limits defined by Keep Out layer primitives.

➡ It is not necessary to define a Keep Out layer prior to using either Local or Global place options – Auto Place will simply use the available board area when no Keep Outs are predefined. See Getting the best results from Auto Place, below, for additional tips and hints about using the Global placement option.

Auto Place options

The Settings and Nets For By-Pass Components options in the Auto Place dialog box pertain to both the basic (Advanced PCB) and Advanced Place systems. Group Components and Rotate Components options work only with Advanced Place.

Settings

Grid and Clearance settings define positioning rules for placement. During placement, the reference point (usually the center of pad 1 or A1) of the component will be placed on the grid. The default grid of 100 mils (or .25 mm) is usually adequate for through hole designs and will work well with 20 or 25 mils routing grids. SMD designs may require finer (or metric) snap grids.

Large components (internally defined as having fourteen or more pins) and small components (thirteen or less pins) can be assigned different minimum clearance values.

The Large Component Clearance defines the clearance between two large components. The Small Component Clearance value defines the clearance between two small components, or between a small and large component.

Nets for By-Pass Capacitors

Up to two nets can be assigned to indicate bypass capacitors (commonly VCC and GND). When identified by their nets, these components will be automatically placed with their associated ICs.

Advanced Place options

Because the Advanced Auto Place routines use a sophisticated series of “global” placement strategies, it will run considerably more slowly than the simple “local” placement tool. Advanced Place is designed for batch (rather then interactive) placement and attacks the placement of the board as a whole. The designer may wish to run these options during lunch, or even overnight if the placement problem is particularly difficult. Because the placement solutions are so effectively optimized, time spent globally placing the board will normally be recovered when routing the board.

Group Components

Advanced Place attempts to arrange components based on the optimization method, grid and clearance selections specified by the user. When the Group Components option is enabled, the system looks for component groupings based on the connections present in the netlist.

Advanced Place will attempt to group components which are heavily connected together first, before it runs the Local or Global place routines. These groups are then treated as if they were individual components.

Rotate Components

When disabled, components are placed in their current orientation. If this option is selected, Auto Place will rotate selected components in increments of 90 degrees for the best fit to the board and connection result, based on the optimization strategy assigned to each net. The default setting is "disabled." It is recommended that you try both options with difficult placements, to see which gives the best result.

The Advanced Place window

The Global (Advanced) placer runs in an independent application window with its own menu bar. When running, both components and the loaded ratsnest are displayed, along with items on the Keep Out layer. The displayed component outlines include the clearances specified under Settings.

Your original PCB document window is not affected, until you choose Update PCB from the Advanced Place File menu. The other File menu commands can be used at any time to save or re-name the placed result. The Display menu allows you to toggle the ratsnest, components and Keep Out layer display on or off.

The placement window Status line displays the elapsed time, pass, number of component moves, and a routing difficulty value, which gives a relative approximation of the improvement in placement, over time. When this value stabilizes, little additional improvement can be expected from the placer and the Global place application will automatically quit and a dialog box will alert the user that placement is finished.

Global Place runs in the background. While running this feature, which can take a considerable amount of time, the user can continue to work on another Advanced PCB document or another Windows application.

Component placement status

The placement tools will only move components whose Place Status is set to “Free to Move.” All components will be “free” when initially loaded from the netlist. Place status only changes when you use the Change Component command and set the Place Status to “Locked in Place.”

Some components, such as connectors or “seed” components (such as processors) should be manually locked into position, early in the layout process. You can use the global change feature to selectively change placement status across your design.

Interactive placement

The Advanced PCB option includes interactive placement tools, which are especially useful for locally optimizing the results generated by Advanced Place.

Spreading components

The Spread Horizontal and Spread Vertical commands can be used to distribute selected components in even horizontal or vertical rows. A manually moved component can then be "locked" into place. You can select a group that includes the locked component and "spread" the other components in an evenly distributed row, using the locked component to fix one end. You can also lock two components at either end of a row, then spread the middle components evenly in a horizontal or vertical array.

Aligning components

The Align Left, Right, Top and Bottom commands are used for lining up a group of components. Selected a group of components, then choose one of the Align commands. You will be prompted to select one component. The rest of the group will align on the selected item.

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.

Expanding or contracting placement

These options spread or squeeze an array of components in increments of one placement grid step (default is 100 mils). Components will expand outward until the minimum clearance to the Keep Out perimeter is encountered or inward until the minimum clearance is encountered. Remember "large" and "small" components can have separate clearance settings.

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, set it to “Locked in Place” (Change Component command) before using the shove command.

Setting the Shove Depth

This command 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.

Getting the best results from Advanced Place

Because of the many user-determined variables, some “trial & error” may be required to find a combination of settings that produces the best result. By experimenting with the available options, you will soon find the strategies best suited for a particular task.

It can be productive to lock-down selected components, such as memory arrays or groups of analog devices, to facilitate auto placement of the balance of the components. Using temporary area fills to define “routing zones” is another good strategy. Remember to remove these fills prior to routing, however. Using Rotate Components makes the placement job more difficult and time consuming – particularly if the design is very dense. Rotating components may also raise manufacturability issues, particularly for boards to be loaded by pick-and-place equipment. Dense designs may also place more successfully with the Group Components option disabled.

Large nets slow down the placer. Another potential strategy for auto placement is to “hide” selected large nets from the placer by creating a temporary version of the netlist, with selected nets removed. Run Advanced Place with the “cut down” netlist, then reload the original and complete placement. You can also temporarily reassign large nets to one or more of the Power Plane layers, thus hiding them from Auto Place.

Remember, automatic placement is a productivity tool – not a replacement for the judgement 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 Auto Place will both speed and ease the design process.

Moving components to a new grid

The Move to Grid command 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 the move to Grid command, select the value 20 and all the component reference points will be moved to the new grid.

The Density command

This command displays a color coded map which represents the connection density of the un-routed board, when a netlist is loaded. This map is intended to help the designer work toward the most routable solution during component placement.

Connection density is a relative measure, with the most dense spots on the board being marked in red and the least dense spots on the board being marked in green. A simple green through yellow to red color scheme is used to indicate the increasingly dense (warmer) areas. A netlist must first be loaded before using this command.

Autorouting

"Routing" applies to both manual and automatic track placement, used to route signals on the PCB. Advanced PCB allows the user to route connections manually, that is, independently of any netlist information. You can lay-out components by hand and manually place tracks and vias to complete a design.

When a netlist is loaded, a series of "from-to" connections is generated and displayed as the visible "ratsnest." The process of converting these displayed connections into the tracks and vias that form an electrical connection, is also referred to as routing. Professional PCB allows you to use this netlist for interactive routing and also allows you to automatically route any two designated pads.

Advanced PCB and Advanced Route include processes for automatically completing netlisted connections – a process called "autorouting."

The successful use of one or more of these routing strategies 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 individual designer.

A basic introduction to autorouting technology will help you to get the most from the autorouter in Advanced PCB.

Advanced PCB currently offers three routing options:

Line probe router

This router is included with the Advanced PCB option. It is a simple router which works efficiently with pattern routes, such as memory arrays.

Advanced Route

This optional router is a rip-up/re-route maze-type router which is capable of producing good quality routes and high completion rates on boards that are primarily digital circuitry with mostly through-hole components or simpler SMD layouts. Advanced Route can route using all 16 Advanced PCB signal layers, with blind/buried vias.

Advanced SB Route

This is an optional shape-based (or gridless) router for Advanced PCB which provides “state-of-the-art” routing for difficult designs including analog, mixed signal and densely packed SMD boards. Advanced SB Route operates in batch mode, with all router setups fully-integrated into the Advanced PCB interface.

Judging Autorouters

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 etc. 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 bussing structures, may benefit from manual routing of the bussing on one or two layers, and then autorouting the remaining random logic. If the board has high speed ECL circuits, then the trace length and termination requirements will put it beyond the reach of most autorouters. Power supply and power control boards, particularly those that use switching technology, need attention to star pointing and overall track shape, length and width.

The class of boards that can be (fully) autorouted successfully with Advanced PCB/Advanced Route, include digital PCBs with through holes and two layers (up to 0.8 sq. in per IC) or four layers (up to 0.6 sq in per IC), 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. Advanced SB Route can be used to route virtually any design, with higher completion rates and superior manufacturability to any other PC-based router.

Layer biasing

In Advanced PCB, you can assign the routing direction for each layer: horizontal, vertical or both (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.

Multilayer board manufacturing technology enables you to route connections across two or more layers, separated by a dielectric (usually fiberglass). 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.

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:

Routers that work one connection at a time until they either finish or run out of options.

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 from one end of the proposed connection, over the board, 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 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."

Shape-based (or "gridless") routers

This class of routers is the most sophisticated type currently available. Unlike the previously discussed routers, which depend upon grid-defined routing channels to solve individual connections, shape-based routers model primitives as geometric shapes. Routing paths are also treated as shapes. The ability to complete routes is limited only by the actual primitive and path shapes, not by a gridded channel. Advanced SB Route incorporates shape-based technology and is the preferred routing option for complex layouts. This router can route to higher completion rates, on fewer layers and will produce the most "manufacturable" result.

Protel autorouter

The Advanced PCB option includes an autorouter which incorporates line probe and memory (or “pattern”) algorithms. It works with up to 16 signal layers plus power and ground planes with grid sizes of 5.000-100.000 mils. You can control all clearances (air gaps) as well as track and via sizes. You can also specify which passes are used and what layers and directions are used for tracks.

Access to autorouting is provided on two levels, depending upon the routing task:

Choose one of three manual and interactive methods to route an individual connection.

Or, choose one of the automatic routing options to route a single connection, a single net, connections to one or more component(s) or the entire layout.

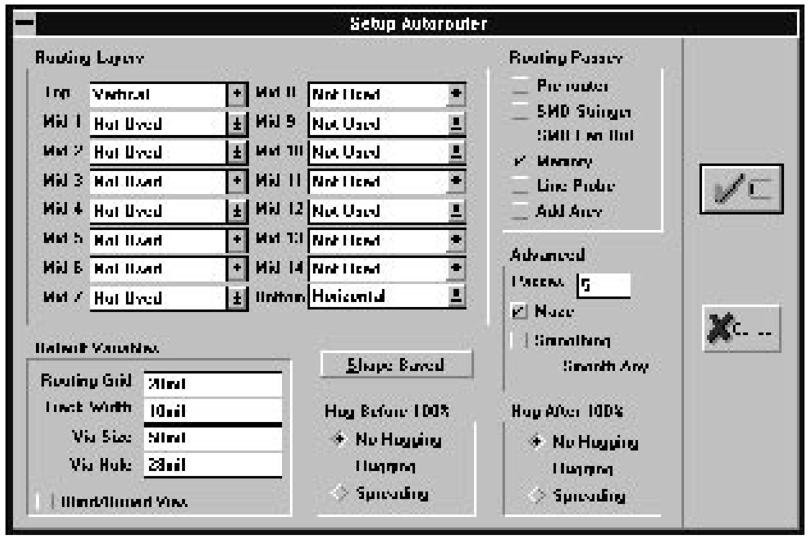
No netlist is required for Pad to Pad autorouting. Other autorouting options, including Auto-Route Manual require a netlist to be loaded.

When you select a routing command, the autorouter attempts to convert the netlisted connection information into a physical arrangement of tracks and vias that constitute a completed layout. This may take a few moments, several minutes or hours, depending upon the routing options, the complexity and number of nets, the design rules and physical characteristics of the layout. The routing pass will be displayed on the Status line, along with a count of the connections completed.

Setting up the router

The Netlist-Clearances command (shortcut: n, r) is used to define the minimum clearances (air gap) applied to autorouted connections. These clearances are additive – selecting a 5 mils clearance for pads and an 8 mils clearance for tracks will yield a total pad-to-track gap of 13 mils.

Using these settings, two tracks would be separated by 16 mils.



Access to Advanced PCB's standard autorouter, Advanced Route and Advanced SB Route features is provided through the Setup Autorouter dialog box.

The Auto-Setup Autorouter command (shortcut: a, t) controls the routing layers, passes, options and other routing variables. These options include:

Routing layers

Sixteen layers are available for routing. This option assigns the layer bias for each routing layer. Four options 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.

No Preference

Router will place tracks either horizontally or vertically. If only one active routing layer is available, router assumes a single sided board with no vias.

If you want to route multilayer without vias, route each of the layers separately using the No Preference option. Then, when running Smoothing, route multiple layers. The smoother will not add vias to the total count.

Routing Passes

These passes can be run independently, in any order. When more than one pass is enabled, the passes run in the order that they appear in the dialog box: Pre-router, SMD, Memory and so on.

Pre-router

Enabling this option protects any hand routed connections (pre-routes) that you make prior to running the autorouter. This pass checks all the connections and makes sure that the router will only attempt to route any that have not been previously completed. When using the Manual Route command, completed connections are removed from the (internal) netlist and this option is not needed to protect these routes. Default is “off.”

SMD Stringers

When using power and ground planes with SMD components, this option will route those pins that need to be connected to the planes (with a short track segment, through hole pad and thermal relief connection to the plane). Default is “off.”

SMD Fan Out

Special pass to preserve routing room around SMD components by routing each pad in a “fan out” configuration.

Memory Router

A fast heuristic pattern router to put in the classic wave structure for memory busses. Default is “on.”

Line Probe

This option enables a number of line probe and pattern routing passes. They use a basic L, C or Z shape, with the separate sections completed by a line probe router. Default is “On.”

Add Arcs

This pass will replace any corners (on one layer) with a 90 degree arc, wherever the current clearance settings (Netlist-Design Rules) permit. Default is “Off.”

Advanced Route

These passes are part of the Advanced Route package. These passes run after the basic router passes are completed. Options include:

Maze Router

This 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 is primarily used for increasing completion. It will rip-up and reroute other connections which block its path. Maze routing places a high priority on layer biasing. 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. Sometimes this re-route will result in some fairly obvious backtracking, so it is always best to run at least one smoothing pass after maze routing.

For really difficult boards it is often better not to use the Line Probe router because it isn't able to hug, or share copper as intelligently as the Maze Router. There is a special pass which only smooths Line Probe routes prior to running the Maze router. This makes the line probe result "conform" to the maze model as much as possible. To disable this pass either don't use line probes, or turn hugging off.

Smoothing

Smoothing checks the board for vias that can be removed by swapping tracks from one side of the board to the other; minimizes the number of track segments; cleans-up parallel tracks on the same net; etc. 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. The smoother will not smooth pre-routes, i.e. manually placed tracks. It will smooth manually routed tracks. If you want the smoother to be able to process a connection use the Auto-Route Manual command, if you want the smoother to leave it the way you placed it, use Place Track. Running multiple smoothing passes will continue to improve the result over several passes. You can use the Router Log to see the progress made during each pass. Smoothing may take a long time on a complex board.

Smooth Any

Smooth any enables the Smoothing passes to work on pre-routes (e.g. manually routed connections) as well as routes completed by the autorouter.

Passes

The Maze and Smoothing passes can be run multiple times – in an attempt to improve the finished result. The board will be routed as high as possible using the maze route then all the connections that will have been routed will be re-routed using the smoothing pass to bring the via and track count down. With each pass, the process is repeated again to improve the completion of those last difficult connections. Maze and Smoothing passes can require considerable time to complete. The Maze and Default setting is “5.”

Default Variables

These values are used by the router. They are independent of the Current (default) settings.

Grid Size

This grid will be used to route the board and will set the location of tracks, vias and arcs placed during autorouting. For all the routing passes, decreasing the grid size will improve the completion rate, but slow the router and increase the memory usage during the Maze router passes. The default value is 25 mils. The range is 5.000 – 100.000 mils.

➡ The placement grid should be an even multiple of the routing grid to prevent off-grid pins that make autorouting more difficult. For example, components placed on a 50 mils grid can be successfully routed using a 50, 25, 10 or 5 mils routing grid. The Auto-Move To Grid command (shortcut: a, g) can be used to SHIFT all placed components onto the routing grid before running the autorouter.

Advanced Route calculates the routing area independent of the origin, so it is not necessary to re-set the origin prior to running this option.

Track Width

This setting controls the width of any tracks, straight or curved, put in by the autorouter. Enter any value from 1 to 9999.999 mils. This value should be set in conjunction with the grid size and clearances to allow successful routing. Default is 12 mils.

Via Size

The diameter of any vias place by the autorouter. Enter any value from 1 to 500.000 mils. Default is 50 mils

Via Hole

The hole diameter of any vias place by the autorouter. Enter any value from 1 to 500.000 mils. Default is 28 mils.

➡ These variables will not be applied to any nets overridden by those track or via settings predefined, using the Netlist-Edit Net command (shortcut: e, n or n, e).

Blind/Buried Vias

If you select this option, the router will try to route between layer pairs where valid blind or buried via placements can be made. This will improve the density of the board when four or more signal layers are used.

It is important that you use the right layer pairs to take maximum advantage of the blind via capability. For example, if you are routing with four signal layers, select the Top layer, Mid layer-1, Mid layer-14 and Bottom layer. Valid blind layer pairs are: Top layer/Mid layer-1 and Mid layer-14/Bottom layer. This gives three via combinations: Top layer to Mid layer-1, Mid layer-14 to Bottom layer and Top layer to Bottom layer which goes through all of the signal layers on board.

Trace hugging

Hugging is a router feature that attempts to place tracks (or traces) adjacent to existing track in the interest of preserving as much of the vacant board space as possible for future track placement. However, once the board has achieved 100 percent there is no advantage in this and the other option of NO Hugging or Spreading can be used. The opposite of hugging is spreading. Spreading will try to equalize the distance between all of the tracks, thereby making maximum use of the available space on the board for a more “manufacturable” result.

Advanced PCB allows you to combine hugging and spreading. These strategies are applied during the Maze and Smoothing passes.

Hug Before 100%

This applies hugging or spreading prior to the initial completion of 100% of all routes. The NO Hugging setting disables this feature.

Hug After 100%

This setting works after the initial routing has been completed to the 100% level.

Track hugging should generally increase the route completion on difficult boards. Because closely hugging tracks may make manufacturing more difficult, it is usually wise to run the Smoothing passes with Spreading after the board is completely routed. If you are routing more than two layers with Hugging activated, memory consumption can be very high.

See page 161 in the *Advanced PCB Reference* for more information about these options.

Interactive routing

The Auto-Manual Route command is used to interactively route individual netlisted connections.

When this command is activated, the cursor will jump to the nearest end of the selected connection. As you move the cursor, a highlighted track will rubberband from the starting point.

➡ The grid, zoom and toggle layer shortcuts (press * to toggle through the active signal layers) are available at any time while routing interactively. Press SPACEBAR to change the track mode between Any Angle, 90/90, 90/45, or Curved.

As you route the connection, the ratsnest will be maintained from the current cursor position to the target pad.

Pressing * will change signal layers and automatically place a via (when the Auto Via option is selected from the Options-Preferences dialog box). Pressing ESC will terminate the current track segment. Note that the ratsnest is maintained from the end of the last segment placed to the target pad and that the prompt "Select Connection" is again displayed on the Status line. You can go back at any time and resume routing the partially completed connection.

While routing tracks, you may need to occasionally press END (or choose Zoom-Redraw) to cleanup the display. If you are unhappy with the result of your route you can use Undo to remove the route and restore the ratsnest. You can go back at any time and use the Auto-Un-Route command to remove a route and restore the ratsnest connection.

Auto Route

The Auto-Auto Route commands are used to automatically route an entire layout, a single connection, a selected set of connections or a single pad-to-pad connection without a netlist.

Auto Route commands include:

All

Routes all netlisted connections on the board using the passes that are currently designated in the Autorouter Setup dialog box. Clearances are maintained under the design rules set in the Netlist-Clearances dialog box.

Net

Routes all connections within a single net. Type the net name, or type *?* to select from a list. The Memory and Line Probe passes will be used to route the net (if enabled). The

Autorouter dialog box will not be displayed during the route.

Connection

Autoroutes a single netlisted connection. When prompted, select a connection. The Memory and Line Probe passes will be used to route these connections (if enabled). The balance of any net will be left un-routed. The Autorouter dialog box will not be displayed during the route.

On Component

Routes the connections to a particular component. Select a component. The Memory and Line Probe passes will be used to route these connections (if enabled). The balance of any net will be left un-routed. This option routes the individual connections only, not the nets. The Autorouter dialog box will not be displayed during the route.

➡ You can un-route a component (Auto-Un-Route-On Component command, below), move the component to a new location (including changes in orientation) and then re-route all of the connections using the Auto-Route-On Component command.

Selected Components

Routes the connections (not nets) to a group of selected components. The Memory and Line Probe passes will be used to route these connections (if enabled). The balance of any net will be left un-routed.

Pad To Pad

Routes any pad pair in the layout, without using a netlist. The Memory and Line Probe passes will be used to route these connections (if enabled).

Advanced Route Connections

Routes a single connection using the Advanced Route (Maze, Cleanup) passes. All Advanced Route features work with this command. The balance of any net will be left un-routed.

➡ While autorouting you can zoom or pan the active window to view the progress. You cannot access other commands in the current Advanced PCB application. However, you can Minimize the current application (to an icon) and launch a second Advanced PCB – to work on another project.

When the router is finished a dialogue box will pop up over the top of which ever application you are working on informing you of the route being finished.

See page 158 in the *Advanced PCB Reference* for additional details about these options.

Un-Route

The Auto-Un-Route commands remove routed primitives (tracks and vias) and restore the ratsnests. These commands are similar to the Auto-Auto Route commands and include: All, Net, Connection, On Component, Track, Selected Components and Selection.

See page 166 in the *Advanced PCB Reference* for more information about these commands.

Autorouting models

To get the most from Advanced Route, 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 net (any two primitives)

Grid: 25 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.

Single Density SMD (50 mils pad centers)

Track: 8 mils

Via: 40 mils

Clearances 8 mils net (4 mils track plus 4 mils pad, etc.)

Grid: 25 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 fairly standard SMD board.

Double Density SMD (50 mils pad centers)

Track: 5 mils

Via: 30 mils

Clearances 5 mils

Grid: 10 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.

Triple Density 1

Track: 8 mils

Via: 42 mils

Clearances 8 mils net

Grid: 16.67 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

Track: 6 mils

Via: 30 mils

Clearances 6 mils net

Grid: 12.5 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.

Netlists and autorouting

A netlist contains all the connections in a circuit, but includes no information about the order in which nets are to be completed. This is why the user is allowed to select an optimization strategy for the whole board or for selected nets, for example shortest connection distance. Advanced PCB also allows you to prioritize nets so that the most critical connections can be routed first.

When you load a netlist, an internal “connection list” is generated. This connection list drives the ratsnest display. The connection list includes all pad pairs, in the order of connection, according to the optimization strategy. This connection list will be re-ordered each time the optimization is changed Re-ordering is done on a net-by-net basis, but only for those nets which have remained completely un-routed since the netlist was loaded.

This connection list remains valid as long as autorouting options are used exclusively, rather than by placing tracks and pads manually (not to be confused with using the Manual Route mode).

When you save a file, the current set of connection lists, including the route status of each connection is saved as part of the file. The netlist is also “attached” to the PCB file, when saved. This allows you to re-load the file and continue routing without re-loading the original (source) netlist or re-optimizing the connection list.

Getting the best result from the autorouter

The designer will nearly always be able to “improve” in some way, 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, improves the designer’s overall productivity.

While the designer 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 the designer 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 grid selected. If you halve the grid, you quadruple the number of potential solutions for each route – however, routing time increases proportionally. Off-grid pads, component layout, pre-routed connections, variables and clearance settings, etc., will also profoundly impact the completion rate and route quality obtained. To improve Router performance:

1. Use the Auto-Move to Grid command to locate all pins on the routing grid. Make sure that you have nominated a grid that “fits” all components used, including SMDs. The minimum placement grid is 5 mils (or .125 mm). Set up the router to route on a grid which is equal to, or a multiple of the placement grid.

2. Minimize the connection distances of discretes, such as resistors. The visible “ratsnest” and Density display are good aids in positioning components to minimize congested routing channels. Use the Netlist-Length command 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. 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.

Routing tips and hints

Memory available for autorouting should be physical – not virtual – because virtual memory (accessing the hard disk) is impractically slow due to the extreme number of computations required to complete each connection. Routing more than two layers with the Hugging or Spreading option activated will also demand a great deal of memory. If you find that routing is proceeding very slowly, you may need additional RAM to successfully use these options. Turning off hugging will halve the memory requirement when maze routing.

Use this formula to estimate the memory requirements for maze routes:

(x size / grid size) x (y size / grid size) x (route layers x 2) = 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 (2X2) = 154 KB

Remember, no hugging will halve this requirement. Or, for a 12" x 14" board, 20 mils grid, top bottom and 2 mid layers:

(12000/20) x (14000/20) x (4x2) = 3.4 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 is less than or equal to the routing grid; via size plus 2 times the via clearance is less than or equal to 3 times the routing grid.

If you are using blind/buried vias it is very important that you select the layers for routing carefully. Make sure that you choose “matching” pairs and that one of each pair is horizontal, and one is vertical (either or both may also be no bias). For example for a 4 layer board you would route on Top, Mid-1, Mid-14, & Bottom – not Top, Mid-1, Mid-2 & Bottom.

Moving components (and therefore most of the pads) to the routing grid will significantly improve the performance of the maze 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.

Advanced PCB produces a text file report of autorouter results, including a listing of any un-routed connections.

Generating PCB artwork

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 directory 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 Postscript compatible 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. Some users may find printer output suitable for producing simple prototype boards, providing the level of detail is fairly coarse. This normally requires a print scale of at least 200%. 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. See the end of this section for additional details.

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. Enlargements of these masks are often required by manufacturers and can be specified (in mils or mm) under the File-Print Setup, Pen Plot Setup, or Gerber Setup options. Solder masks are normally plotted in reverse, for efficiency. This masks can include an enlargement (or expansion) at each pad to accommodate manufacturing tolerances.

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 normally plotted in reverse, for efficiency. This masks can include an enlargement (or expansion) at each pad to accommodate manufacturing tolerances.

Drill Guide

All holes are plotted, using a single pad size. These will normally be printed with holes to provide a visual target for drilling. Specify the hole under the Guide Hole Size option in the Output Options dialog box.

DRC Errors

This layer shows highlighted areas where clearance violations have been detected by the Design Rule Check feature. This layer can be printed as an aid to trouble-shooting the layout.

Drill Drawing

This layer includes special symbols, indicating NC drill hole positions. A legend is included in the plot which lists a hole count, by size.

Top layer

This is the "component side" layer. All the tracks, arcs, fills and edge connector pads that have been placed on this layer are normally included when printing, as well as pads and vias.

Mid layers (1–14)

There are 14 mid-layers available in Advanced PCB. Tracks, arcs, fills and text strings are normally included with these layers, along with any other items the designer wishes to include in the artwork.

Bottom layer

Also known as the "solder side" of the PCB. Components can also be placed on this layer. Tracks, arcs, fills and edge connector pads placed on this layer are normally printed, as well as pads and vias.

Top Overlay, Bottom Overlay

Also called the silkscreen layers, these are normally used for component outlines and component text. These layers would normally be printed with pads and vias "off."

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 "hold back" the solid plane. 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 Fab & Assy drawings, showing dimensions, trim marks, datum marks, holes, assembly instructions and other mechanical details of the board. Special options allow the designer to combine one or more of these layers with other layer items when printing.

Generating a print

Advanced PCB printing is handled similarly to other Windows applications. Windows manages the printing (or plotting) process and provides a range of raster and Postscript printer drivers. These range from 9 pin dot matrix printers to high-resolution raster imagesetters.

Advanced PCB will print all the layers (including DRC errors, ratsnest, etc.) that have been selected in the Setup Composite or Final Artwork dialog box, opened when you click the Options button from the Printer Setup dialogs. Tracks, pads, vias, arcs, fills, and text strings will be printed in final (filled) or draft (outline) mode, according to the options in the Printer Setup dialog box.

Some special purpose layers, such as the Drill Drawings, Solder masks or Paste masks cannot be directly edited from inside Advanced PCB, although primitives can be placed in these layers. These layers are derived from other PCB data and include additional detail required for board tooling.

Two print commands are provided: Print-Composite and Print-Final Artwork. See page 26 in the *Advanced PCB*

Reference for step-by-step printing procedures and detailed descriptions for each printing option.

Printer types supported

The available output device options will include those that have been installed using the Windows Control Panel (see your *Microsoft Windows Users Guide* for details).

➡ 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

Show Holes

This option prints or plots pads and vias with the holes showing. For example, you may wish to include target holes for prototype boards that are being manually drilled. Generally, if the artwork is to be used to produce plated-through boards you should leave this option off and rely upon the NC drill file and drill drawing to provide setup information for the NC drill operator.

Fit Board On Page option

The check print or plot will be expanded / contracted to fit on the page size set up for your printer. The plot will be shrunk or expanded to use the available space, keeping the 1:1 aspect ratio.

➡ Make sure that you have set the Portrait/Landscape mode on you printer to best fit you PCB shape when using the Fit Board On Page option.

Print Scale

Most pen plots for PCB tooling are produced at a scale of 2:1 or greater. For example, to generate 200% artwork for photo-reduction set the Print Scale to 2.0

– for check prints, you may wish to leave the scale at the default 1.0000 value.

The X and Y correction factors are available 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.

Printer button

Clicking the Printer button opens the setup dialog box for the target device.

This dialog box is part of the printer driver and is not part of Advanced PCB. For example, a Postscript device setup dialog selects the paper source, the paper size, the orientation of the paper - either portrait or landscape, and a scaling factor.

➡ This dialog 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 Setup dialog box should be used for scaling the output.

The available options vary with the features of the selected device. Windows supports background printing from the Print Manager. The queuing of prints and other options can also be controlled from the Print Manager. See your *Microsoft Windows Users Guide* or printer / plotter documentation for additional information.

➡

If printing or plotting from another computer, plots will need to be generated as files. To do this, open the Control Panel's Printer dialog box and click Configure. Select File from the Ports menu. You will be prompted to name the file when generating the plot.

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 Panelled option or you will see the layers drawn superimposed, if a composite or multi-sheet option is selected.

If using a local printer (one directly connected to the computer) and the application is not occupying the whole screen, the windows Print Manager icon appears at the bottom of the screen.

The print can be interrupted at any time. Choose any key 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.

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.

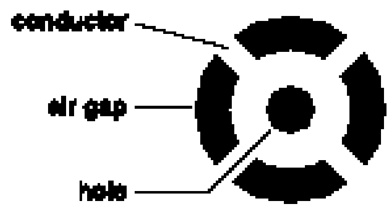
Enlargements in Solder and Paste masks

Options are provided to enlarge Solder mask and Paste mask pads. These allow the designer to expand the pad area in all directions by a set amount, to adjust the artwork for fabrication tolerances.

Thermal reliefs

Thermal reliefs are special shapes plotted on internal power plane layers to indicate a connection at the pin location. The shape of a relief provides isolation between the solid copper plane and the solder point.

These layers, like the solder and paste masks, are always plotted in reverse (as negatives). That is, the reliefs, which represent voids in the solid copper plane are plotted, the “copper” is not plotted.



Thermal relief, plotted in reverse.

Three thermal relief variables can be used to control the exact format of the clearances and air gaps for the reliefs.

The conductor width is the width of each of the entry points to the pad via thermal relief.

The air gap is the width of the line that is used to draw the arc sections of the thermal relief. You can select two or four entry point for the thermal relief.

The radius of the relief is determined by a combination of the pad hole size and any Expansion applied to the plot, described above.

➡ Typical PCB fabrication requirements call for a minimum pin-to-plane clearance of .4 mm (16 mils) for boards less than 305 mm (12 inches) on a side to prevent shorts due to slight misalignment during fabrication. The 28 mils hole defined for a standard 62 mils pad will meet this requirement with the default enlargement. Larger boards require a clearance of at least .5mm (28 mils). These values are provided as a guideline only. Your board manufacturer will give you specific requirements for your design.

Drill Drawing and Drill Guide plots

Drill Drawing plots are generated as a programming aide, for setting-up NC drill equipment. Drill Guide plots are used to verify hole locations in plot layers or are used to generate targets for hand drilling.

Three marker types are provided. The selected marker is printed at each hole location on the Drill Drawing. If you select Characters, each different hole size will be assigned a unique character from A to Z.

A legend will be placed at the edge of the drawing specifying the letter for each hole size. The holes will also be listed in metric.

If you select the Size of Hole String, no legend is used but wherever a hole appears, the string will be placed reflecting the actual size of the hole. So wherever you have a 28 mils hole in a pad then the string “28” will appear on the plot. If you have a metric grid set then the size will be specified in millimeters.

The Graphics Symbol option plots a unique graphic symbol for each of the different hole sizes. A legend, indicating symbols and hole sizes, will be included at the corner of the plot.

Layer 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.

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 long axis as 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 noncritical 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.

Pen plots

Two basic choices confront the user who wishes to produce draft or final artwork on pen plotters:

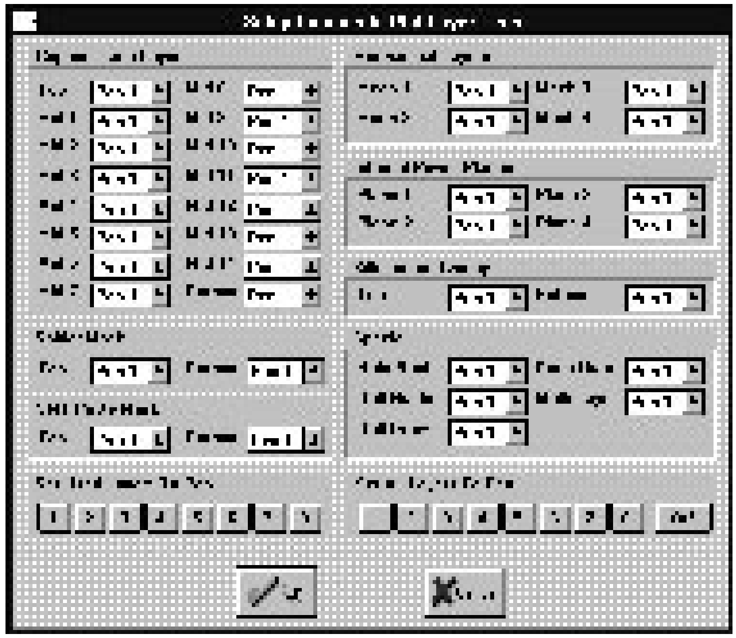
1. You can use the standard Windows plot driver for your device. This means that you will use the Advanced PCB File-Print command to generate your plots. This option is preferred if your device is well-supported by a current Windows device driver. This is most likely to be true for newer devices that use raster, rather than vector plot routines, such as the newer large format "ink jet" type plotters.

2. You can bypass the Windows low-level plot routines by using Advanced PCB's File-Pen Plot command. This option allows direct control over vector-type HP-GL and DM-PL plotters. If you have a conventional pen plotter, this is the only way to produce final quality pen plots from Advanced PCB and draft composite plots will be generated more efficiently and with superior plot quality.

Many plotters can be configured to support either the HP-GL or DM-PL language. See your plotter documentation for more information about available emulation modes. New Windows device drivers are continually being released or updated by both device manufacturers and third parties and many of these are supplied by Microsoft on the current Windows device drivers diskettes. For up-to-date information about plotter driver support under Windows, contact Microsoft or your plotter manufacturer.

➡ 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, open the Control Panel's Printer dialog box and click Configure. Select File from the Ports menu. You will be prompted to name the file when you generate the plot.

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 Printer Options dialog box.

See page 33 of the *Advanced PCB Reference* for detailed descriptions of pen plot options.

Using extensions to identify plot layers

If you have directed the output to a file, and intend to use the services of a plotting bureau, use of standardized extensions to identify each plot layer is highly recommended. Special coded extensions are automatically generated by Advanced PCB when producing print, pen plot or Gerber files. See page 225 of the *Advanced PCB Reference* for details.

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 1 mil (.001 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 (.028").

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 fibre 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 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 this first time, this section explains the relevant RS232-C conventions.

The RS232-C standard defines the signals for bidirectional 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 interfacing and handshaking 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.

Gerber plot generation

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 open and view Gerber files directly from 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 the Plot/Print command. For information about Postscript printing, see the section on Plot/Print options.

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 Gerber file area 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 multi (either flash or stroke).

When targeting a vector plotter, the apertures in the .APT file will correspond to the apertures available on the actual aperture wheel or slide to be used. Raster plotters will use the aperture file to translate draft codes directly into an image "map." In either case, the aperture file (or table) defines the shapes which make up the finished film.

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. Automatic aperture file generation will only be appropriate if you are targeting a raster, rather than vector, plotter. 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.

Editing aperture files

The Library-Apertures command (shortcut: l, a) is used to load, create, or edit the apertures used by the photoplot output routines. When you choose this command, the currently loaded aperture file is 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 lists. 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 command. See Library-Apertures in the *Advanced PCB Reference* for a description of each option.

You can edit the current aperture list globally, similarly to the global editing of your PCB primitives and components. Press Options to choose from the global edit options. When you have defined the changes, click OK to accept your inputs and return to the Apertures dialog box or CANCEL to return to the Apertures dialog box without adding the new definition.

➡ 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.

Tips on aperture use

If targeting a vector plotter, use primitives (track and pad sizes and shapes) for which there is a matching aperture. If the designer familiarizes himself 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.

However, it is not always possible to limit the range of design primitives to the apertures supported by a specific photoplotter. Some system must be provided to allow the successful interpretation of any pad shape or track size or filled area present in the layout. In these cases, Advanced PCB will select from the available apertures (if they fit within set tolerances) and modify the design data base to match the final photoplot.

Generating a photoplot

Setting up a Gerber plot file is similar to setting up pen plots or prints. As when printing or pen plotting, photoplots will be generated with all the physical layers and primitives that are currently displayed. Tracks, pads, vias, arcs, fills, and text strings will always be photoplotted in final mode, irrespective of the Options-Display mode settings.

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.

From this point on, the manual makes three assumptions:

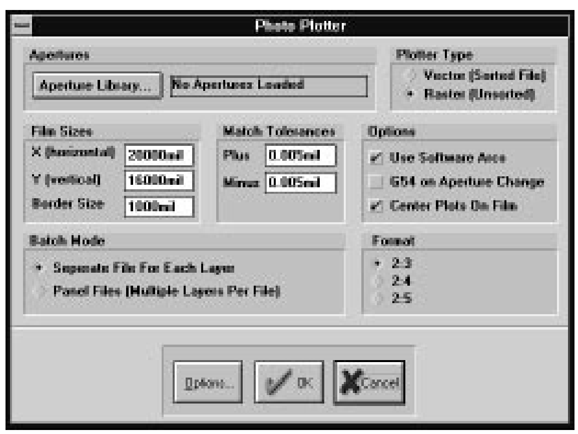
1. That you have targeted a specific photoplotter, and are aware of its output capabilities and file format requirements.

2. That you have created an aperture file for the target plotter or that you intend to use a file generated directly from your PCB file (raster plotters only).

3. That you have pre-determined the contents of each plot layer (pads "on" or "off," etc.).

➡ 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.

See page 18 of the *Advanced PCB Reference* for more information.



Prior to generating Gerber plot files, it is vital that the designer understand the special requirements and limitations of the target photoplotter.

To generate a Gerber output file, choose the File-Gerber Gerber Output command (shortcut: f, g, g) to open the

Photoplotter dialog box. See File-Gerber in the *Advanced PCB Reference* for detailed descriptions of these options.

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 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.

Film Sizes

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.

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 Border setting 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.

Match Tolerances

While generating a photoplot, Advanced PCB looks for an aperture to match each item in the plot layers. If no exact match is available in the current aperture table, the design data base will be changed per the Match Tolerances specified under this option. You can protect the PCB file from these changes by saving the file before plotting. After the plot is complete, you can save the altered file, using a new filename (File-Save As command).

For example, circular 62 mils pads will be efficiently plotted using a 62 mils aperture and then “flashing” a light on to the film. But what happens if the photoplotter does not have a 62 mils aperture?

Matches can 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 change.

Matches that fall outside the specified tolerance are achieved by either “stroking” or “painting” the desired shape where the available apertures permit. However, 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.

Software Arc

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 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

The plot generation process

As the plotting sequence begins, the board is redrawn to fill the whole screen and the prompt Confirm Photoplot PCB is displayed. As you proceed with the plot, all of the tracks and pads disappear from the screen. On the Status line, the message “Building the shape list” is 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. If generating a vector photoplot, the sorting routines runs, and all primitives will be drawn on the screen as they are sorted. The order of display matches the order which they will be plotted. Once sorted, the plot files are generated. At the end of the process, the prompt “Photoplot is Finished” will be displayed.

You can minimize the window while Advanced PCB is photoplotting and the process will run in the background. When finished, the “Photoplot is Finished” dialog box will pop up over the current application.

Loading Gerber Files

There are two ways of loading Gerber files. The File-Gerber In (shortcut: f, g, i) command and Gerber Batch Load (shortcut: f, g, b) command. To load a single Gerber file, or a Gerber panel file, onto the current layer, select Gerber In. The Load File Name dialog box will open, using the default mask .G** to screen for Gerber format files in the current directory.

➡ The Load File dialog box will display any file, whose extension starts with “G” – whether or not the file is Gerber format. To load Gerber files (from other systems), first rename the files, using the .G** convention, if necessary.

To correctly load the file, the correct aperture file (used to generate the file) must be loaded. If no aperture file is loaded, all lines and flashes will come in at a size of 1 mil.

As the Gerber file is loaded, all the primitives will be placed on the current layer on the Advanced PCB workspace. Strokes will be converted to tracks, and flashes will be converted to pads. There are some 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. If a rectangular aperture has been used to draw a stroke that is neither horizontal nor vertical then it will be converted to a standard track. This can produce some errors when loading Gerber files. The only place where non-vertical/horizontal strokes are used is when filling octagonal pads.

To load a series of Gerber files onto their correct layers use the Gerber Batch Load command. This works in a similar way to the Gerber In command, but a number of files will be loaded onto separate layers in a single PCB file. The user is prompted to specify the file name (no extension), and the named Gerber files will be loaded onto the correct layers. For example, the .GTL file will be loaded and placed on the top layer under .GBL file will be loaded and placed on 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.

➡ Gerber Batch Load 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. If the layers were produced without the pads “on” this will not be a problem – but the pads will have to be globally edited to restore their layer assignment, if multi-layer.

The Gerber In command can be used to achieve some things which would not otherwise be possible under Advanced PCB. For example, to produce a Drill Drawing with complex information, generate the Drill Drawing Gerber file, then use Gerber In to load the Drill Drawing onto a Mechanical layers. Add information as you please, then regenerate the "Mechanical" layer file – now a composite of Drill Drawing and Mechanical layer elements.

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 listed in the *Advanced PCB Reference*.

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 duplicated the edited file – free of clearance violations and similar “surprises.” If you are targeting a raster plotter, you can allow Advanced PCB to generate you aperture list directly from the complete layout by using the automatic aperture creation option in the Library-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 files

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 a coordinates and drill tool assignments for each hole in the PCB file.

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

Choose the File-NC Drill command to automatically generate the file, using the current hole attributes for all pads and vias in the active document window. Three files will be generated for each fabrication layer:

Excellon format file with the extension .DRL, DR1, DR2, etc. (this is the binary Excellon format file).

A report file (.DRR) is generated listing the number of holes for each drill and the size in both metric and imperial measurement.

An ASCII version of the binary file with the extension (.TXT, TX1, TX2, etc). You can use this file to verify the contents of the binary version. Use the Windows Notepad to open and view this file.

You should supply all three of these files to your board fabrication house for each fabrication layer, together with a print out of the .DRR file.

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 file (Extension DRL, DR1, etc) is written in the EIA character (binary) format in the EXCELLON language. The data is specified in mils (.001 in) with trailing zero suppressed. Drill tools are numbered from 1 up to 255.

See the *Advanced PCB Reference* pages 41 and 226 for additional details.

Design information

Once your board has been laid-out and routed, Advanced PCB provides additional features for design verification. For example you can use the Info menu commands to check:

System Status

System information: disk space, memory, date and time settings. If running under 386 enhanced mode, free memory refers to the amount of virtual memory available – the sum of your physical memory left and your disk space available on the drive where your swap file is set-up.

Board Status

Board statistics: Dimensions, primitive counts, number of nets and routing completion.

Components on PCB

Lists all components currently placed, includes designator and comment, if any. Double-click inside the menu to generate a text file of the menu contents using the extension .DMP.

Selected Pins

All the pins in a selected net are listed, providing a convenient way to verify the connections within a net. Double-click inside the menu to generate a text file of the menu contents using the extension .DMP.

Nets

Lists all currently loaded nets, by name. Double-click inside the menu to generate a text file of the menu contents using the extension .DMP.

Measure Distance

Allows you to measure distance on any angle in mils or mm inside the current board window. Click to define a starting point and ending point and the distance measured will be displayed. A dialog box displays the straight line and Manhattan (x + y) distance between the two points.

Length of Selection

Generates a measurement of the total length of selected tracks. This feature allows you to measure the connection length of any selected connection or net.

Power Planes

Lists the pins assigned to any of the 4 mid-plane power layers. Double-click inside the menu to generate a text file of the menu contents using the extension .DMP.

Glossary

absolute origin The absolute workspace origin (0,0 coordinates) or lower-left corner of the workspace.

active layer Any Board window layer which has been activated using the Options Setup dialog box or Current Layer command.

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).

arc Circular or semi-circular design elements. Advanced PCB generates arcs of 1 degree resolution.

area fill See fill.

Assembly layer Artwork that is used as a reference for assembling a fabricated PCB. See also Mechanical layer

attributes The characteristics of an item which can be edited, or changed. For example track attributes include width and layer assignment.

Auto Place Options for automatically loading netlisted components from the current library into a predefined board outline and interactively repositioning components for optimum routing.

Autoroute Options for automatically (or interactively) routing the connections in a PCB.

Batch In Command to read multiple Gerber files back into Advanced PCB file format.

Batch Mode Option for choosing multiple print types (or print/plot layers) to be generated in a single operation.

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.

blind via Via assigned exclusively to one of the outer board layer pairs, e.g. Top and Mid-1 or Bottom and Mid-14.

Bottom layer Edit layer for the bottom (or "solder side") of the PCB.

break Conversion of a single track segment into two connected segments.

browse Viewing library items from the Advanced PCB workspace.

buried via Via assigned exclusively to inner layer pairs, i.e. pairs that do not include the Top and Bottom layer.

Check print A composite print or plot of multiple artwork layers, used to verify PCB artwork.

class A number (1-32,000) used to group nets together for routing by Advanced SB Route.

clearance The specified minimum air gap that separates each electrical primitive (signal layer pads, vias, tracks, fills, etc.) from other conductors. Each primitive type can have its own minimum clearance specified (Netlist-Clearances command).

Clear To remove an selection permanently from the workspace. Same effect as Delete. See Also Cut, Copy.

clipboard Reserved memory used to hold Cut or Copy command selections.

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 library. Components consist of one or more multi-layer pads and tracks and/or arcs on the top overlay, which define the component shape.

component text Text that is part of a component display. Component text is created at the time the component is placed. 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.

connection The logical or physical link between any two netlist nodes. Logical connections are indicated by the ratsnest display. Physical connection are completed routes. Advanced PCB allows connections to be partially logical and physical, e.g. partially routed.

copper Any non-etched (conductive) portion of any layer of a printed circuit board. Also refers to Top layer of PCB.

copper sharing Automatic connection of placed primitives to polygon planes associated with the same net.

Copy To add a selection to the clipboard without removing it from the workspace. See also Clear, Cut.

cross probe Special process for viewing corresponding parts or nets from Advanced Schematic-to-Advanced PCB or from PCB-to schematic.

current layer Board window layer displayed on the Title bar. Tracks, text, arcs, or fills will be placed on the current layer only.

cursor The graphic “pointer” or selection tool used to select or position objects in the workspace.

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. Most defaults are stored in a file called PFW.INI.

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 Routines for checking a routed PCB file using a netlist to verify connections and user-defined clearance settings to check for clearance violations.

design rules Clearances, grids, track widths and via types used to interactively or automatically route the PCB.

document User-generated or auxiliary file (Windows terminology).

double-sided Refers to a PCB with tracks on both sides of a single laminate layer.

draft code A code used to identify each aperture in an aperture file or in a Gerber format photoplotter file. Aperture draft codes are typically in the format of a “D” followed by two or three digits.

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 tool assignment for numerically controlled (N/C) drill files.

Drill Guide A special plot, similar to a pad master, which indicates the position of all holes on a PCB.

Excellon Standard format for numeric control (NC) drill equipment used to automatically drill PCB holes.

fabrication layer Artwork used as a reference in fabricating PCB layers. See also Mechanical layer.

fill An area fill, or solid rectangular copper on a PCB layer, used for shielding and/or supply of current. Fills can also be placed for “non electrical” uses, such as defining “no-go” areas on the Keep Out layer.

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 primitives All signal layer tracks, free pads, vias and fills which are not associated with placed components. These items can be selected as a group by Advanced PCB.

free text Text placed in a document window using the String tool or Place Text command. See also component text.

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.

Gerber plot A photoplot stored in a Gerber format file, also used generically to refer to any photoplot.

global change Any change which can be assigned to like attributes of other primitives of the same type in the PCB.

grid A system of visible and invisible points on the workspace used to locate a precise coordinate position.

guide hole Hole used for manual drilling of (typically prototype) PCBs.

hardware arc When plotting, arcs which are created by the plotter, from coordinate, line width and radius information. Some plotters support this option, others depend upon software arc descriptions generated by Advanced PCB.

highlight A special display state that outlines items as an aid to identification or editing. For example, when placing or moving tracks, they are displayed in a highlighted condition. Not to be confused with selection.

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.

integrity checking Process performed on each ASCII (text) format PCB file during load sequence. Integrity checking scans the file for problems and attempts to repair detected problems.

Keep Out layer Special document window layer used to define a perimeter and “no go” areas for auto component placement and autorouting.

lattice Polygon Plane filled with an open gridwork 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.

layer bias Practice of alternating the principal direction for track routing on PCB layer pairs.

layer pair Set of two signal layers assigned to a blind or buried via.

Mechanical layers Any of four layers which can be used for displaying fabrication and assembly details.

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.

Mid layer Any of fourteen layers (other than the Top and Bottom layers) which can be used for routing the connections on a multi-layer PCB.

mils Unit of imperial measure equal to .001 inch.

mm Millimeter - unit of metric measure.

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.

Multi layer The special display layer used for pads and vias that are placed 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.

net An individual (logical) connection of any series two or more netlisted nodes.

netlist A text file which lists all the connections of an electronic circuit. Netlists are used to verify the contents of a design, or to transfer design information between CAD (computer aided design) systems.

origin Location of the 0,0 coordinate in either a PCB file or a plot. Advanced PCB uses the extreme lower left corner of the workspace as the default origin.

orthogonal Drafting standard where lines are constrained to either vertical, horizontal or 45 degree 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, dimensions, etc.

Pad master A special plot type that includes all the pads and vias in the PCB, typically used for drilling all pad and via holes in prototype boards .

pad A design element used to locate and connect tracks to component pins on a PCB, also called a land.

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 when placing or moving selected items.

Paste Mask Special plot of SMD pads, used to define a mask for applying solder paste for “hot re-flow” fabrication.

Polygon Plane Copper solid or lattice plane placed by defining a polygon on a signal layer. Advanced PCB Polygon Planes can share copper with placed tracks, pads, fills, etc. associated with designated nets.

pre-route Connection completed by the designer prior to using netlist-based autorouting options. Advanced PCB processes these connections independently from autorouted connections.

primitive Individual tracks, pads, vias, fills, etc. – stored individually in the PCB data.

ratsnest Special straight line display of netlisted connections on the “placed” PCB layout.

schematic capture CAD package for circuit design capable of generating a netlist or report output allowing the design information to be transferred to another CAD environment.

selection A special display state that indicates items included in a selection. Items must be selected in order to be Cut or Copied to the Clipboard. See also highlight.

serial Refers to RS-232C and RS-422 standard for data terminal equipment (DTE) communications.

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 of the PCB without using holes, carriers or mounts.

snap grid An invisible array of regularly spaced points on the screen which defines the current cursor position and the available location for any object in the Advanced PCB workspace.

snap to Special property of track placement in Advanced PCB where tracks will "attach" to pads without blocking the plotted pad holes, if the track is led to within 10 mils (.010) of the pad center when manually routed.

software arc When plotting, arcs which are generated by Advanced PCB using straight line chord segments. See also Hardware arcs.

Solder mask Special plot used to create a mask for the top or bottom layers of a PCB. The mask is a "resist" layer which leaves pads exposed to the solder, while protecting any tracks, etc.

solder side Refers to the Bottom side layer of a PCB.

Status line The window at the bottom-left of the screen which displays the current X and Y coordinates of the cursor; grid, track and vias defaults; as well as user prompts.

string Individual element of free or component text.

through hole PCB technology where all component pins pass through all layers of the assembled PCB.

track Also called traces, used to carry current or signals on a PCB.

vector font In Advanced PCB, special fonts that support pen plotting and photoplotting.

vector plotter A vector photoplotter creates a plot by “drawing” each stroke and flash individually. To create a stroke, the film is moved relative the light source. Flashes are made with the film and light source stationary. See also raster plotter.

vertex Point where two connected polyline (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, blind or buried.

visible grid Either of two independent, user-definable display layers which provide a visual reference for positioning items accurately on-screen.

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

Z