

Protel for Windows Online Help Index



To choose a topic, Press Tab to select the underlined topic you want to view, and then Press **Enter**.

With the mouse, point to the underlined topic you want to view, and then click the mouse button.

Basic Concepts

Commands

Printing

Managing Files

Screen Regions

Reference

Compared to other Windows applications

Register your software

After sales support

About Protel for Windows

Protel for Windows PCB capabilities

For information on how to use Help, Choose **F1** or choose Help Using Help.

Registering 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 PCB License Number is stamped on the registration card tab in the front of the manual binder.

Always refer to this number when making any enquiries regarding your PCB package.

After sales support

Protel products are supported by an international network of distributors and dealers. When you register your software you become eligible for technical support.

Support policies and procedures vary from country to country. If you have questions about your software, please direct your initial enquiry to your Protel dealer.

When enquiring about your Protel software, always quote your software license registration number.

The license number for this package appears on the registration card shipped with your Protel for Windows software.

About Protel for Windows

Protel for Windows brings the professionalism of Protel's proven PCB design system to the highly-productive Windows 3 environment. Protel for Windows users benefit from the high level of support provided by an established international network of regional offices, distributors, dealers and users.

Protel for Windows has been created for today's preferred PC environment. This system combines the natural advantages of Windows with the advanced capabilities of Protel to yield a powerful, flexible design system.

New users will find Protel for Windows easy to learn and use. Experienced Protel users will be pleased by the extensive improvements and enhancements incorporated into the highly-productive Windows environment.

Because Windows is graphically-based, learning any new application is a highly-intuitive process. Once you have mastered one Windows application, you already know a lot about other applications because of the standardized way that many tasks are performed under Windows, such as loading files or transferring information between applications.

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.

Protel for Windows supports 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 or even between applications, using Windows multi-tasking capabilities.


Protel for Windows PCB capabilities

Protel for Windows PCB is a complete PCB layout environment with many attractive features for productive design work. You can use Protel for Windows PCB as a stand-alone manual board layout. When combined with a schematic capture package, Protel for Windows PCB becomes the backbone of a fully-automated, end-to-end design system that features a high degree of design automation and integration. However you use Protel for Windows PCB, you'll appreciate its ease-of-use and the high degree of flexibility built into this proven PCB design system.

PCB generates through-hole and SMD designs of up to sixteen signal layers, plus four mid-layer power planes and four mechanical drawing layers. Board size can be as big as 32 inches (or 81 cm) square. Placement accuracy is to 1 mil (.001 inch or .025 mm). A metric/imperial grid system allows you to work accurately in both measurement systems and the grid can be "toggled" between metric and imperial modes as you design.

Full support is included for popular netlist formats. Importing a schematic netlist allows you to take full advantage of Advanced PCB's auto component placement, autorouting and design rule checking facilities. Windows 3 features support for many output devices including printers, PostScript printers and imagesetters, and pen plotters. PCB also includes Gerber plot and Excellon NC drill file capabilities.

Protel for Windows compared to other Windows applications

The way Protel for Windows uses tools, menu commands and shortcuts will already be familiar to experienced Windows users - such as using combinations of  and other keys to execute menu commands.

In short, Protel for Windows looks and runs like other Windows applications - and it's as flexible as you are. However you should be aware that Protel for Windows is different from other Windows "drawing" applications in some of its fundamentals.

A PCB design is a series of layers - layers which correspond to the individual "tools" used to create the board such as the Top and Bottom signal layers or the silkscreen Overlay layer. PCB displays and manages these layers independently and some operations, such as track placement are layer dependent - you must first select the layer, then place the track. PCB print/plot options also reflect this requirement for "layered" design.

PCB design differs from other drawing tasks in its requirement for extreme precision. As a result, PCB is more of a "placing" environment than a freehand "drawing" environment. Another fundamental difference is connectivity - PCB's ability to recognize connections between track segments, tracks and component pads, etc. For example, PCB 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 PCB.

Mouse and keyboard shortcuts

Protel for Windows supports 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 can take slightly longer than using the Edit Change command.

To move an item, hold down **Ctrl** click 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 Protel for Windows.

Some shortcuts that were part of the Protel Autotrax default key macro set have been retained in Protel for Windows, for the convenience of experienced Protel DOS system users.
See also: hot keys.

Windows allows you to assign operations to specific key combinations by using the Recorder feature. See you 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.

Hot Keys

Hot keys	Command or action
	Alt +
E	Edit Menu
E	Change Menu
P	Place Menu
D	Delete Menu
M	Move Menu
J	Jump Menu
S	Select Menu
	Alt
C or	
C	Current Menu
Z	Zoom Menu
	Alt +
O or	
O	Options Menu
	Alt +
F or	
F	File Menu
	Alt +
L or	
L	Library Menu
	Alt +
N or	
N ,	NetList Menu
	Alt +
A or	
A	Auto Menu
	Alt +
I or	
I	Info Menu
	Alt +
w or	
w	Window Menu
	Alt +
H or	
H	Help Menu
U ,	Autoplace Tools Menu
T	Toggle Select Menu
X	DeSelect Menu
	Ctrl +
A	Place Single Arc By Center
	Alt +
H	Select Physical Net (Highlight Net)
	Alt +

- G Set Snap Grid Size
 - Alt +
- Alt Set Current Layer
 - Alt +
- Alt Measure Single Distance
 - Alt +
- Alt Set Current String Height
 - Alt +
- Alt Set Current Track Width
 - Alt +
- Alt Set Current Pad Type
 - Alt +
- BkSpc or
 - Alt +
- Alt Do Undo
 - Alt +
- V Set Current Via Size
 - Alt +
- Alt Zoom Window
 - Alt Help Index
 - Shift +
- Alt
 - Shift +
- F5
 - PgUp Zoom Expand
 - PgDn Zoom Contract
 - Alt +
- PgUp Zoom In Maximum
 - Alt +
- PgDn Zoom All
 - * Next Active Signal Layer
 - + Next active layer
 - Previous active layer
- Home Zoom Pan
 - End Zoom Redraw
- Ctrl +
 - Home Jump Absolute Origin
 - Ctrl +
- End Jump Relative Origin
 - ↑,
 - ↓,
 - ←,

→ Move One Grid point
Ctrl+



Move Ten Grid point

Ctrl+



Cut

Ctrl+



Copy

Ctrl+ins

Paste

Ctrl+



Clear

Reference

Keyboard

Messages

Special Files

Photoplotter output files

Basic Concepts

Design Environment

System Options

Moving Around The Board

Searching

Jumping

Loading and Saving PCB Files

Edit The PCB Design

Selection

Component Libraries

Pad Types

Automatic Tools

Output

Design Environment

PCB Layers

Metric and imperial units

PCB Primitives

System Options

Default Settings

Preferences

Layer and Display Colours


Grids

Introduction to Grids

Protel for Windows supports two different grids which work completely independently of one another. The first is the snap grid and the second is the visible grid. Both grids can be set using through the **Options Menu**. Both grids are relative to some point (the current origin), and can have a value between 1 and 1000 mil.

The Snap Grid
The Visible Grid

Snap Grid

The snap **grid** is an array of points on the screen to which cursor movement is restricted. Whenever the cursor keys or the mouse are used to move the cursor around the screen the cursor follows (snaps) from one grid position to the next. For example, setting the snap grid to 100 mils will mean the cursor can only be on points, 0.0 inch 0.1 inch 0.2 inch etc. If you are using the cursor keys to move the cursor, you will find that Choosing  and a cursor key makes the cursor jump move 10 times the current setting of the snap grid.

This grid also controls the default display of measurements. If it is set to **imperial** then measurements will be display as imperial. If it is **metric** then measurements will be display as metric.

See Also [**Metric and imperial units.**](#)
[**Current Snap Grid**](#)

The Visible Grid

This is a guide which appears as a piece of transparent graph paper placed over the PCB design.

The **visible grids** are tools to assist in accurate placement and alignment. On the **Options Layers** dialogue box, there are two separate Colors for these grids. If the grid lines will be closer than 5 pixels apart on the screen, then these grids will not be displayed.

See Also: **Metric and imperial units.**
 Current Visible Grid 1
 Current Visible Grid 2

Automatic Tools

Netlists

Autorouting

Auto Placement

Output

Printing/Pen Plotting

Photoplotting

NC Drilling

Autorouting

The process of converting a "from-to" connection to a collection of tracks and vias that from an electrical connection, is referred to as ROUTING. When this process is performed automatically by a computer program, it is referred to autorouting.

Protel for Windows contains a collection of autorouters which can be run separately or together, to perform this task. The successful use of autorouters is dependent on a number of different parameters. The quality of the component placement, the density of the PCB, the number of copper layers that can be used, not to mention the aesthetic requirements of individual designer. A basic introduction to autorouting technology will help you to get the most from the autorouter in Protel for Windows.

Judging Autorouters

Layer Biasing

Autorouter Types

Protel Autorouter

Autorouter Setup

Autoroute All

Autoroute Connection

Autoroute Net

Autoroute Component

Autorouting Models

Judging Autorouters

The success of an autorouter, is usually judged primarily by the percentage of the board that it can autoroute. As a designer you have to consider a number of other success measurements, depending on the design problem. If the board is a standard digital board with two signal layers, to be manufactured as a plated through board, then completion percentage may be your 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.

For mixed analog and digital boards, you should consider manually routing the analog section, using the powerful editing tools in Protel for Windows, and then autorouting the remaining digital sections.

For digital boards that have large and complex bussing structures, consider manually routing 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 standard 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 autorouted successfully with Protel for Windows, 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 SMT 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 high density boards.

Layer Biasing

Multilayer board manufacturing technology enables you to route connections across two or more layers, separated by a dielectric (usually fibreglass).

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.

In Protel for Windows, each layer can be set as horizontal, vertical, both (for single layer routing) or turned off altogether.

For this reason, layer biasing is commonly used in designing boards with two or more layers. That is, placing predominantly vertical tracks on one layer, horizontal on the next, vertical on the next and so on. This does not mean that routing is restricted entirely to this direction, but rather, that the majority of tracks on a given layer will travel in the same direction.

Autorouter Types

Basic classes

There are two main classes of autorouter, those that work through the board, one connection at a time until finished, and those that, after finishing, 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 sections, "rip-up" and "shove aside".

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.

See Also

Pattern routers

Line Probe Routers

Wave Expansion Routers

Pattern routers

Sometimes referred to as 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 get a small percentage of 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 to them.

Line Probe Routers

These routers move a test track from both ends of a connection until an obstacle is encountered. They move sideways until the obstacle is not there and then continue on there way. When they meet, the connection is complete. There is usually a limit to how far sideways they 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 fail to solve what would appear to be very simple problems, and as a result, they tend to slow down and fail as the board gets fuller. This is because they are not evaluating all possibility, but simply following the best one.

When combined with a heuristicly guided route shape, such as a Z shaped 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 water will spread out in all directions until, eventually, if at all possible, the water will get to the other end of the connection. This type of router 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 in the wrong biasing direction. The final solution will then be a balance of these parameters, controlled by the size 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 grid type, each point of space on the board is represented as a point in a two dimensional array, either filled or empty. The problem here is that the memory requirements go up as the square of the grid resolution.

In the gridless type of wave router, the expansion is performed by expanding the current point as rectangle that fits over the current point and goes in all directions until obstacles are hit. When an obstacle is hit, 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 polygon calculations (as opposed to point calculations).

Although tremendously powerful and flexible, generally, all wave type routers are slow and memory hungry.

Protel Autorouter

The Protel for Windows autorouter is a multi-algorithm router. It works with up to 16 signal layers plus power and ground planes with grid sizes of 5-100 mil. The passes include various pattern, line probe and a maze router. You can control all clearances (air gaps) and, track and via sizes. You can specify which passes are used and what layers and directions are used for tracks. The autorouter passes are:

Pre-Router

SMT Stringers

Memory Router

Pattern Routers

Line Probe

Maze Router

Smoothing

Add arcs

Autorouter Pre-Router

If you place some items on the board to complete connections (pre-routing) then set this pass on. It will check all the connections and make sure that the router will only attempt to route the unrouted ones. If you have used the Auto Manual Route option to complete some connections, then this option is not needed for those connections.

Autorouter SMT Stringers

If you are using power and ground planes with SMT components, use this option to route those pins that need to be connected to the planes (with a short track segment and a thermal relieved pad).

Autorouter Memory Router

This is a very fast heuristic pattern router to put in the classic wave structure for memory busses.

Autorouter Pattern Routers

These are very fast simple pattern routers which allow zero, one or two vias, and up to three track segments. They will stick rigidly to the layer biasing rules and will never travel in the "other" direction on any given layer.

Autorouter Line Probe

These are combination line probe and pattern routers. They use a basic L, C or Z shape, with the separate sections completed by a line probe router.

Autorouter Maze Router

This is a gridded wave expansion router. This pass is used after all the others have failed. It copies a description of the board into a cell map and then attempts to route un-routed connections. The amount of memory required by this router is determined by the board size and layers. Basically, one byte for each layer at each grid point, plus, if hugging is on, one byte for each layer at each grid point. For example, on a 25 mil grid, a 4 inch x 5 inch board with 4 signal layers, will require $4 \times 5 \times 40$ (point per inch) $\times 40 \times 4$ (layers) = 64000 bytes, or 128K if hugging is turned on.

Autorouter Smoothing

After all routing is completed, this pass checks the board for vias that can be removed by swapping tracks from one side of the board to the other. This pass should be switched off when routing. When you are happy with the result, then turn it on, and turn the routing passes off, and run the router.

Autorouter Add arcs

If you wish to use curved tracks, then turn this pass on after you are happy with the router result. It will attempt to replace corners on tracks with arcs. You should use this pass only after finishing board routing. After all the arcs are installed, the screen redraw time may be significantly increased, and the PCB file will be larger. This pass will add 90 and 45 degree arcs only.

Autorouter Setup

All the values described below should be set after consultation with your board manufacturer to make sure he can work to the tolerances you require.

Autorouter Grid Size
Autorouter Track Width
Autorouter Via Size.
Clearances

Autorouter Grid Size

The autorouter grid can be different from the snap grid. All tracks, vias or arcs placed by the autorouter, will be on this grid. It can have any value from 5 to 100 mils. For all the routing passes, decreasing the grid size will slow the program down. For the maze router, the memory requirements, for storing the grid model of the board, go up as a square of the grid resolution, so a 25 mil grid will use four times as much memory to model the board, as 50 mil grid. Making the grid smaller will. You should make sure that most of your pads are on the currently set router grid. Use the command Netlist Router Off Grid to highlight any off grid pads. Use the command Netlist Autoplace Move to grid to move components to the nearest grid point.

Autorouter Track Width

This is the width of any tracks, straight or curved, put in by the autorouter. Enter any value from 1 to 255 mils. This value should be set in conjunction with the grid size and clearances to allow successful routing.

Autorouter Via Size

The diameter of any vias place by the autorouter. Enter any value from 1 to 255 mils

Autorouting Models

To get the most from the router, you must select a combination of clearances, track size, via size and routing grid that will allow the router to work. Here are some examples.

See Also: **Single Density 100**
 Single Density SMT 50 mil
 Double Density SMT 50 mil
 Triple Density 1
 Triple Density 2

Single Density 100 mil pad centers

Track size	12 mil
Via size	50 mil (or 62 mil if you like)
Clearances	13 mil
Grid size	25 mil

To use this model, the majority of your pads should be 62 mil or less in diameter. This will enable a single track, maintaining clearances, to pass between two pads 100 mil apart.

Single Density SMT 50 mil pad centers

Track size	8 mil
Via size	40 mil
Clearances	8 mil
Grid size	25 mil

To use this model, the majority of your pads should be 26 mil 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 mil apart. The via size can be varied as required.

Check with you board manufacturer before giving him this board to make. This a fairly standard SMT board.

Double Density SMT 50 mil pad centers

Track size	5 mil
Via size	30 mil
Clearances	5 mil
Grid size	10 mil

To use this model, the majority of your pads should be 24 mil or less wide (in one axis), to leave a gap of 26 mils between pads. This will enable two 5 mil tracks, maintaining clearances, to pass between a pair of pads 50 mil apart. The via size can be varied as required.

Check with your board manufacturer before giving him this board to make. This is a very demanding level of technology.

Triple Density 1

Track size	8 mil
Via size	42 mil
Clearances	8 mil
Grid size	16.67 mil

To use this model, the majority of your pads should be 42 mil or less in diameter. This will enable three tracks, maintaining clearances, to pass between a pair of pads 100 mil apart.

Check with you board manufacturer before giving him this board to make.

Triple Density 2

Track size	6 mil
Via size	30 mil
Clearances	6 mil
Grid size	12.5 mil

Comments: To use this model, the majority of your pads should be 56 mil or less in diameter. This will enable three tracks, maintaining clearances, to pass between a pair of pads 100 mil apart.
Check with you board manufacturer before giving him this board to make.

PCB Layers

Copper Trace Layers

Silk Screen Overlays

SMT Paste Mask Layers

Solder Mask Layers

Post Processed Power Planes

Keep Out Layer

Mechanical Layers

Drill Layer

Copper Trace Layers

There are 16 layers for track routing. Anything placed on these layers will appear as copper on the final PCB.

Top Layer

Sometimes referred to as the component side of the PCB. For boards with **components** on one side only, this is the side for component placement.

Bottom Layer

In conventional **through hole** boards this is referred to as the **solder side**.

Mid Layers 1 To 14

These are only used for track routing in **multilayer** boards.

Silk Screen Overlays

Top and bottom for text of component designator and comments to be silkscreened onto.

SMT Paste Mask Layers

Top and bottom for photo mask or silkscreen mask of solder paste locations. Used in the manufacture of boards with surface mount devices.

Solder Mask Layers

Top and bottom for photo mask or silkscreen **solder mask**. This separates all the areas where a wave solder machine will or wont place solder (usually only on **component**)

Post Processed Power Planes

The Power Plane attribute on pads is used to generate these **layers**. **Tracks**, **arcs**, **fills**, etc can be placed on this layer but they are plotted as negative, so a track becomes a break in the copper. To preview these, generate them as photoplots and load them back using the **Gerber In** command.

Keep Out Layer

Specify multilayer **keep outs**.

The Keep Out layer is used to define the physical limits for both automatic component placement and autorouting.

This is achieved by placing a perimeter of tracks or by placing area fills to define any "no-go" areas;

Mechanical Layers

Used to place dimensions and instructions for fabrication and assembly of the board.

Drill Guide Layer

The **Drill hole guide** is a plot of all the holes in your design (sometimes called a pad master). The plot includes all multi-layer pads and vias. You can specify a guide hole, which locates the hole center, from Output Options dialog box)

Metric and imperial units

Resolution: The metric resolution of the program is 0.025 mm so you can enter three decimal places but the least significant will be rounded to the nearest 0.005 mm.

All measurements can be entered in **metric** or **imperial**. To enter in imperial (mils or 0.001 inch units) just type the number with no decimal point. To enter in metric, type the number in millimetres with a decimal point. If there is a decimal point then the program assumes a metric measurement.

Most measurement entry menus have both metric and imperial selections or you can select the "other" option and enter any value.

PCB Primitives

Arcs

Tracks

Vias

Pads

Fills

Text strings

Components

Arcs

Arcs are essentially circular curved tracks. They can be placed on any layer with any radius and any line width from 1 to 255 mils wide. The angular resolution is 45 degrees. The number of arcs is limited by memory only. Arcs can be placed using the place arc commands or as tracks using the place track command.

See also: Replace arcs pass of the autorouter.

Tracks

Tracks can be placed on any layer with any width from 1 to 255 mils wide. The number of tracks is limited by memory only. Tracks can be placed using the place track command or by the autorouter.

Vias

Vias are always multi layer and can be any diameter from 1 to 255 mils wide. The number of vias is limited by memory only. Vias can be placed using the **place via** command or by the autorouter. The hole size of vias can be set from 1 to 255 mil.

Pads

Pads can be rectangular, rounded or octagonal with X and Y size specified. Circular is rounded with the same X and Y value.

Pads can be placed on any layer or made multi layer in which case they are copper on all layers.

Sizes can be from 1 to 4000 mils on both axis with the hole size from 0 (SMT) to 4000 mils. Pads which are to be used in components can be identified with a 4 character designator.

When a pad is first placed, its designator is empty. If the pad is to remain as a free pad, then leave it like this. If the Pad is to be a mounting hole, then leaving its designator as empty allows the DRC to ignore it (ie not report it as an empty pad).

Fills

Rectangular **fills** are simply rectangular areas of any size on any layer.

Text strings

Text **strings** can be placed on any layer with any height and can be any one of system **Fonts**. The number of text strings is limited by memory only. Strings can be placed using the **place string** command.

Fonts

The Protel for Windows 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. ComponentsGL_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 European languages other than English.

The default font supplied with the system is built into the software and cannot be unloaded). To load or unload fonts use the Library Fonts command.

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 its current position.

Strings from converted Autotrax files will be converted to the Default font.

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.

Components

Components consist of any number of the above items to any size grouped together. Both through-hole and **SMT** components can be placed on either side 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 or value of the component can be up to 32 characters and can be placed anywhere relative to the component on either copper or overlay. Both comment and designator can be hidden from view and set to any size and font.

Netlists

Netlist Commands

Definition A net, in electronic design, is a collection of component pins (nodes on the net) which are electrically connected. For example, all the pins connect to +5v on a PCB are connected together and hence constitute a net. (The term "net" is shortened from "network", but is now standard). Each net is given a name (+5v , VCC or GND for example) . A netlist is the collection of nets that define all the electrical connections in a design. A netlist file is the list of nets (the netlist) stored in a disk file.

PFW Nets Each net has a name (up to 20 characters long) and each node consists of a component designator and a pad designator, separated by a hyphen. For example U23-16 refers to pin 16 on component U23.

Loading After a netlist has been loaded to a PCB, the program converts each net to a series of logical connections. These connections are simply two of the nodes from the net and have no physical existence. This process is referred to as "optimising" and the basic idea of this is to produce the series of connections that will complete the net with the shortest track length. The display of all the connections is referred to as a rats nest).

When a connection is converted to tracks and vias (ie when it is routed), then logical connection is no longer displayed. When all the connections in a net are routed, then all the nodes in the net will be physically connected together. When all the nets are routed, the PCB design is complete.

Source of Netlist The netlist for a design can be obtained either automatically (from a schematic capture program) or manually (using a text editor).

Optimize Methods

When you first loaded the netlist, you were advised to use the Shortest option for optimizing the nets. There are a number of other optimization methods, and these can be applied to individual nets or groups of nets.

Use the **Netlist Optimize commands** to optimize nets.

The list of optimization methods will appear. The options are

- | | |
|------------------------------|--|
| Shortest | This simply looks for the set of connections (with any topology) that connects all the nodes together. |
| X Bias | Looks for the set of connections (with any topology) that connects all the nodes together. The program will prefer horizontal shortness five times more than vertical shortness. Use this to force routing strictly in the horizontal direction. |
| Y Bias | Looks for the set of connections (with any topology) that connects all the nodes together. The program will prefer vertical shortness five times more than horizontal shortness. Use this to force routing strictly in the vertical direction. |
| Daisy Chain | This 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. They will appear in the connection list in exactly the same order as the order of the nodes in the original netlist file. Each node will have two connections to it, except the first and last which will have only one. No minimisation is performed. This option may be used for highly critical ECL designs. |
| Minimum Daisy Chain | Using this method, the nodes 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. Minimisation is performed, within the limits of the daisy chain topology, to reduce the total connection length. |
| Start End Daisy Chain | Using this method, the nodes 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. Minimisation 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. |

Command Reference

File

Edit

Library

Netlist

Auto

Current

Options

Zoom

Info

Window

Help

Gerber Menu

Gerber In
Batch Load
Gerber Output

Reports Menu

Bill of Material

Board Specifications

Netlist Status

File Menu

The file menu in Protel for Windows performs tasks associated with loading and saving PCB files, exiting the program and printing/plotting designs.

New

Open

Close

Save

Save As

Save All

Gerber

Plot/Print

NC Drill

DXF

Export Selection

Reports

Exit

Select Menu

Inside Area

Outside Area

All

Free Prims

All on Layer

Physical Net

Physical Connection

De-Select Menu

- Inside Area
- Outside Area
- All
- Free Prims
- All on Layer

Toggle Menu

[Arcs](#)
[Components](#)
[Fills](#)
[Pads](#)
[Strings](#)
[Tracks](#)
[Vias](#)

See Also: [Toggling the selection](#)

Delete Menu

Arc
Component
Fill
Pad
String
Track
Via

Change Menu

Arc
Component
Fill
Pad
String
Track
Via

See Also: Making Changes

Move Menu

Move Selection
Flip Selection
Rotate Selection
Break Track
Drag End
Whole Track
Re-Route
Arc
Component
Fill
Pad
String
Via

Place Menu

Arc (Center)

Arc (Edge)

Component

Fill

Pad

String

Track

Via

Polygon Plane

Coordinate

Dimension

ArrayPlace

Search Menu

Component

Net

Pad

String

Jump Menu

Absolute Origin

Current Origin

New Location

Edit Menu

Undo
Redo
Cut
Copy
Paste
Clear
Select
De-Select
Delete
Change
Move
Place
Search
Jump
Set Origin

Library Menu

Components

Un-Group

Pad Types

Apertures

Optimize Menu

Net
On Component
All

Show Connections Menu

Net
On Component
All

Hide Connections Menu

Connection

Net

On Component

All

Netlist Menu

Load

Clear

Optimize

Show Connections

Hide Connections

Edit Net

Identify

Length

Export

Generate

DRC

Clearances

Power Planes

Autoroute Menu

All
Net
Connection
On Component
Pad To Pad

Un-Route Menu

All
Net
Connection
On Component
Selection

Auto Menu

Auto Place

Move To Grid

Density

Manual Route

Autoroute

Setup Auto Route

Un-Route

Free Text Menu

[Height](#)
[Default](#)
[Sans Serif](#)
[Serif](#)

See Also: [Fonts](#)

Via Menu

Diameter
Hole Diameter
Through Hole
Blind/Buried

Current Menu

Pad Type

Track

Via

Component Text Comments

Component Text Designators

Free Text

Snap Grid

Visible Grid 1

Visible Grid 2

Layer

Track Mode Menu

Any Angle

90/90 Line

45/90 Line

90 Arc/Line

Options Menu

Layers

Preferences

Display

Track Mode

Tool Bar

Status

Scroll Bars

Zoom Menu

Window

Point

Select

All

In

Out

Pan

Redraw

Info Power Planes Menu

Plane 1
Plane 2
Plane 3
Plane 4

Info Menu

System Status

Board Status

Components On PCB

Selected Pins

Nets

Measure Distance

Length of Selection

Power Planes

Window Menu

Tile

Cascade

Arrange Icons

Close All

Help Menu

[Index](#)

[Using Help](#)

[Basic Concepts](#)

[Commands](#)

[Printing](#)

[Managing Files](#)

[Screen Regions](#)

[Reference](#)

[About](#)

File New

Shortcut Keys:

Ctrl

Ctrl

Summary: Displays a new document window with the title "PCB_#".

Procedure: Choose File New. A new document window becomes the current window.

File Open

Shortcut Keys:



Summary: This will open a new document window with the contents of the selected file.

Procedure: Choose File Open. The file may be of any type supported by Protel For Windows.

When you start PCB, The Load PCB File dialog box is opened. The Menu bar displays three options: File, Info and Help. The Tool bar 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 new (empty) document window:

1. Type the filename (include the full path, if different from the path listed after Directory;
2. Click OK to open the files.

You can also double-click on the desired filename in the Files window, if any. To change directories, click under any of the options listed in the Directories window.

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

You can abort the drawing (or redraw) of the document window, at any time, by pressing . This allows you to move directly to another menu command or Tool button without waiting for the entire screen redraw to be completed.

See Also: [File Open Dialog Box](#)
[File Save](#)
[File Save As](#)
[File Formats](#)

File Close

Shortcut Keys:



Summary: Closes the current document window.

Procedure: Choose a document window, then choose File Close. If modifications have been made to the contents of the document window, a message is displayed that prompts to save, discard, or cancel.

See Also: [File Exit](#)

[File Save](#)

[File Save As](#)

File Open dialog box

Open File Name

Type a PCB file name or select a name from the Files box. Select the name of the PCB Window. This box lists files with .PCB extensions.

Directories

Select the drive and directory where the PCB Window is stored. Choose OK to see a list of PCB Windows in the selected directory that have the extension specified in the Open File Name box. Default is current drive and directory.

See Also:

[File Open](#)

[File Save](#)

[File Save As](#)

[File Formats](#)

File Save

Shortcut Keys:



Summary: Saves the contents of the current document window in Protel Binary format using the existing path and filename.

Procedure: Select a document window, then choose File Save.

See Also: [Save As Dialog Box](#)
[File Save As](#)

File Save As

Shortcut Keys:

Ctrl

Ctrl

Summary: Save the contents of the current document window to a new path, filename, and file format.

Procedure: Select a document window, then choose File Save AS.

See Also: [File Open](#)

[File Save](#)

[File Formats](#)

File Save As dialog box

Save File Name Type a new file name to save in a different file or leave the current name to save with the same name.

Directories Select the disk drive and directory where you want the PCB Window stored. Default is current drive and directory.

See Also: [**File Open**](#)
[**File Save**](#)
[**File Formats**](#)

File Save All

Shortcut Keys:



Summary: Saves the contents of all the document windows to the Protel Binary file format using the existing paths and filenames.

Procedure: Choose File Save All.

See also: [File Save](#) command

[File Save As](#)

[File Formats](#)

Gerber In

Shortcut Keys:



Summary: Loads a Gerber file into the current document window. During loading, flashes are converted to free pads and lines are converted to free tracks.

See Also: [Gerber Batch Load Library Apertures](#)

Gerber Batch Load

Shortcut Keys:

Ctrl

B

Ctrl

Summary: Loads all Gerber files that match the filename into the current document window. During loading, flashes are converted to free pads and lines are converted to free tracks. Each file will automatically be assigned to a layer determined by its file extension (see filename conventions)

Procedure: Select a document window. Open an aperture table using the Library Apertures

Choose File Gerber Batch Load,

During the Gerber loading process, lines will be converted to free tracks and flashes converted to free pads.

See Also: [Gerber In](#) [Library Apertures](#)

Gerber Output

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Generates Gerber files from the current document window. Gerber files can be single layer or composite, automatic panelization and batch plotting are supported. All layers and primitives that are displayed will be plotted.

Procedure: Select a document window and choose Options Layers to turn on or off the layers as needed. All layers that are displayed will generate photoplot files. Next use the Options Display to hide or display primitives as necessary. Primitives that are not displayed will not plot. Draft mode is not supported during photoplotting.

Choose File Gerber Gerber Output, set the options and use the OK button.

See Also:

[Options Layers](#)

[Options Display](#)

[Library Apertures](#)

File Plot/Print

Shortcut Keys:

Ctrl

Ctrl

Summary: Generate Printer or plotter output. Displayed layers and primitives will print in single layer or composite modes. Draft outline mode is supported.

Procedure: Choose a document window, then choose Options Layers. Toggle the layers on or off as needed. All layers that are displayed will print or plot. Next use the Options Display to hide or display primitives as necessary. Primitives that are not displayed will not plot. Draft mode is also supported during plotting and printing.

Choose File Plot/Print, after these settings are complete use the OK button.

See Also: [Options Layers](#)
[Options Display](#)
[Printing](#)

File NC Drill

Shortcut Keys:

Ctrl

Ctrl

Summary: Generates binary and ASCII **NC Drill files**.

Procedure: Choose a document window, then choose File NC Drill.

See Also: **Introduction to NC-Drilling**
Generating NC Drill files

File DXF

Shortcut Keys:



Summary: Generates a DXF format file.

Procedure: Choose a document window and use Edit Select and Edit De-Select to select the primitives and components to include in the DXF file. Choose File DXF.

Export Selection

Shortcut Keys:



Summary: Generate a Protel format PCB file from the selected items.

Procedure: Choose a document window and use Edit Select and Edit De-Select to select the primitives and components to include in the DXF file. Choose File Export Selection.

Any selection can be saved as a Protel for Windows 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 import these files into any other PCB document. To use this feature:

1. Make sure that the current selection includes only those items you wish to save;
2. Choose File Export Selection
This command opens the Export Selection File Name dialog box.
3. Type a filename for the selection at the prompt;

You can include a new path in the file name or change the directory by clicking in the Directories box.
4. Click OK.

The file will be generated. This may take a few moments if the selection is large.

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

See Also: [Edit Select](#)
[Edit De-Select](#)
[Selection](#)

Bill Of Material

Shortcut Keys:



Summary: Generates an ASCII parts list file.

Procedure: Choose File Reports Bill of Materials, enter a filename and the bill of materials will be written to the file. The extension will be forced to BOM.

An example of the format used is

Bill of Material for F:\BOARDS\DEMO.PCB
On 10-Nov-1991 at 17:44:01

Comment	Pattern	Quantity	Components
0.1uF	RAD0.2	7	C4, C5, C6, C3, C9, C10, C1
100uF	RB.2/.4	1	C7
10uF	RB.2/.4	1	C8
1488	DIP14	1	U11
1489	DIP14	1	U10
330R	AXIAL0.4	1	R5
4.00 MHz	XTAL1	1	XTAL1
4.7K	AXIAL0.4	1	R6
4.915 MHz	XTAL1	1	XTAL2
4040	DIP16	1	U12
470R	AXIAL0.4	4	R2, R3, R4, R1
4k7	AXIAL0.4	1	R7
6264/2764	DIP28	4	U2, U1, U4, U3
74LS04	DIP14	1	U9
74LS138	DIP16	1	U8
8255 PPI	DIP40	1	U7
BAUD SELECT	DIP16	1	SW1
IO LINES	IDC40	1	J4
PB RESET	RAD0.2	1	SW2
Power	FLY4	1	J1
SERIAL 1	DB9	1	J2
SERIAL2	DB9	1	J3
Z80A CPU	DIP40	1	U5
Z80A SIO	DIP40	0	U6

Board Specifications

Shortcut Keys:



Summary: Generates an ASCII report file that lists of primitives and components on the board, together with holes counts, copper areas and layers used.

Procedure: Choose File Reports Board Specifications, this will generate a report of the following format.

Specifications for F:\BOARDS\DEMO.PCB

On 10-Nov-1991 at 17:44:15

Size of board 5.912 x 5.012 sq in
Equivalent 14 pin components 0.98 sq in/14 pin component
Components on board 36

Layer	Route	Pads	Tracks	Fills	Text	Arcs
Top Layer	Vert	0	427	0	4	0
Bottom Layer	Horiz	0	924	0	1	0
Top Overlay	-	0	203	0	0	15
Multi-Layer	-	435	0	0	0	0

Total		435	1554	0	5	15

Layer Pair	Vias

Top Layer - Bottom Layer	85

Total	85

Hole Size	Pads	Vias

28 Mil (0.700 mm)	0	85
30 Mil (0.750 mm)	435	0

Total	435	85

Netlist Status

Shortcut Keys:



Summary: Generates an ASCII file that lists the routed/unrouted status of the board together with the names of nets loaded.

Procedure: Choose File Reports Netlist Status, this will generate a file of the following format

Nets report for D:\GEMINI\TPU\READY50.PCB
On 10-Nov-1991 at 17:44:50

```
+12
  J1-3 to U11-14      Unrouted

-12
  J1-4 to U11-1      Unrouted
BAUDCLK
  U6-27 to U6-14     Unrouted
  U6-14 to U6-13     Unrouted
  U6-13 to SW1-9     Unrouted
  SW1-16 to SW1-15   Unrouted
  SW1-15 to SW1-14   Unrouted
  SW1-14 to SW1-13   Unrouted
  SW1-13 to SW1-12   Unrouted
  SW1-12 to SW1-11   Unrouted
  SW1-11 to SW1-10   Unrouted
  SW1-9 to SW1-10    Unrouted

C10.1_U9.2
  U9-2 to R1-2       Unrouted
  C10-1 to R1-2     Unrouted

C10.2_U9.3
  R2-1 to C10-2     Unrouted
  C10-2 to U9-3     Unrouted
to U8-8             Unrouted

INT
  U6-5 to R7-2       Unrouted
  R7-2 to U5-16     Unrouted

IORQ
  U5-20 to U6-36    Unrouted
```

File Exit

Shortcut Keys:

Ctrl

Ctrl

Summary: Quits the current session of Protel For Windows and prompts to save modified files.

Procedure: Choose File Exit, if any document window has been modified since last saving, the CONFIRM dialog box will be displayed that prompts to save the file.

Edit Undo

Shortcut Keys:

Ctrl

Ctrl

Summary: Restores the contents of the current document window to its state prior to the previous

Procedure: Choose Edit Undo

See Also: [Edit Redo](#)

Edit Redo

Summary: Restores the previous changes made by the Edit Undo


Procedure: Choose Edit Redo.

See Also: [**Edit Undo**](#)

Edit Cut

Shortcut Keys:  + 

Summary: Removes the selected items from the current document window and places them into the Protel for Windows clipboard.


Procedure: Use Edit De-Select to de-select objects that are not to be removed. Use Edit Select to select the objects that will be removed. Choose Edit Cut. Move the cursor to a reference location, the direction keys may be used to precisely align the cursor with the **snap grid**, and click or press .

See Also: [Selection](#)
[Edit Paste](#)
[Edit Select](#)
[Edit De-Select](#)
[Edit Toggle Selection](#)

Edit Copy

Shortcut Keys: +


Summary: Copies the selected items from the current document window to the Protel for Windows clipboard.

Procedure: Use Edit De-Select to deselect objects that are not to be copied. Use Edit Select to select the objects that will be copied. Choose Edit Copy. Move the cursor to a reference location, the direction keys may be used to precisely align the cursor with the **snap grid**, and click or press .

See Also: [Selection](#)
[Edit Paste](#)
[Edit Select](#)
[Edit De-Select](#)
[Edit Toggle Selection](#)

Edit Paste

Shortcut Keys: +


Summary: Copies the items from the Protel for Windows clipboard into the current document window.

Procedure: Choose a document window, then choose Edit Paste. The same contents of the clipboard may be pasted multiple times.

See Also: [Edit Copy](#)
[Edit Cut](#)

Edit Clear

Shortcut Keys: +


Summary: Removes the selected items from the current document window. Items in the clipboard are not affected.

Procedure: Choose a document window. Use Edit De-Select to deselect objects that should not be cleared. Use Edit Select to select the objects that will be cleared. Choose Edit Clear.

See Also: [Selection](#)
[Edit Paste](#)
[Edit Select](#)
[Edit De-Select](#)
[Edit Toggle Selection](#)

Select Inside Area

Shortcut Keys:



Summary: Selects primitives and components within a rectangular region.

Procedure: Choose a document window. Choose Edit Select Inside Area, then choose two opposite corners of the region to be selected.

See Also: [Selection](#)

[Edit Select](#)

[Edit De-Select](#)

[Edit Toggle Selection](#)

Select Outside Area

Shortcut Keys:



Summary: Selects primitives and components outside a rectangular region.

Procedure: Choose a document window. Choose Edit Select Outside Area, then choose two opposite corners of a region.

See Also: [Selection](#)

[Edit Select](#)

[Edit De-Select](#)

[Edit Toggle Selection](#)

Select All

Shortcut Keys:

Ctrl

Ctrl

Summary: Selects all primitives and components within the current document window.

Procedure: Choose a document window, then choose Edit Select All.

See Also: [Selection](#)

[Edit Select](#)

[Edit De-Select](#)

[Edit Toggle Selection](#)

Select Free Primitives

Shortcut Keys:

Ctrl

Ctrl

Summary: Selects all free primitives within the current document window.

Procedure: Choose a document window, then choose Edit Select Free Primitives.

See Also: Selection

Edit Select

Edit De-Select

Edit Toggle Selection

Select All On Layer

Shortcut Keys:



Summary: All items on the **Current Layer** within the current document window become selected.

Procedure: Choose a document window, then use Current Layer to choose the layer that will be selected. Choose Edit Select All On Layer.

See Also: **Selection**

Edit Paste

Edit Select

Edit De-Select

Edit Toggle Selection

Select Physical Net


Shortcut Keys:



Summary: This command selects all free primitives (tracks, vias, fills) that connect a physical net within the current document window.

Procedure: Choose a document window, choose Select Physical Net, then choose any primitive on a copper trace layer. All primitives that touch will be selected.

To use this feature:

1. Choose Edit Select Physical Net;
You will be prompted "Select Pad."
2. Position the cursor over a component pad within the desired net and press  or left mouse.

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

See Also:

[Selection](#)

[Edit Select](#)

[Edit De-Select](#)

[Edit Toggle Selection](#)


Select Physical Connection

Shortcut Keys:  

Summary: Selects all primitives on a physical connection.

Procedure: Choose a document window. Choose Select Physical Connection, then choose any non-pad primitive on a physical connection.

To use this feature:

1. Choose Edit Select Physical Net;
You will be prompted "Select Pad."
2. Position the cursor over a component pad within the desired net and press  or left mouse.

The continuous physical net extending from the selected pad to other pads (but not beyond) will highlight with the selection color.

See Also: [**Selection**](#)
[**Edit Select**](#)
[**Edit De-Select**](#)
[**Edit Toggle Selection**](#)

De-Select Inside Area

Summary: De-selects all primitives and components within a rectangular region.

Procedure: Choose a document window. Choose Edit De-Select Inside Area, then choose two opposite corners of the region to be de-selected.

See Also: [**Selection**](#)
[**Edit Paste**](#)
[**Edit Select**](#)
[**Edit De-Select**](#)
[**Edit Toggle Selection**](#)

De-Select Outside Area

Summary: De-selects all primitives and components outside a rectangular region.

Procedure: Choose a document window. Choose Edit De-Select Outside Area, then choose two opposite corners of a region. The area outside the region will be de-selected.

See Also: [**Selection**](#)
[**Edit Select**](#)
[**Edit De-Select**](#)
[**Edit Toggle Selection**](#)

De-Select All

Summary: De-selects all primitives and components within the current document window.

Procedure: Choose a document window, then choose Edit De-Select All.

See Also: [**Selection**](#)
[**Edit Select**](#)
[**Edit De-Select**](#)
[**Edit Toggle Selection**](#)

De-Select Free Primitives

Summary: De-selects all free primitives within the current document window.

Procedure: Choose a document window, then choose Edit De-Select Free Primitives.

See Also: [**Selection**](#)
[**Edit Select**](#)
[**Edit De-Select**](#)
[**Edit Toggle Selection**](#)

De-Select All on Layer

Summary: De-selects all primitives and components on the **Current Layer** within the current document window.

Procedure: Choose a document window. Choose a layer, then use Current Layer to choose the layer that will be de-selected. Choose Edit De-Select All on Layer.

See Also: **Selection**
Edit Select
Edit De-Select
Edit Toggle Selection

Toggle Arcs

Summary: Selects all arcs were not previously selected within the document window, and de-selects all arcs that were previously selected within the document window.

Procedure: Choose a document window, then choose Edit Toggle Selection Arcs.

See Also: [**Selection**](#)
[**Edit Select**](#)
[**Edit De-Select**](#)

Toggle Components

Summary: Selects all components that were not previously selected within the document window, and de-selects all components that were previously selected within the document window.

Procedure: Choose a document window, then choose Edit Toggle Selection Components.

See Also: [**Selection**](#)
[**Edit Select**](#)
[**Edit De-Select**](#)

Toggle Fills

Summary: Selects all fills that were not previously selected within the document window, and de-selects all fills that were previously selected within the document window.

Procedure: Choose a document window, then choose Edit Toggle Selection Fills.

See Also: [Selection](#)
[Edit Select](#)
[Edit De-Select](#)

Toggle Pads

Summary: Selects all pads that were not previously selected within the document window, and de-selects all pads that were previously selected within the document window.

Procedure: Choose a document window, then choose Edit Toggle Selection Pads.

See Also: [Selection](#)
[Edit Select](#)
[Edit De-Select](#)

Toggle Strings

Summary: Selects all text that was not previously selected within the document window, and de-selects all text that was previously selected within the document window.

Procedure: Choose a document window, then choose Edit Toggle Selection Strings.

See Also: [Selection](#)
[Edit Select](#)
[Edit De-Select](#)

Toggle Tracks

Summary: Selects all tracks that were not previously selected within the document window, and de-selects all tracks that were previously selected within the document window.

Procedure: Choose a document window, then choose Edit Toggle Selection Tracks.

See Also: [Selection](#)
[Edit Select](#)
[Edit De-Select](#)

Toggle Vias

Summary: Selects all vias that were not previously selected within the document window, and de-selects all vias that were previously selected within the document window.

Procedure: Choose a document window, then choose Edit Toggle Selection Vias.

See Also: [**Selection**](#)
[**Edit Select**](#)
[**Edit De-Select**](#)

Delete Arcs

Shortcut Keys:

Ctrl

Ctrl

Summary: Removes an arc.

Procedure: Choose a document window. If the arc is not on the Current Layer and is below other arcs, then use Current Layer to choose the layer that contains the arc. Now choose Edit Delete Arc and choose the arc.

Delete Components

Shortcut Keys:

Ctrl

Ctrl

Summary: Removes a component.

Procedure: Choose a document window. If the component is not on the **Current Layer** and is below other components, then use Current Layer to choose the layer that contains the component. Now choose Edit Delete component and choose the component.

Delete Fills

Shortcut Keys:

Ctrl

Ctrl

Summary: Removes a fill.

Procedure: Choose a document window. If the fill is not on the **Current Layer** and is below other fills, then use Current Layer to choose the layer that contains the fill. Now choose Edit Delete Fill and choose the fill.

Delete Pads

Shortcut Keys:

Ctrl

Ctrl

Summary: Removes a pad.

Procedure: Choose a document window. If the pad is not on the **Current Layer** and is below other pads, then use Current Layer to choose the layer that contains the pad. Now choose Edit Delete Pad and choose the pad.

Delete Strings

Shortcut Keys:

Ctrl

Ctrl

Summary: Removes text.

Procedure: Choose a document window. If the string is not on the Current Layer and is below other strings, then use Current Layer to choose the layer that contains the string. Now choose Edit Delete String and choose the string.

Delete Tracks

Shortcut Keys:

Ctrl

Ctrl

Summary: Removes a track.

Procedure: Choose a document window. If the track is not on the Current Layer and is below other tracks, then use Current Layer to choose the layer that contains the track. Now choose Edit Delete Track and choose the track.

Delete Vias

Shortcut Keys:

Ctrl

Ctrl

Summary: Removes a via.

Procedure: Choose a document window. If the via is not on the Current Layer and is below other vias, then use Current Layer to choose the layer that contains the via. Now choose Edit Delete Via and choose the via.

Change Arcs

Shortcut Keys:



Summary: Modifies any attribute of an arc. The modifications can be applied globally to other arcs.

Procedure: Choose a window. If the arc is not on the **Current Layer** and is below other arcs, then use Current Layer to choose the layer that contains the arc. Now choose Edit Change Arc and choose the arc. Use the options button if the changes should also be applied to other arcs.

See Also: [**Changing Arcs**](#)
[**Making Changes**](#)
[**Selection**](#)

Change Components

Shortcut Keys:



Summary: Modifies any attribute of a component. The modifications can be applied globally to other components.

Procedure: Choose a document window. If the component is not on the **Current Layer** and is below other components, then use Current Layer to choose the layer that contains the component. Now choose Edit Change Component and choose the component. Use the options button if the changes should also be applied to other components.

See Also: [Changing Components Making Changes Selection](#)

Change Fills

Shortcut Keys:

Ctrl

Ctrl

Summary: Modifies any attribute of a fill. The modifications can be applied globally to other fills.

Procedure: Choose a document window. If the fill is not on the **Current Layer** and is below other fills, then use Current Layer to choose the layer that contains the fill. Now choose Edit Change Fill and choose the fill. Use the options button if the changes should also be applied to other fills.

See Also: [Changing Fills](#)
[Making Changes](#)
[Selection](#)

Change Pad

Shortcut Keys:

Ctrl

Ctrl

Summary: Modifies any attribute of a pad. The modifications can be applied globally to other pads.

Procedure: Choose a document window. If the pad is not on the **Current Layer** and is below other pads, then use Current Layer to choose the layer that contains the pad. Now choose Edit Change Pad and choose the pad. Use the options button if the changes should also be applied to other pads.

See Also: [**Changing Pads**](#)
[**Making Changes**](#)
[**Selection**](#)

Change Strings

Shortcut Keys:



Summary: Modifies any attribute of text. The modifications can be applied globally to other text.

Procedure: Choose a document window. If the string is not on the **Current Layer** and is below other strings, then use Current Layer to choose the layer that contains the string. Now choose Edit Change String and choose the string. Use the options button if the changes should also be applied to other strings.

See Also: [Changing Strings](#)
[Making Changes](#)
[Selection](#)

Change Tracks

Shortcut Keys:



Summary: Modifies any attribute of a track. The modifications can be applied globally to other tracks.

Procedure: Choose a document window. If the track is not on the **Current Layer** and is below other tracks, then use Current Layer to choose the layer that contains the track. Now choose Edit Change Track and choose the track. Use the options button if the changes should also be applied to other tracks.

See Also: [Changing Tracks](#)
[Making Changes](#)
[Selection](#)

Change Vias

Shortcut Keys:



Summary: Modifies any attribute of a via. The modifications can be applied globally to other vias.

Procedure: Choose a document window. If the via is not on the **Current Layer** and is below other vias, then use Current Layer to choose the layer that contains the via. Now choose Edit Change Via and choose the via. Use the options button if the changes should also be applied to other vias.

See Also: [**Changing Vias**](#)
[**Making Changes**](#)
[**Selection**](#)

Move Selection

Shortcut Keys:



Summary: Moves all selected items within a document window.

Procedure: Choose a document window. Position the cursor at the location that will be used as reference, the direction keys can be used to precisely align the cursor to the **snap grid**. If cursor is not locked (see Options Preferences), click at this location then choose Edit Move Move Selection. If the cursor is locked, leave the cursor at this position until the keyboard is used to choose Edit Move Move Selection.

See Also: [Selection](#)
[Edit Select](#)
[Edit De-Select](#)

Move Flip Selection

Shortcut Keys:





Summary: Mirrors the orientation of all selected items within the current document window.

Procedure: Choose a document window. Use Edit Select and Edit De-select as needed, then choose Edit Move Flip Selection.

See Also: [Selection](#)

[Edit Select](#)

[Edit De-Select](#)

Rotate Selection

Shortcut Keys:



Summary: Rotates all selected components within the current document window. The degree and center of rotation may be defined.

Procedure: Choose a document window. Use Edit Select and Edit De-select as needed, then choose Edit Move Rotate Selection.

The selection becomes free to rotate in 1 degree increments.

When you choose this command Protel for Windows will ask you for a number of degrees you wish to rotate. This command has a one 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. You may wish to change component pad shapes to round before rotating, unless you are rotating in increments of ninety degrees.

See Also: [Selection](#)
[Edit Select](#)
[Edit De-Select](#)

Break Track

Shortcut Keys:

Ctrl

Ctrl

Summary: Creates a vertex in a track at the chosen location. The vertex may be moved to a new location.

Procedure: Choose a document window. If the track is not on the **Current Layer** and is below other tracks, use Current Layer to choose the layer that contains the track. Now choose Edit Move Break Track and choose the track.

See Also: [Move track shortcuts](#)

Move Drag End

Shortcut Keys:



Summary: Moves the end point of a track to a new location.

Procedure:

1. Choose Edit Move Drag End.
You will be prompted "Select track." If the track is not on the **Current Layer** and is below other tracks, use Current Layer to choose the layer that contains the track.

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

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 to the new location and press  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.

See Also: [Move track shortcuts](#)

Move Whole Track

Shortcut Keys:



Summary: Moves an entire track (and all connecting track ends and vias). The track retains its orientation.

Procedure: If the track is not on the **Current Layer** and is below other tracks, use Current Layer to choose the layer that contains the track. Now choose Edit Move

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 ;

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 to the new location and press 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.

See Also: **Move track shortcuts**

Move track shortcuts


To select and move a track:

Hold down command and click left mouse once on a 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 

 together and click on the track.

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

Move Re-Route

Shortcut Keys:



Summary: Creates and moves a vertex in a track. The vertex may be placed in a new location. This sequence will continue as new vertices form after each placement.

Procedure: This feature is similar to the Break Track command. When re-routing, the cursor jumps to the nearest pad on the physical connection and the selected segment highlights. Each time you press 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

You will be prompted "Select Track."

2. Position the cursor over the track segment and press 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 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 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 ,



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).

To move both the complete track segment (and all connecting track ends and vias), choose a document window. If the track is not on the **Current Layer** and is below other tracks, use Current Layer to choose the layer that contains the track. Now choose Edit Move Re-Route and choose the track.

Move Arc

Shortcut Keys:



Summary: Moves an arc to a new location.

Procedure: Choose a document window. If the arc is not on the Current Layer and is below other arcs, use Current Layer to choose the layer that contains the arc. Now choose Edit Move Arc and choose the arc.

Move Component

Shortcut Keys:



Summary: Moves a **component** to a new location.

Procedure: Choose a document window. Now choose Edit Move Component and choose the component.

For SMD components, the component will be placed on the Top Layer, unless the **Current Layer** is specifically set to the Bottom Layer.

While moving a component, you can also rotate it around the cursor and flip it in the x and y axis about the cursor using the following keys



Rotates the component 90 degrees clockwise. Choose repeatedly to continue the rotation.



Flip the component in the X axis.



Flip the component in the Y axis

Move Fill

Shortcut Keys:



Summary: Moves a **fill** to a new location.

Procedure: Choose a document window. If the fill is not on the current layer and is below other fills, use **Current Layer** to choose the layer that contains the fill. Now choose Edit Move Fill and choose the fill.

If you select the edge of a fill, then it can be resized in the X or Y axis. If you select the corner, then the fill by resized in X and Y at the same time.

Move Pad

Shortcut Keys:

Ctrl

Ctrl

Summary: Moves a pad to a new location.

Procedure: Choose a document window. If the pad is not on the Current Layer and is below other pads, use Current Layer to choose the layer that contains the pad. Now choose Edit Move Pad and choose the pad.

Move String

Shortcut Keys:

Ctrl

Ctrl

Summary: Moves text to a new location.

Procedure: Choose a document window. If the string is not on the Current Layer and is below other strings, use Current Layer to choose the layer that contains the string. Now choose Edit Move String and choose the string.

Move Via

Shortcut Keys:



Summary: Moves a via to a new location. No tracks are dragged.

Procedure: Choose a document window. If the via is not on the Current Layer and is below other vias, use Current Layer to choose the layer that contains the via. Now choose Edit Move Via and choose the via.

Place Arc By Center

Shortcut Keys:

Ctrl

Ctrl

Summary: Places an arc. Center, start angle, radius, and end angle are defined during placement.

Procedure: Choose a document window and use the **Current Layer** to choose a layer for placement. Now choose Edit Place Arc (Center) and follow the prompts. The **Current Track Width** is used for the line width of the arc.

Change Layers: You can change layers with the Ctrl,

Ctrl or

Ctrl keys.

This command can also be selected by pressing the ARC button on the **Tool Bar**.

Tab

Sets the default track width on the fly.

Place Arc By Edge

Shortcut Keys:

Ctrl

R

Summary: Places of an arc. Start angle, radius, Center, and end angle are defined during placement.

Procedure: Choose a document window and use the **Current Layer** to choose a layer for placement. Now choose Edit Place Arc (Edge) and follow the prompts. The **Current Track Width** is used for the line width of the arc.

Change Layers: You can change layers with the Ctrl ,

Ctrl or

Ctrl keys.

Tab

Sets the default track width on the fly.

Place Component

Shortcut Keys:

Ctrl

Ctrl

Summary: Retrieves a library component for placement. During placement the component may be moved, rotated, or mirrored.

Procedure: Choose a document window and use the Place Component command. Enter a component pattern name, or enter a question mark to choose from a list of available component patterns. Follow the prompts.

When you choose this command you will be prompted for a component pattern. Enter a name (or ? to select from a displayed list). for example DIP16 or TO-3.

If the component you are placing is an SMT component, then the **current layer** should be the top or bottom layer. All components will be placed on the top of the board unless you go specifically to the bottom layer.

The program will now ask for the component designator. This is the actual name of the component, U1 for example. Note that this must be different from all other component designators currently used in the design. If not, then an error message will appear.

Designators can be up to 8 characters long and any alphanumeric character. It is suggested that you don't start designators with a number and that you do finish them with a number. This allows the auto numbering system to function at its best.

The program will supply you with a suggested designator based on the last component placed with the same pattern. If you wish you can enter Ctrl? or

Ctrl?. The program will then supply the next available number for this designator. For example, if the last designator starting with the letter Ctrl is U12 and you enter

Ctrl?, then the program will assign U13 to the component. If you just enter ? then the program will use the lowest letter (starting at A) with the lowest number (starting at 1). Now you will be asked for the component comment. This is the value or type of the component, 74LS123 or 100K for example.

Comments can be up to 32 characters long and any alphanumeric character. The program will supply you with a suggested comment based on the last component placed with the same pattern.

The component will now be loaded from the library and displayed on the screen.

Rotating: Move it around with the cursor, rotate with Ctrl or mirror with

Ctrl or

Y.

This command can also be selected by pressing the COMP button on the **Tool Bar**.

See Also: [Library Components](#)

Place Fill

Shortcut Keys:

Ctrl

Ctrl

Summary: Places a rectangular fill region.

Procedure: Choose a document window and use the **Current Layer** to choose a layer for placement. Now choose Edit Place Fill and follow the prompts.

Change Layers: You can change layers with the Ctrl,

Ctrl or

Ctrl keys.

This command can also be selected by pressing the FILL button on the **Tool Bar**.

Place Pad

Shortcut Keys:

Ctrl

Ctrl

Summary: Places a pad. Its attributes match those defined by the Current Pad. The pad may be moved and rotated during placement.

Procedure: Choose a document window and if placing a single layer pad, use the **Current Layer** to choose a layer for placement. Now choose Edit Place Pad and follow the prompts.

Use Ctrl to rotate.

SMD Pads: If you are placing an **SMT** pad or an edge connector (a single sided pad), then the **current layer** should be top layer or bottom layer. It is possible to place single layer pads on mid-layers, but this should only be done in special circumstances.

Change Layers:

You can change layers with the Ctrl,

Ctrl or

Ctrl keys.

Designator: Note that the designator of placed pads is set empty by default. To **place multiple pads** with incremental numbers, use the **Edit Change Pad** command to change the designator.

This command can also be selected by pressing the PAD button on the **Tool Bar**.

Tab

Sets the default pad type on the fly.

Place String

Shortcut Keys:

Ctrl

Ctrl

Summary: Places a line of **text**. During placement, text can be moved, rotated, or mirrored.

Procedure: Choose a document window and use the **Current Layer** to choose a layer for placement. Now choose Edit Place String and follow the prompts.

Change Layers: You can change layers with the Ctrl,

Ctrl or

Ctrl keys.

Size and Font: The string will have size and font as defined by **Current String**.

Moving: Move during placement with the cursor, rotate with Ctrl or mirror with

Ctrl or

Y.

This command can also be selected by pressing the STRING button on the **Tool Bar**.

Tab Sets the default string height on the fly.

This command can also be selected by pressing the STRING button on the **Tool Bar**.

Place Track

Shortcut Keys:

Ctrl

Ctrl

Summary: places tracks. During placement, each end point becomes a vertex and the starting point of a new track. Track width is predefined using the Current Track.

Procedure: Choose the document window and use the **Current Layer** to choose a layer for placement. Now choose Edit Place Track and follow the prompts.

The **Track Place Mode** can be changed on the fly with Ctrl.

Change Layers: You can change layers with the Ctrl,

Ctrl or

Ctrl keys. If the **Auto Via Option** is on, then when you change from one signal layer to another, a via will be inserted.

Tab

Sets the default track width on the fly.

Place Via

Shortcut Keys:

Ctrl

Ctrl

Summary: Places a via. The via can be moved during placement, its attributes are predefined using the **Current Via**.

Procedure: Choose a document window and use the **Current Layer** to choose a layer for placement. Now choose Edit Place Via and follow the prompts.

The layer pair, diameter and hole size are setup using the **Current Via Commands**.

Tab

Sets the default diameter on th fly.

Place Polygon Plane

Shortcut Keys:



Summary: Creates a plane of copper. clearances are maintained around all existing primitives and components and thermal ties are made to pads on a defined net.

Procedure: Choose the document window and use the Current Layer to choose a layer for placement. Now choose Edit Place External Plane and follow the prompts.

Place Coordinate

Shortcut Keys:

Ctrl

Ctrl

Summary: Places text that indicates the coordinates.

Procedure: Choose a document window and use the **Current Layer** to choose a layer for placement. Now choose Edit Place Coordinate String and follow the prompts.

Place Dimension

Shortcut Keys:

Ctrl

Ctrl

Summary: Places a dimension line that indicates the length of the line.

Procedure: Choose a document window and use the **Current Layer** to choose a layer for placement. Now choose Edit Place Dimension and follow the prompts.

Place Array

Shortcut Keys:



Summary: Generates a repeated placement of the contents of the clipboard. The placement can be defined as linear or circular.

Procedure: Choose a document window and select the primitives or components that will be placed in the array. Choose **Edit Copy** or **Edit Cut** to place the selection in the clipboard. Choose **Edit Place Array**, set the dialog box options and use the Place Array button.

The options are:

Circular/Linear Linear will place items in a straight line, circular will place in a circular array.

Count The number of items to be placed by repeat command.

X-offset The horizontal distance between the next placed and the last placed item.

Y-offset The vertical distance between the next placed and the last placed item.

Increment This is used for designators on pad. Setting this to 1 will give a series, for example A1, A2, A3 or 1, 2, 3. For example, to place a vertical row of pads for one half of DIP16 component, place the first pad manually. **Edit the pad** and set its designator to 1.

Cut the pad into the clipboard with the **Edit Cut** Command. Now Select Place Array, Set Y-offset to -100 mils X-offset to 0, text increment to 1, and linear. Set count to 8. Press the Place Array button and a row of pads will appear, descending down from the manually placed pad. There designators will be 1,2,3,4,5,6,7,8 from top to bottom.

See Also: **Selection**
 Edit Select
 Edit De-Select
 Edit Paste
 Edit Copy
 Edit Cut
 Edit Toggle Selection

Search For Component

Summary: Moves the cursor to the component's reference point.

Procedure: Choose a document window. Use the Edit Search For Component. Enter a component designator, or enter a question mark to choose from a list.

Search For Net

Summary: Moves the cursor to the nearest pad on a net.

Procedure: Choose a document window. Use the Edit Search For Net and enter a net name or enter a question mark to choose from a list.

Search For Pad

Summary: Moves the cursor to a pad.

Procedure: Choose a document window, then choose Edit Search For Component. Select the component on which to search, then enter a pin number.

Search For String

Summary: Moves the cursor to the text.

Procedure: Choose a document window. Use the Edit Search For String then enter the text.

The program will perform three searches when searching for a string.

1. Firstly for a string that matches the specified string in case, characters and length.
1. If not found then it searches for a string with same characters and length but ignoring case.
2. If still not found then it searches for a string with same characters in it but perhaps having more characters (and ignoring case).

For Example: Entering "Component" will find the string 'Component' first, if not found it will find the string 'COMPONENT' and if still not found it would find the string 'COMPONENT LAYER'.

Jump To Absolute Origin

Shortcut Keys:  

Summary: Moves the cursor to the lower left corner of the work space.

Procedure: Choose a document window, then choose Edit Jump Absolute Origin.

Jump To Current Origin

Shortcut Keys:  

Summary: Moves the cursor to the origin defined using the Edit Set Origin command.

Procedure: Choose a document window, then choose Edit Jump Current Origin.

See Also: [Edit Set Origin](#)

Jump New Location

Shortcut Keys:

Ctrl

Ctrl

Summary: Moves the cursor to new X and Y coordinates.

Procedure: Choose a document window, then choose Edit Jump New Location and follow the prompts.

Set Origin

Summary: Defines a location as the new origin and resets its coordinates to X0 Y0.

Procedure: Choose a document window. Choose Edit Set Origin, then choose the location for the new current origin.

See Also: **Jump Current Origin**

Component Libraries

Libraries in Protel for Windows, are collections of component foot prints (patterns). Each pattern is made up of a collection of items such as tracks, pads, fills etc. The process of creating a component is very simple.

- First: Place down all the items you want in the component, not forgetting to number any pads using the **Change Pad** command.
- Then: Select all the primitives using the Select All command. **Cut** or **Copy** the selected items into the clip board.
- Finally: Choose **Library Components**, select the library you wish to add to, and choose new to Add the selected primitives to the library.

Library Components

Shortcut Keys:

Ctrl

Ctrl

Summary: Allows creation, management, and browsing of pattern libraries.

Procedure: Choose Library Components. The Options are:

Add: Adds an additional library to the current list. The library file extension will be forced to LIB.

New: Creates a new (and empty) library, When the command is complete, the new library will added to the current list.

Display: Choosing this command will expand the dialog box. A window for viewing the components is shown together with zoom control buttons. From now on, as you select a component from the components list box, it will be drawn. You can zoom in and out using the Expand and Contract buttons. The XY coordinates displayed are the position of the cross hair in the center of the view window relative the reference point of the component.

Add: Adds all **selected** primitives to the selected library as a new component.

Compact: When you delete a component from the library, a gap is left in the library file, fragmenting it. This gap is not closed after every deletion (this would make the process very slow). This command will sort the library and remove any fragmentation.

Merge: Copies a set of components from one library file to the another. You will be asked to select first the source library and then the destination library. After choosing these you will be asked to select components from the source library to copy to the destination library. If the component already exists in the destination library then you can overwrite with a new copy.

Rename: Changes the name of a component pattern stored in a library.

Delete: Deletes a component pattern stored in the library. (You cannot undo this).

See Also: **Place Component**

Library Un-Group

Shortcut Keys:

Ctrl

Ctrl

Summary: Returns a component to its free primitives.

Procedure: Choose a document window. If the component is not on the Current Layer and is below other components, use the Current Layer to choose the layer that contains the component. Now choose Library Un-Group and choose the component.

Library Pad Types

Shortcut Keys:  

Summary: Allows generation and management of pad libraries.

Procedure: Choose Library Pad Types.

See Also: [Current Pad Type Pad Types](#)

Library Apertures

Shortcut Keys:



Summary: Allows generation and management of aperture tables.

Procedure: Choose a document window, then choose Library Apertures.

The Options Are:

Clear Choose Library Aperture Clear to remove any details of apertures currently stored in memory. If you have made any changes to these, then save your aperture list first. No changes or deletions are made to any aperture files on you disk.

Delete If you wish to remove an unused aperture from the aperture list Choose Library Aperture Delete and select the aperture to be deleted from the displayed list of all current apertures.

Edit See **Edit Aperture**

Load Choose Library Aperture Load to load an aperture file to the current list. When typing the file name you can use masks and select from a list.

Aperture files always have the extension APT. If you have made any changes to your current aperture list then save it before loading another list or you changes will be lost.

Merge If you wish, you can copy a single aperture, from another aperture file, into the current list. To do this, Choose Library Apertures Merge. Enter the name of the aperture file (as in Loading a list of apertures), then, when prompted, enter the draft code of the aperture to merge form the other file. When you have entered a valid draft code, the aperture is copied from the aperture file on disk to the list of apertures in memory.

New The program will ask you for a new draft code. When you have entered a valid new draft code, the program will move you to Edit.

Create List To create a list of apertures that match the current PCB designs tracks, pads, vias etc, Choose Library Apertures Create.

Save After you have changed the current aperture list (which is stored in memory) you will need to save it to disk to make the changes permanent. Choose Library Aperture Save to do this. The file name you specify will be forced to extension APT. If the file already exists you will be given the option to over write it.

See Also: **Apertures**

Aperture Edit

To modify an existing aperture Choose Library Aperture Edit and select the aperture to be edited from the displayed list of all apertures in the current list. The aperture parameters which can be changed are

Draft Code	This is the aperture name. It can be any code from D04 to D999.
Hole Size	This is the size of the hole in the center of the aperture image. For no hole set this to zero.
Shape	The options for shape are, circular, rectangular, Octagonal, rounded rectangular, cross hair target or moire target.
X Size	This is the horizontal size of the aperture in mils (or millimetres). Any value up to 32000 can be used for this.
Y Size	This is the vertical size of the aperture in mils (or millimetres). Any value up to 32000 can be used for this.
Use	This specifies whether the aperture can be used to draw tracks, flash pads or both.

Netlist Load

Shortcut Keys:



Summary: Opens a netlist, retrieves missing components from the parts libraries, and assigns thermal relief connections to the plane layers.

Procedure: Choose a document window, use the Netlist Power Planes to assign nets to the plane layers as needed. Choose Netlist Load.

See Also: [Library Components](#)

[Netlist Clear](#)

[Netlist Power Planes](#)

[Netlists](#)

Netlist Clear

Shortcut Keys:  

Summary: Removes the netlist from memory.

Procedure: Choose a document Window, then choose Netlist Clear.

See Also: [Netlist Load](#)
[Netlist Power Planes](#)
[Netlists](#)

Optimize Net

Shortcut Keys:



Summary: Reorders the all netlist connections on a net. The sorting method is predefined using Netlist Edit Net command.

Procedure: Choose a document window, then choose Netlist Optimize Net. Enter the name of the net or enter a question mark to choose from a list.

See Also: [Optimize Methods](#)
[Netlist Edit Net](#)

Optimize On Component

Shortcut Keys:



Summary: Reorders the netlist connections on a Component. The sorting method is predefined using Netlist Edit Net.

Procedure: Choose a document window. If the component is not on the **Current Layer** and is behind other components, use the current layer to choose the layer that contains the component. Now choose Netlist Optimize On Component and choose the component.

See Also: **Optimize Methods**
Netlist Edit Net

Optimize All

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Reorders all the netlist connections within the current document window. The sorting method is predefined using the Netlist Design Rules or the Netlist Edit Net.

Procedure: Choose a document window, then choose Netlist Optimize All.

See Also: [Optimize Methods](#)
[Netlist Edit Net](#)

Show Connections On Net

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Displays the unrouted ratsnest connections on a named net (only if unrouted).

Procedure: Choose a document window, then turn on the Rats Nest layer using the Options Layers. Choose Netlist Show Connections Net, then enter a net name or enter a question mark to choose from a list.

See Also: Options Layers
Netlists

Show Connections On Component

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Displays the unrouted Ratsnest connections on a component (only if unrouted).

Procedure: Choose a document window then turn on the Rats Nest layer using the Options Layers. Choose Netlist Show Connections On Component, then choose a component.

See Also: Options Layers
Netlists

Show All Connections

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Displays all unrouted ratsnest connections on all nets within the current document window (only if unrouted).

Procedure: Choose a document window then turn on the Rats Nest layer using the Options Layers. Choose Netlist Show Connections All.

See Also: Options Layers
Netlists

Hide Connection

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Turns off the display of a ratsnest connection.

Procedure: Choose a document window. Choose Netlist Hide Connections Connection and choose a ratsnest connection.

See Also: Options Layers
Netlists

Hide Connections On Net

Shortcut Keys: 





Summary: Turns off the display of the ratsnest connections on a net.

Procedure: Choose a document window then choose Netlist Hide Connections Net. Enter a net name or enter a question mark to choose from a list.

See Also: [Options Layers](#)
[Netlists](#)

Hide Connections On Component

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Turns off the display of ratsnest connections on a component.

Procedure: Choose a document window. If the component is not on the **Current Layer** and is behind other components, use the current layer to choose the layer that contains the component. Now choose Netlist Hide Connections On Component and choose a component.

See Also: **Options Layers**
Netlists

Hide All Connections

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Turns off the display of ratsnests connections on all nets within the current document window.

Procedure: Choose a document window then choose Netlist Hide Connections All.

See Also: Options Layers
Netlists

Netlist Edit Net

Shortcut Keys:

 or







Summary: Modifies the attributes assigned to a net. These include Net Name, Routing Track Width, Routing Via Diameter, Optimize Method, and Routing Priority. The modifications can be applied globally to all other nets.

Procedure: Choose a document window then turn on the Rats Nest layer using the Options Layers. Display the ratsnest using the Netlist Show. Choose Netlist Edit Net, then choose a ratsnest connection.

See Also: [Options Layers](#)
[Netlists](#)
[Netlist Show](#)
[Netlist Hide](#)

Netlist Identify

Shortcut Keys:



Summary: Displays the net name for a chosen ratsnest connection.

Procedure: Choose a document window then turn on the Rats Nest layer using the Options Layers. Display the ratsnest using the Netlist Show. Choose Netlist Identify, then choose a ratsnest connection.

See Also: Options Layers

Netlists

Netlist Show

Netlist Hide

Netlist Length

Shortcut Keys:

Ctrl

Ctrl

Summary: Displays the total length of all ratsnest connections within the current document window.

Procedure: Choose a document window, then choose Netlist Length.

See Also: Netlists

Netlist Generate

Shortcut Keys:

Ctrl

Ctrl

Summary: Produces a netlist from the current document window. The netlist produced by this method has no connection with the netlist loaded from a net file. It is purely based on the electrical connection between component pads, based on the physical layout of the board. You can use this method to get a netlist from a manually routed file (for later submission to an autorouter for example).

Procedure: Choose a document window, then choose Netlist Generate.

Netlist Export

Shortcut Keys:

Ctrl

Ctrl

Summary: Produces a netlist from the current document window. The netlist produced by this method is purely base on the loaded nets. Physical tracks, vias etc are not considered. If you have deleted some components, then any nodes in a net that referred to that component, will not be written with the netlist.

Procedure: Choose a document window, then choose Netlist Generate.

Netlist DRC

Shortcut Keys:



Summary: Produces a design rule check report and displays the areas of violation. The rules are predefined using the Netlist Design Rules. The types of violations that are reported may be defined.

Procedure: Choose a document window, then use the Netlist Design Rules to set clearances. Choose Netlist DRC, set the report options and use the OK button.

See Also: [Netlist Clearances](#)

Netlist Clearances

Shortcut Keys:



Summary: Allows definition of clearances for routing and design rule checking.

Procedure: Choose Netlist Design Rules.

See Also: [Netlist Design Rule Check](#)

Netlist Power Planes

Shortcut Keys:



Summary: Assigns net names to the plane layers. Pins on these nets are automatically assigned relief connections during any netlist load.

Procedure: Choose a document window, then choose Netlist Power Planes.

See Also: [Netlist Load](#)

Auto Place

Shortcut Keys:



Summary: Arranges parts within an area outlined on the keepout layer.

Procedure: Choose a document window. Use the Netlist Load to load the netlist and retrieve the components, then choose Auto Auto Place.

See Also: Netlist Load
Auto Placement

Auto Placement

Auto Placement Theory

Advanced Placer

**Additional Comments on the Advanced Placer
Placement Tools**

Auto Placement Theory

Placement is a critical process in manufacturing a PCB. The success in manufacturing is almost completely dependent on a good placement technique.

Placement involves specifying the exact location and orientation of each component on the PCB, given the board size and the area of the board on which placement is permissible.

The most important objective in placement, after the fitting of all the components on the board without violating any design rules, is to make the job of routing easier or, in most cases, possible!

There is also one more consideration which is not as critical as the others, and this is the aesthetics of the board.

In short, the main goals in placement of a PCB are :

- Fitting all the components on the board.

- Allowing no design rules violations.

- Creating a routable board.

- Creating a board with a good aesthetics.

Theory of optimization

Global and Local Optimizations

Optimisation for Automatic Placement

Theory of optimization

Optimization is a formal technique used by mathematicians to find the extremes of a function. In mathematical terms this problem can be stated as follows:

Given the function $f(x)$ with x belonging to the set s and s being the set of all the possible x 's find a w from the set s for which $f(w)$ is a maximum (or a minimum).

Solving this problem has turned out to be one of the most difficult areas in mathematics. The complexity of this problem is governed by the type of the function, and the size and dimension of s .

There are hosts of different techniques developed by mathematicians and other scientists to solve this problem. Most of them, however, only work on a small subset of the real-world problems.

If $f(x)$ is an analytical function, using differential calculus maxima and minima of $f(x)$ can be found. This method is the fastest in terms of the computational time required. Moreover it always guarantees a solution can be found. For most real-world problems $f(x)$ is not an analytical function and consequently this method can not be used.

One optimization technique which is almost diametrically opposite to the analytical method, uses an exhaustive search to find the solution. What exhaustive search means, is to try all the possible x 's and pick the one which gives the maximum for the function (or the minimum). This method also guarantees to produce the best solution. By contrast, it can be applied to any problem regardless of the type of the function with the heavy price of needing some geological time to carry out the computations involved for most of the real-world problems.

Although the real-world problems seem to be almost impractical for the optimization techniques to tackle they have, however, one major handy attribute which allows us to successfully apply an optimization technique to a real-world problem.

That attribute is that in a real-world problem, often we are not after the maximum or minimum, rather an approximation to them. In other words, we are not after the x which gives the maximum(or the minimum), rather an x which produces a high enough(or low enough in case of minimisation) value for f which can satisfy the problem in hand.

This aspect of the real-world problems has given rise to a spectrum of different techniques in between the two extremes, ie. the analytical and the exhaustive search methods. Most techniques in that range make a compromise between the computational time and the quality of the approximation. they generally attempt to find a good solution by searching a very small subset of the set of all possible points in the solution space. The criteria by which a small subset is selected for search is the most intriguing part of optimization problem, and often the cause of diversity in the methods of optimization.

Global and Local Optimizations

Approximation of a solution is a very loose goal when we do not have any idea about the solution! How can we approximate a number when we do not know anything about the magnitude of that number? All we know is, "the higher the better"(or "the lower the better" in the case of minimisation). The answer to this is that most of the optimization techniques only find the best out of the ones which have been tried, and that, they only try the ones which are expected to make an improvement . A method like this can, in no wise, claim to be able of finding one of the best solutions. This problem is commonly known as the problem of Local Maxima (or Local Minima in the case of minimisation).

An analogy to this is the case of a blind man wanting to get to the highest point in a valley. Starting from a random point on the valley, he only moves to a direction where he can ascend. It is obvious that the chances for this poor man to get to the highest point in that valley is very small. In fact, a small hill which happens to be near his starting point can fool him and drag him to its top and making him believe he can not get any higher, as there is no point near by the top of that hill which is of a higher altitude. The techniques exhibiting this or similar kind of problems are commonly referred to as local optimization or greedy methods. By contrast the methods which find one of the best solutions are commonly referred to as global optimization methods.

Optimisation for Automatic Placement

Developing a Function for the Problem

Choosing an Optimization Technique

Simulated Annealing Method of Optimization

Developing a Function for the Problem

In order to apply optimization to the problem of automatic placement we first need to define a function which can measure the goodness of a placement job. In this case our variable x is a proposed placement job and s is the set of all the possible placements. It is obvious what the next move is!

We use a suitable optimization method to find the placement which gives us a high value of goodness.

This function, depending on a particular method of optimization employed, is referred to as the Cost Function, Objective function, evaluation function, fitness evaluator, etc. We have used the term Routing-Difficulty Function.

This function measures the following :

- Connection length.
- Connection density on the board.
- Component's alignment.
- Design rules violations.

Our goal is to minimize this function.

It is very important to realize that the overall success of an optimization technique is highly dependent on how good this function models the actual problem.

Optimization Techniques

Now we have to choose an optimization technique which results in success. Our policy is to find a global solution rather than a local one. In the context of that blind man we want to get to the highest point or some point very close to the highest. This means we have to employ a global optimization technique in order to minimize the Routing-Difficulty function.

There are only a few global optimization techniques and almost all of them have been developed very recently. Amongst them are the simulated annealing method, simulated evolution method and genetic algorithms. These methods all, have received inspiration from nature as their names suggest. In the annealing process of a metal, atoms array themselves in a way that crystals of low energy are formed. In natural evolution, only those species which are of better fitness survive and in genetics through cross-over and mutation, on average, better chromosome are produced from generation to generation.

The optimization technique which was chosen, was the Simulated Annealing method.

Simulated Annealing Method of Optimization

In a metal raised to a temperature above its melting points, the atoms are in violent random motion. As with all physical systems, the atoms tend toward minimum energy state(a single crystal in this case), but at high temperatures the vigour of atomic motion prevents this. As the metal gradually cooled, lower and lower energy states are assumed until finally the lowest of all possible states, global minimum.

A simple way of describing the application of this to the PCB placement problem is to consider each component as an atom in the metal in the annealing process. Now all we need is some artificial temperature, base on which, we can control the random movements of components. Starting at a high temperature, allowing components to move freely, we gradually drop the temperature and thereby applying some loose restriction on the moves. As the temperature drops lower we apply more restriction until we get close to the freezing point around which only moves which result in improvements are allowed.

Advanced Placer

The first thing that could be said, in describing the PFW advanced placer is that it is easy to use. Despite the underlying complexity, the PFW placer appears quite simple to the eyes of the user. All the user needs to know, before he or she runs the placer, is some very basic facts about the design. The user at no time needs to ever worry about configuring the placer. The placer has been finely tuned over a wide spectrum of boards for an optimal performance.

Setting Up the Advanced Placer

Preplacing Components

Board Area for Placement

Advanced Placer Window

Advanced Placer Results

Background Running

Auto Place Setup

To set up the advanced placer, the user needs to run the Autoplace Set Up Command. PFW responds by displaying the Autoplace Dialogue Box. The following information is needed by the placer:

- The Placement Grid Size.
- The allowable Clearance for small components.
- The allowable Clearance for large components.
- The names of the power and ground nets. (if used)

The power and ground nets are required by the placer in order to detect the by-pass capacitors. The by-pass capacitors are not directly placed by the advanced placer. The placer allows some space for the by-pass capacitors on top of each chip on the board.

There are four options in the Autoplace Set Up Dialogue Box, they are:

Local Placement
Global Placement
Component Rotation
Grouping

Local Placement

The Local Placement option is an independent Greedy Autoplacer. It is an improved version of the Protel Autotrax Autoplacer.

Global Placement

The Global Placement option is the PFW advanced placer.

Auto Place Grouping

The Grouping option, if enabled, causes the placer to go through all the components on the board and group together all the components which are tightly connected. The placer then performs a relative placement on each group. These groups are treated by the placer as super components and their relative placement is kept untouched during the main placement cycle.

This option is generally useful. It could be counter productive if there is not going to be enough space on the board. This is so because the relative placement within each group is not changed during the main placement cycle and the placer might waste some space in order to accommodate the group.

Board Area for Placement

This can be done by putting tracks in the keep-out layer. If there is no track found in the keep-out layer, the entire area of the board is assumed. To disallow placement on any particular area on the board, area fills in the keep-out layer can be used.

Advanced Placer Window

The advanced placer has its own window. When running, it displays the current state of the board. It draws the rats nets, components and the keep-out areas (any of them can be set, using the display menu.) The size of the components displayed includes the clearances specified in the Autoplace Dialogue Box. The main menu for the advanced placer is very short. The only specific options in the main menu are the Display and the UpdatePCB option in the File menu.

The advanced placer has its own data structure. This means the PCB is not changed normally during the placement. The user, however, can update the PCB to the most current state of the board by selecting the UpdatePCB option in the File menu.

The placement window has its own status bar, within which, the following information is available :

Elapsed time	Time since start of placement.
Placement Cycle	The Placement Cycle refers to the annealing cycle. There are a total of 70 cycles in any placement task. The first 40-50 cycles are very fast as most moves are accepted. However, as the temperature decreases, more and more moves are made in order to satisfy the requirements of a cycle. This means the cycles get slower towards the completion.
Number of Moves	The Number of Moves, displayed in the status bar, is the total number of times that the placer has moved a component to a new position in order to improve the routability of the board.
Routing Difficulty	The Routing Difficulty is the value of the cost function for the current board. This cost reaches a very high value after a few cycles. It then continues to decrease until a global minimum is reached.

Advanced Placer Results

Before the placer starts to run, the user is asked to provide a name for the PCB file for saving the placement results. The placer, at the completion, updates the current PCB and saves the PCB in the file whose name was provided by the user at the start of the placement.

Auto Place Background Running

One interesting feature of the Advanced Placer is that it runs in the background. While placement is in progress, the user can work on another PCB or run another application. The user can not, however, run the placer for the second time in the same application. To have two placements running at the same time, the user needs to run a

second copy of the main application. The user, while the placer is in progress, can not close the PCB window or close the application. Once the placement is finished a dialogue box appears to inform the user of the completion of the placement.

Additional Comments on the Advanced Placer

The Advanced Placer is highly robust and generally does not require any special direction by the user. However, should any difficulty arise, the following points could provide useful information.

Preplacing Components

Use of the keep-out Area Fills

Rotation Option

Grouping Option

Large nets slow down the placer

Preplacing Components

The user can preplace any component before running the Advanced Placer. The placer leaves untouched all the components that are specified by the user as being preplaced. In order to specify that a component has been preplaced, the Place Status of that component needs to be set to 'Locked in Place'. This can be done through the Change Component Command. The Place Status for all the other components needs to be set to 'Free to Move'.

Apart from those components which have to be placed on certain locations on the board, such as edge connectors and heat sinks, etc., sometimes it is useful to preplace a component which has no restriction on its placement. For example, it might be desirable to preplace the memory chips. This could facilitate the placement of the other components. In the case of a group of analogue components connected together, it again might be beneficial to preplace them.

Use of the keep-out Area Fills when auto placing

This could be very useful when it comes to routing. It is advisable, in most cases, to put keep-out area fills all around the board. This will give some open space for the router to go around. It is also good to put keep-out area fills next to connectors and some preplaced components.

Auto Place Rotation

The Component Rotation option only applies to the Global Placement, where as the Grouping option applies to the both Autoplacers.

If component rotation is allowed, the Advanced Placer rotates components in order to find an optimal orientation. It only considers four orientations, namely, 90,180,270 and 360 degrees.

The Rotation option can sometimes make the placement task more difficult. Rotation in general requires more board space. Therefore, in those boards where there is not going to be a lot of open space, rotation could be counter-productive. On the other hand, if the board space is large the rotation option can be highly useful.

Auto Place Large Nets

Large nets slow down the placer. The reason for this is the computation involved with the rearranging of a net, is exponentially proportional to the net size. An interesting observation is the insignificant role of these large nets in the overall placement. Therefore it could be advantageous to hide the large nets from the placer. To do this one can either save the net-list, remove the large nets, run the placer and at the end reload the original net-list, or assign some of the large nets to the power and ground planes and thus hide them from the placer. When supplied by the user, the power and ground nets are not passed to the placer, as power and ground nets are usually connected to most of the components.

Placement Tools Menu

Shortcut Keys:



 or



Summary: The outcome of the Advanced Placer is a board, in which, the relative positions of the components are optimal. Due to its global nature, the Advanced Placer often produces boards which are not entirely polished. For instance, there could still be some overlaps after the placement is completed, or some components might not be aligned properly. These anomalies need to be removed. The placement tools are specifically designed to facilitate this phase.

Spread Horizontal

Spread Vertical

Align Left

Align Right

Align Top

Align Bottom

Center On Horizontal

Center On Vertical

Expand Horizontal

Expand Vertical

Contract Horizontal

Contract Vertical

Shove

Set Shove Depth

Auto Place Spread Horizontal

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Spread out selected components horizontally.

The Placement Grid can be set to a new value in the Autoplace Dialogue Box.

Procedure: De-Select any selected components.

Select the components to spread and choose Auto Placement Tools Spread Horizontal.

The selected components will be spread evenly horizontal.

See Also: [Selection](#)
[Auto Placement](#)

Auto Place Spread Vertical

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Spread out selected components vertically.

Procedure: De-Select any selected components.

Select the components to spread and choose Auto Placement Tools Spread Vertical.

The selected components will be spread evenly vertically.

See Also: [Selection](#)

[Auto Placement](#)

Auto Place Align Left

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Move all selected components to align with the left hand side of a chosen component.

Procedure: De-Select any selected components.

Select the components to align and choose Auto Placement Tools Align Left.

You will be asked to choose a component. The one you choose will be used as the reference and all the other selected components will be move to align with its left hand side. No Vertical movement will be made.

See Also: [Selection](#)
[Auto Placement](#)

Auto Place Align Right

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Move all selected components to align with the right hand side of a chosen component.

Procedure: De-Select any selected components.
Select the components to align and choose Auto Placement Tools Align Right.

You will be asked to choose a component. The one you choose will be used as the reference and all the other selected components will be move to align with its right hand side.

No Vertical movement will be made.

See Also: [Selection](#)
[Auto Placement](#)

Auto Place Align Top

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Move all selected components to align with the top of a chosen component.

Procedure: De-Select any selected components.

Select the components to align and choose Auto Placement Tools Align Top.

You will be asked to choose a component. The one you choose will be used as the reference and all the other selected components will be move to align with its top.

No horizontal movement will be made.

See Also: [Selection](#)
[Auto Placement](#)

Auto Place Align Bottom

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Move all selected components to align with the bottom of a chosen component.

Procedure: De-Select any selected components.

Select the components to align and choose Auto Placement Tools Align bottom.

You will be asked to choose a component. The one you choose will be used as the reference and all the other selected components will be move to align with its left hand side.

No horizontal movement will be made.

See Also:

[Selection](#)

[Auto Placement](#)

Auto Place Center On Horizontal

Shortcut Keys:



Summary: Move all selected components to align their centres on a horizontal line which passes through the centre of a chosen component.

Procedure: De-Select any selected components.

Select the components to align and choose Auto Placement Tools Center On Horizontal.

You will be asked to choose a component. The one you choose will be used as the reference and all the other selected components will be moved so as to align their centres on a horizontal line which passes through the centre of the reference.

See Also: [Selection](#)
[Auto Placement](#)

Auto Place Center On Vertical

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Move all selected components to align their centres on a vertical line which passes through the centre of a chosen component.

Procedure: De-Select any selected components.

Select the components to align and choose Auto Placement Tools Center On Vertical.

You will be asked to choose a component. The one you choose will be used as the reference and all the other selected components will be moved so as to align their centres on a vertical line which passes through the centre of the reference.

See Also: [Selection](#)
[Auto Placement](#)

Auto Place Expand Horizontal

Shortcut Keys:



Summary: Move selected components apart horizontally so as to increase the spacing between them by one autoplacement grid point.

Procedure: De-Select any selected components.

Select the components to expand and choose Auto Placement Tools Expand Horizontal.

The selected components will be moved apart horizontally so as to increase the spacing between them by one autoplacement grid point.

See Also: [Selection](#)
[Auto Placement](#)

Auto Place Expand Vertical

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Move selected components apart vertically so as to increase the spacing between them by one autoplacement grid point.

Procedure: De-Select any selected components.

Select the components to expand and choose Auto Placement Tools Expand Vertical.

The selected components will be moved apart vertically so as to increase the spacing between them by one autoplacement grid point.

See Also: [Selection](#)
[Auto Placement](#)

Auto Place Contract Horizontal

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Move selected components closer together horizontally so as to decrease the spacing between them by one autoplacement grid point.

Procedure: De-Select any selected components.

Select the components to expand and choose Auto Placement Tools Contract Horizontal.

The selected components will be moved closer together horizontally so as to decrease the spacing between them by one autoplacement grid point.

See Also: [Selection](#)
[Auto Placement](#)

Auto Place Contract Vertical

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Move selected components closer together vertically so as to decrease the spacing between them by one autoplacement grid point.

Procedure: De-Select any selected components.
Select the components to expand and choose Auto Placement Tools Contract Vertical.

The selected components will be moved closer together vertically so as to decrease the spacing between them by one autoplacement grid point.

See Also: [Selection](#)
[Auto Placement](#)

Auto Place Shove

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Moves all components of a selected component. And then move components of the moved components.

Procedure: The Shove command causes all the components which overlap the selected component to slide away and leave the selected component clear. It then attempts to shove all the slided components and so on.

It is not advisable to use the Shove command for intricate jobs as the 'Shove and Push' process involved could upset the overall placement. The scope of the Shove command, however, can be limited by setting the Shove Depth to a small value. Shove Depth can be set through the **Set Shove Depth command**.

For example, if the Shove Depth is set to 1, only the selected component is cleared - the components which were cleared off the selected components are not shoved again.

Although exceedingly dangerous to the health of your board, if the depth is set large, you can undo shoves.

Auto Place Set Shove Depth

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Used to limit the scope of the shove depth command.

Procedure: Choose Auto Auto Place Set Shove Depth. Enter a number greater or equal to 1. This is the depth of expansion. If this is set to 2 for example, the chosen component will move all the components on it. The the moved components will move all the components on them.

See Also: [Auto Place Shove](#)

Auto Move To Grid

Shortcut Keys:

Ctrl

Ctrl

Summary: Moves each free to move component to the nearest location that will align its reference point with the grid.

Procedure: Choose a document window. Choose Auto Move To Grid, choose a grid, or choose other to enter another value.

Auto Density

Shortcut Keys:



Summary: Temporarily displays a map of the ratsnest connection density.

Procedure: Choose a document window. If a netlist is not loaded, use Netlist Load. Choose Auto Density.

This is a relative color display. Red is the most dense area, and green is the least dense area.

See Also: [Netlist Load](#)

Manual Route

Shortcut Keys:







Summary: Places tracks manually using a ratsnest connection.

Procedure: To manually convert a connection to a series of tracks and vias, Choose Manual from the router command menu and select a connection to work on. The cursor will jump to the nearest end of the connection. Manual routing works in a very similar way to the **Place Track** command, except that as you lay tracks you will see a rubber band line from the current cursor position to the other end of the connection.

You can use the 

 and

 keys to change layers and vias will be inserted automatically as you go.

Press

 or



to place a track section. When you are ready to place the last section (and complete the connection), Press

 or



A track section will run from the current cursor position to the end of the connection. You can manually route another, or Press

 or



to finish manual routing.

Autoroute All

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Routes all connections on all nets within the current document window.

Procedure: Select Auto Autoroute All, the route status dialog will appear and the router will attempt to route all the unrouted connections on the board.

See Also: [Autorouting](#)

Autoroute Net

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Automatically routes a net.

Procedure: To select a net and autoroute all the connections on it, Choose Auto Autoroute Net. Enter a net name, or enter ? to select from a list. All currently selected routing passes will be tried on all the unrouted connections in the selected net.

See Also: [Autorouting](#)

Autoroute Connection

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Automatically routes a ratsnest connection.

Procedure: To select individual connections and autoroute them, Choose Auto Autoroute Connection. When prompted, select a connection. All currently selected routing passes will be tried on the connection.

See Also: [Autorouting](#)

Autoroute On Component

Shortcut Keys: 





Summary: Automatically routes all ratsnest connections on a component.

Procedure: To select a component and autoroute all the connections on it, Choose Auto Autoroute On Component Select a component. All currently selected routing passes will be tried on all the connections that go to that component (not nets, just individual connections).

See Also: [**Autorouting**](#)

Autoroute Pad to Pad

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Automatically routes tracks between two pads. This autorouting function does not require a netlist.

Procedure: Choose Auto Autoroute Pad to Pad. Select 2 pads and the router will attempt to route them. The maze router is not used by this pass.

See Also: [Autorouting](#)

Setup Auto Route

Shortcut Keys:

Ctrl

Ctrl

Summary: Defines use of autorouting algorithms and attributes.

Procedure: Choose Auto Setup Autorouter. The autorouter setup dialog box will appear.
Use this to set the routing layers, routing passes and router variables.

See Also: [Autorouting](#)

Un-Route All

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Removes all tracks and vias on all valid netlist connections within the current document window.

Procedure: Choose a document window, then choose Auto Un-Route All.

See Also: [**Autorouting**](#)

Un-Route Net

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Removes all free primitives on all valid netlist connections on a net.

Procedure: Choose a document window. Choose Auto Un-Route Net and follow the prompts.

See Also: [Autorouting](#)

Un-Route Connection

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Removes all free primitives on a valid netlist connection.

Procedure: Choose a document window. Choose Auto Un-Route Connection, then choose a free primitive.

See Also: [Authorouting](#)

Un-Route On Component

Shortcut Keys: 





Summary: Removes free primitives on valid netlist connections on a component.

Procedure: Choose Auto Un-Route On Component, then choose a component.

See Also: [**Autorouting**](#)

Un-Route Selection

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Removes all selected free primitives on valid netlist connections within the document window.

Procedure: Choose a document window. Use Edit Select and Edit De-Select as necessary, then choose Un-Route Selection.

See Also: [Selection](#)





[Edit Select](#)

[Edit De-Select](#)

[Autorouting](#)

Current Pad Type

Shortcut Keys:

 or
 or
 +


Summary: Defines the pad attributes that will be in effect during subsequent pad placements.

Procedure: Choose Current Pad Type.

This command allows you to pick one of the pad types from the current pad type list. . When you **place a pad**, this pad type will be placed.

The current pad type is also displayed on the status line.

See Also: [Library Pads Types](#)

Current Track

Shortcut Keys:



Summary: Defines the track width that will be in effect during subsequent manual track placements.

Procedure: Choose Current Track, choose a track width, or choose Other to enter another value between 1 and 255 mils. This value is the width of tracks placed with the **Place Track** Command. The current track width is also displayed on the status line.

Current Via Diameter

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Defines the via diameter that will be used during subsequent manual via placements, and manual routing.

Procedure: Choose Current Via Diameter, choose a size or choose Other to enter another value.

See Also: [Current Via Hole Diameter](#)

Current Via Hole Diameter

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Defines the via hole diameter that will be used during subsequent manual via placements, and manual routing.

Procedure: Choose Current Via Hole Diameter, choose a size or choose Other to enter another value.

See Also: [Current Via Diameter](#)

Via Type Through Hole

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Ctrl

Summary: Defines Through Hole as the type of via that will be in effect during subsequent manual via placements and manual routing.

Procedure: Choose Current Via Type Through Hole.

See Also: [Current Via Type Blind/Buried](#)

Via Type Blind/Buried

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Ctrl

Summary: Defines Blind/Buried as the type of via that will be in effect during subsequent manual via placements and manual routing.

Procedure: Choose Current Via Type Blind/Buried.

See Also: [Current Via Type Through Hole](#)

Component Text Comments

Shortcut Keys:



Summary: Defines the component comment attributes that will be in effect during subsequent component placements.

Procedure: Choose Current Component Text Comments.

This dialogue box controls the settings used for displaying the text of component comments after they have been placed.

All these options are defaults, you can change them later using the move and change commands.

The Options Are:

Layer: Component designators can be either copper (on the top layer for a top layer component) or overlay (on the top overlay for a top layer component).

Height: The height of the designators displayed letters (in mils or millimetres).

Font: The displayed text of the designator or comment can be displayed in any one of the system fonts available.

Hide Status: The show attribute can be set to either Hidden or Show. If hidden, then the text will not be shown on either the screen or on subsequent plots. Note that no information is lost by showing or hiding the designator, and that it can be changed at some later time.

Vertical Align: Controls the vertical position of the text relative to the component.

Horizontal Align: Controls the Horizontal position of the text relative to the component.

See Also: [Current Component Text Designators](#)

Component Text Designators

Shortcut Keys:



Summary: Defines the component designator attributes that will be in effect during subsequent **component placements**.

Procedure: Choose Current Component Text Designators.

This dialogue box controls the settings used for displaying the text of component designators after they have been placed.

All these options are defaults, you can change them later using the move and change commands.

The Options Are:

Layer: Component designators can be either copper (on the top layer for a top layer component) or overlay (on the top overlay for a top layer component).

Height: The height of the designators displayed letters (in mils or millimetres).

Font: The displayed text of the designator or comment can be displayed in any one of the system fonts available.

Hide Status: The show attribute can be set to either Hidden or Show. If hidden, then the text will not be shown on either the screen or on subsequent plots. Note that no information is lost by showing or hiding the designator, and that it can be changed at some later time.

Vertical Align: Controls the vertical position of the text relative to the component.

Horizontal Align: Controls the Horizontal position of the text relative to the component.

See Also: **Current Component Text Comments**

Free Text Height

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Defines the text height that will be in effect during subsequent **placements of free text**.

Procedure: Choose Current Free Text Height, choose a size or choose Other to enter another value between 36 and 4000 mils.

Font Default

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Ctrl

Summary: Defines that the font in effect during subsequent **placements of free text** will be the internal Protel for Windows font.

Procedure: Choose Current Free Text Font Default.

Font Sans Serif

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Ctrl

Summary: Defines that the font in effect during subsequent **placements of free text** will be the font that is mapped to Sans Serif.

Procedure: Choose Current Free Text Font Sans Serif,

Font Serif

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Ctrl

Summary: Defines that the font in effect during subsequent **placements of free text** will be the font that is mapped to Serif.

Procedure: Choose Current Free Text Font Serif.

Current Snap Grid

Shortcut Keys:  

Summary: Defines the alignment grid for manual movement and placement of objects.

Procedure: Choose a document window. Choose Current Snap Grid, then choose a size or choose Other to enter another value.

See Also: [Snap Grid](#)

Current Visible Grid 1

Shortcut Keys:  

Summary: Defines the first display grid.

Procedure: Choose Current Visible Grid 1, then choose a size or choose Other to enter another value.

See Also: [Visible Grid](#)

Current Visible Grid 2

Shortcut Keys:



Summary: Defines the second display grid.

Procedure: Choose Current Visible Grid 2, then choose a size or choose Other to enter another value.

See Also: [Visible Grid](#)

Current Layer

Shortcut Keys:

Ctrl

Ctrl

Summary: Displays and changes to a layer, even if display is toggled off.

Procedure: Choose a document window. Choose Current Layer, then choose a layer.

Any layer can be selected. If you select a layer that is toggled off, then it will be turned on. The current layer is displayed on the Tool Bar, together with a color box to help you identify which items are on the layer. When you use the edit, move, delete or other design commands, the current layer is the layer searched first during selections. You can change layers at any time during editing (except when menus or dialogs are on the screen) with the Ctrl,

Ctrl or

Ctrl keys.

Ctrl switches to the next signal layer that is turned on.

Ctrl and

Ctrl move down and up through the turned on layers respectively.

See Also

Options Layers

Layer and Display Colors

Layer and Display Colors

The Color that layers are displayed in is controlled by the **L**ayers dialog box from the options menu.

Some of the specialized settings are:

Back Ground: The physical background Color.

Connections: The Color of the rats nest. Connections are generated, from the net list, by the **netlist optimize** command.

Drc Errors: Primitives that have failed clearance checks made by DRC are drawn outlined in this color.

Selection: All highlighted items are drawn outlined in this Color.

Grid Color 1, 2: The colors of the 2 **Visible Grids**. These are only display if the lines will be 5 or pixels apart.

Options Layers

Shortcut Keys:



Summary: Toggles layer display and defines colors.

Procedure: Choose a document window, then choose Options Layers.

Transparent You can 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 during the final clean-up of the board.

Dithered Dithered colors are created by mixing two solid colors. Leave this option off if you want to display solid colors only. This can be helpful if you are displaying primitives in Draft mode (Options Display command).

All Layers Off Provides a shortcut for turning all the layers off in one-step.

Used Layers On Restores the layer selections to include all the layers in the current PCB file.

All Layers On Activates every layer with a single command.

Current Layer On Activates the Current layer. Leaves other layer selections undisturbed.

See Also: [Layer and Display Colors](#)
[Setting up the workspace layers](#)

Setting up the workspace layers

Normally you will want to start a new project by selecting the layers needed for the design. In PCB this will include the signal (or Copper Trace) layers, Internal Power Plane layers, Mechanical layers, Silkscreen Overlay layers, and Special layers used to display details of your design.

Before we 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 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 PCB. Any layers that you have activated will be active the next time you start the PCB application. 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 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 color assignment used if the Dithered Colors option is not selected (Dithered colors are colors derived by mixing two solid colors). Depending upon your graphics card and display type, you can choose from up to 16 solid and 48 dithered colors.

3. Click in one of the 64 color boxes to re-set the layer assignment;
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`.

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.

See Also: **Customizing Colors**

Customizing Colors

To customize one of the 64 available colors from the Color Selector dialog box:

1. Choose Custom Selections - shortcut: **Ctrl**
Ctrl - to expand the Color Selector.

There are three methods for customizing a color:

RGB You can set the Red/Green/Blue values between 0 and 255 units;

HSL You can set the Hue, Saturation and Luminance between 0 and 255 units;

Palette You can click in the color or grayscale palettes to set the color directly. 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 re-draw.

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 title bar and the layer name is displayed (along with the filename, if any) in the title bar.

See Also **[Color assignments](#)**

Color assignments

On the bottom of the Set-up Colors dialog box is another command 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 re-set to display as many available colors as possible when you close the Setup Layers dialog box.

Windows reserves the first 20 colors in the palette for its own system use. 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 Protel for Windows.

On 16 color VGA systems, all 16 colors are set by Windows for the system palette, which cannot be re-programmed by the user. On EGA systems there are only 16 (from a 64 palette) colors, so the quality of the standard windows display is poor and no reprogramming is possible.

Options Preferences

Shortcut Keys:

Ctrl

Ctrl

Summary: Defines various system settings. These options affect all document windows.

Procedure: Choose a document window, then choose Options Preferences.

See Also

Cursor Shape

Component Track Drag Mode

Auto Pan

Auto Via Insertion

Delete Confirmation

Draft/final Mode

Tracks, Arcs, Fills, Pads, Strings or Vias, can be draw as hollow, solid or hidden. If hidden, they will not be display and cannot be selected as individual items.

Cursor Shape

Choose Options Shape to select the type of cursor to use for editing in the work space. This does not affect the cursor which is used for working on menus and dialogue boxes.

Component Track Drag Mode

This option controls what happens to the tracks connected to a component when the component is moved. If set, when you select a component to move, all of the tracks connected to it will be dragged, maintaining 45/90 line, with it.

Auto Pan

Choosing Options Auto Pan controls what happens when you touch the window edge with the cursor (while selecting or dragging an item). If this is on, the when you hit the edge, the display will Pan in the required direction.

See Also [**Zoom menu**](#)

Auto Via Insertion

While placing tracks, you can change layers and have two tracks on two different layers. The program will automatically, if this option is set, place a via at the join of these to track ends.

Delete Confirmation

Every time you delete an item, the program will ask for confirmation. To control this option, Choose Options Preferences and Check or Un-Check then Delete Confirmation check box.

Options Display

Shortcut Keys:

Ctrl

Ctrl

Summary: Defines display attributes for each type of primitive. Three modes allow displaying, not displaying, or displaying primitives as outlines. These attributes affect all document windows.

For pads, the Name and Net Name of the pad can also be displayed.

Procedure: Choose a document window, then choose Options Display.

See Also **Display Mode (Draft/Final/Hidden)**

Track Mode Any Angle

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Changes the manual track placement mode to permit track placement at any angle.

Procedure: Choose a document window, then choose Options Track Mode Any Angle.

Track Mode 90/90 Line

Shortcut Keys:

Ctrl

Ctrl

9

Summary: Changes the manual track placement mode to permit track placement at only 90 degree angles.

Procedure: Choose a document window, then choose Options Track Mode 90/90 Line.

Track Mode 45/90 Line

Shortcut Keys:

Ctrl

Ctrl

4

Summary: Changes the manual track placement mode to permit track placement at either 45 or 90 degree angles.

Procedure: Choose a document window, then choose Options Track Mode 45/90 Line.

Track Mode 90 Arc/Line

Shortcut Keys:

Ctrl

Ctrl

Ctrl

Summary: Changes the manual track placement mode to permit track placement at 90 degree angles or arcs.

Procedure: Choose a document window, then choose Options Track Mode 90 Arc/Line.

Options Tool Bar

Shortcut Keys:

Ctrl

Ctrl

Summary: Toggles the Tool Bar display on or off.

Procedure: Choose Options Tool Bar.

Options Status

Shortcut Keys:

 Ctrl

 Ctrl

Summary: Toggles the Status Line display on or off.

Procedure: Choose Options Status.

Options Scroll Bars

Shortcut Keys:  

Summary: Toggles the Scroll Bars display on or off.

Procedure: Choose a document window, then choose Options Scroll Bars.

Zoom Window

Shortcut Keys:



Summary: The region chosen within two opposite corners is displayed to fill the document window.

Procedure: Choose a document window. Choose Zoom Window and follow the prompts.

See Also: [Zoom Point](#)

Zoom Point

Shortcut Keys:

Ctrl

Ctrl

Summary: The region chosen by a center point and a corner is displayed to fill the document window.

Procedure: Choose a document window. Choose Zoom Point and follow the prompts.

See Also: [Zoom Window](#)

Zoom Select

Shortcut Keys:



Summary: The display magnification of the chosen document window is changed to reflect a chosen value. The range of values is 1 to 400. 1 is the most zoomed in.

Procedure: Choose a document window. Choose Zoom Select and Choose a magnification level or choose other to enter another value.

Zoom All

Shortcut Keys:

 or

+





Summary: The region defined by objects at the highest and lowest coordinate points is displayed to fill the current document window. In order to utilize this feature with a new document, you must first use the **Info Board** command.

Procedure: Choose a document window, then choose Zoom All.

Zoom In

Shortcut Keys:

 or






Summary: Increases display magnification in the current document window to the next higher default value.

Procedure: Choose a document window, then choose Zoom In.

Zoom Out

Shortcut Keys:



 or



Summary: Decreases display magnification in the current document window to the next lower default value.

Procedure: Choose a document window, then choose Zoom Out.

Zoom Pan

Shortcut Keys:

 or






Summary: Redraws the current document window with the previous cursor location at the center of the window.

Procedure: Choose a document window, then choose Zoom Pan.

Zoom Redraw

Shortcut Keys:

 or





Summary: Redraws the current document window.

Procedure: Choose a document window, then choose Zoom Redraw.

Info System Status

Shortcut Keys:

 Ctrl

 Ctrl

Summary: Displays disk and memory status.

Procedure: Choose a document window, then choose Info System Status.

Info Board Status

Shortcut Keys:

Ctrl

Ctrl

Summary: Displays the quantity of components and primitives within the current document window.

Procedure: Choose a document window, then choose Info Board Status.

Info Components on PCB

Shortcut Keys:



Summary: Displays a list of placed components within the current document window.

Procedure: Choose a document window, then choose Info Components on PCB. To save the list, double click on any component in the list. Enter a valid filename to save the list as an ASCII file or enter the DOS reserved name PRN and the list will be dumped directly to the printer port.

The format is: U1 (74LS123) Designator (Type/Value/Comment)

Info Selected Pins

Shortcut Keys:

Ctrl

Ctrl

Summary: Displays the component and pin designators of selected pins within the current document window.

Procedure: Choose a document window, then choose Info Selected Pins. To save the list, double click on any pin name in the list. Enter a valid filename to save the list as an ASCII file or enter the DOS reserved name PRN and the list will be dumped directly to the printer port.

The entries are sorted by component designator first and pad designator second.

The format is: U1-16 Component designator - Pad designator

Info Nets

Shortcut Keys:

Ctrl

Ctrl

Summary: Displays the names of nets within the current document window.

Procedure: Choose a document window, then choose Info Nets. To save the list, double click on any net name in the list. Enter a valid filename to save the list as an ASCII file or enter the DOS reserved name PRN and the list will be dumped directly to the printer port.

Info Measure Distance

Shortcut Keys:

Ctrl

Ctrl

Summary: Displays the distance between two chosen points.

Procedure: Choose a document window. Choose Info Measure Distance and follow the prompts.

Info Length of Selection

Shortcut Keys:  

Summary: Displays the total physical connection length of selected primitives within the current document window.

Procedure: Choose a document window. Use Edit Select and Edit De-Select as necessary, then choose Info Length of Selection.

See Also: [Edit Select](#)
[Edit De-Select](#)
[Selection](#)

Power Planes Plane 1

Shortcut Keys:

Ctrl

Ctrl

1

Summary: Displays the designators of all pins with direct or thermal relief connections on Plane 1 within the current document window.

Procedure: Choose a document window, then choose Info Power Planes Plane 1. To save the list, double click on any pin name in the list. Enter a valid filename to save the list as an ASCII file or enter the DOS reserved name PRN and the list will be dumped directly to the printer port.

Power Planes Plane 2

Shortcut Keys:

Ctrl

Ctrl

2

Summary: Displays the designators of all pins with direct or thermal relief connections on Plane 2 within the current document window.

Procedure: Choose a document window, then choose Info Power Planes Plane 2. To save the list, double click on any pin name in the list. Enter a valid filename to save the list as an ASCII file or enter the DOS reserved name PRN and the list will be dumped directly to the printer port.

Power Planes Plane 3

Shortcut Keys:

Ctrl

Ctrl

3

Summary: Displays the designators of all pins with direct or thermal relief connections on Plane 3 within the current document window.

Procedure: Choose a document window, then choose Info Power Planes Plane 3. To save the list, double click on any pin name in the list. Enter a valid filename to save the list as an ASCII file or enter the DOS reserved name PRN and the list will be dumped directly to the printer port.

Power Planes Plane 4

Shortcut Keys:

Ctrl

Ctrl

4

Summary: Displays the designators of all pins with direct or thermal relief connections on Plane 4 within the current document window.

Procedure: Choose a document window, then choose Info Power Planes Plane 4. To save the list, double click on any pin name in the list. Enter a valid filename to save the list as an ASCII file or enter the DOS reserved name PRN and the list will be dumped directly to the printer port.

Window Tile

Shortcut Keys:

 Ctrl

 Ctrl

Summary: Displays all non-minimized document windows without overlapping.

Procedure: Choose Window Tile.

Window Cascade

Shortcut Keys:



Summary: Arranges all document windows and overlaps all except the title bars.

Procedure: Choose Window Cascade.

Window Arrange Icons

Shortcut Keys:



Summary: Arranges all document icons across the lower portion of the application window.

Procedure: Choose Window Arrange Icons.

Window Close All

Shortcut Keys:



Summary: Closes all Document windows and icons, prompting to save any modified documents.

Procedure: Choose Window Close All.

See Also: File Save, File Save As

Help Index

Shortcut Keys:



 or



Summary: Displays a list of help topics, these offer access to all areas of the help system.

Procedure: Choose Help Index.

Help Using Help

Shortcut Keys:

Ctrl

Ctrl

Summary: Displays information about using the help system.

Procedure: Choose Help Using Help.

Help Basic Concepts

Shortcut Keys:

Ctrl

Ctrl

Summary: Displays an index of basic operation topics, these offer access to detailed information about Protel for Windows techniques.

Procedure: Choose Help Basic Concepts.

Help Commands

Shortcut Keys:

Ctrl

Ctrl

Summary: Displays the Advanced PCB Commands index, these offer access to detailed information about each command.

Procedure: Choose Help Commands.

Help Printing

Shortcut Keys:

Ctrl

Ctrl

Summary: Displays an index of printing topics, these offer access to detailed information about printing and plotting.

Procedure: Choose Help Printing.

Help Managing Files

Shortcut Keys:

Ctrl

Ctrl

Summary: Displays an index of file management topics, these offer access to detailed information about using file in Protel for Windows.

Procedure: Choose Help Managing Files

Help Screen Regions

Shortcut Keys:

Ctrl

Ctrl

Summary: Displays an index of screen area topics, these offer access to detailed information about using the Protel for Windows interface.

Procedure: Choose Help Screen Regions

Help Reference

Shortcut Keys:

Ctrl

Ctrl

Summary: Displays an index of reference topics, these offer access to detailed information about these topics.

Procedure: Choose Help Reference.

Help About

Shortcut Keys:

Ctrl

Ctrl

Summary: Displays the application name and version number.

Procedure: Choose Help About.

Tool Bar

The tool bar, when displayed, appears below the menu bar. It contains buttons and tools to speed up operations.

You can use the tool bar to change layer, redraw the screen, zoom on a section of the PCB and place items.

Layer box

Displays the current layer for the active window. You can change the layer by selecting the layer you want from the list.

Zoom Button

Choose this button to select an area of the screen to zoom into. Same as Zoom Window.

Placement

Use these buttons to place tracks, Vias, Pads, Components, Fills and Text strings on the PCB.

Redraw Button

Use this to update the screen display. No changes of viewing position or window are made. Same as Zoom Redraw.

Window Sizing Border

The border around a window that mouse users can drag to change the window's shape and size.

Drag the top, bottom, or side border to size in one direction only.

Drag a corner of the border to size in two directions at once.

You can also size a window by using the Control Size command.

Application Window Control Menu

- Restore** Restores the active window to its former size and location.
- Move** Displays a four-headed arrow you can use to reposition the active window or an open dialog box that has a Control menu.
- Size** Displays a four-headed arrow you can use to change the size of the active window.
- Maximize** Enlarges the active window to fill the available space.
- Close** Closes the active window or an open dialog box that has a Control menu.
- Switch To** Lists all currently open applications and activates the one you select. This command appears only on the application Control menu.
- Minimize** Reduces the application window to a small icon at the bottom of the screen.

PCB Window Control Menu

- Restore** Restores the active PCB window to its former size and location.
- Move** Displays a four-headed arrow you can use to reposition the active window or an open dialog box that has a Control menu.
- Size** Displays a four-headed arrow you can use to change the size of the active window.
- Maximize** Enlarges the active PCB window to fill the available Application work space.
- Close** Closes the active PCB window or an open dialog box that has a Control menu.
- Next Window** Activates the next open PCB window.
- Minimize** Reduces the PCB window to a small icon at the bottom of the screen.

Application Control Menu Close

Closes the active application window or PCB Window window. If a dialog box is open and has a Control menu, choosing this command closes the dialog box.

For mouse users, double-clicking the Control Menu icon is the same as choosing Control Close.
Closing the application window is the same as choosing File Exit.

If you close a PCB Window window, any other open windows for that PCB Window remain open
To close all open PCB Windows, use the Window Close All command.

Shortcuts +

 (closes PCB Window window)

+

 (closes application window)

Application Control Menu Control Maximize

Enlarges the active window to fill the available space.

For mouse users, clicking the Maximize icon is the same as choosing Control Maximize.


Expands the Protel for Windows window to fill the entire screen.

Expands a PCB Window to fill the entire Protel for Windows workspace.

You can restore a maximized window to its former size by using the Control Restore command.

Shortcut



 (maximizes the PCB Window)

Application Control Menu Control Minimize

Reduces the Protel for Windows window to an icon at the bottom of the screen.

For mouse users, clicking the Minimize icon is the same as choosing Control Minimize.

Minimizing is useful when you are running another application at the same time as Protel for Windows.



You can restore a minimized window to its former size by using the Control Restore command.

Application Control Menu Control Move

Displays a four-headed arrow you can use to move the active window.

If a dialog box is open and has a Control menu, you can also use this command to move the dialog box.

If the window is maximized, this command is unavailable.


Shortcut +
 to move the PCB Window

Application Control Menu Control Size

Displays a four-headed arrow you can use to change the size of the active window.

If the window is maximized, this command is unavailable.

Shortcut +

 (resizes PCB Window window)

Application Control Menu Control Restore

Restores the active window to the size and location it had before you chose the Control Maximize or the Control Minimize command.

With the mouse, click the Restore icon of the window you want to restore.

Does not affect changes made to a window with the Control Move or Control Size command.

Shortcut 

 (restores PCB Window window)

Application Control Menu Switch To

Lists all the open applications.

Switch To Activates the selected application.

End Task Closes the selected application.

Cancel Closes the Task List window.

Cascade Sizes all open applications so they overlap and each title bar is visible.

Tile Arranges all open applications and sizes them so that they are all visible and do not overlap.

Arrange Icons Arranges the icons of all minimized applications across the bottom of the screen.

PCB Window Control Close


Closes the active application window or PCB Window window. If a dialog box is open and has a Control menu, choosing this command closes the dialog box.

For mouse users, double-clicking the Control Menu icon is the same as choosing Control Close.
Closing the application window is the same as choosing File Exit.


If you close a PCB Window window, any other open windows for that PCB Window remain open

To close all open PCB Windows, use the Window Close All command.

Shortcuts

 (closes PCB Window window)



 (closes application window)



PCB Window Control Maximize

Enlarges the active window to fill the available space.

For mouse users, clicking the Maximize icon is the same as choosing Control Maximize.

Expands the Protel for Windows window to fill the entire screen.

Expands a PCB Window to fill the entire Protel for Windows workspace. You can restore a maximized window to its former size by using the Control Restore command.

Shortcut 
 (maximizes the PCB Window)

PCB Window Control Minimize

Reduces the Protel for Windows window to a small icon at the bottom of the screen.

For mouse users, clicking the Minimize icon is the same as choosing Control Minimize.

Minimizing is useful when you are running another application at the same time as Protel for Windows.

You can restore a minimized window to its former size by using the Control Restore command.


Window Control Move

Displays a four-headed arrow you can use to move the active window.

If a dialog box is open and has a Control menu, you can also use this command to move the dialog box.

If the window is maximized, this command is unavailable.

Shortcut 


 to move the PCB Window

PCB Window Control Size

Displays a four-headed arrow you can use to change the size of the active window.

If the window is maximized, this command is unavailable.

Shortcut 

 (resizes PCB Window window)

Ctrl+CB Window Control Next Window

Activates the next PCB Window window .

Shortcut Ctrl+

F6

PCB Window Control Restore

Restores the active window to the size and location it had before you chose the Control Maximize or the Control Minimize command.

With the mouse, click the Restore icon of the window you want to restore.

Does not affect changes made to a window with the Control Move or Control Size command.

Shortcut 

 (restores PCB Window window)

Managing Files

Change the drive or directory

Change to a subdirectory of the current directory

To change to the parent directory of the current directory

Close the active PCB Window

View a list of files in a directory

View PCB files in a directory

View all files in a directory

To change the drive or directory

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

To change to a subdirectory of the current directory:

- 1.** Choose File Open.
- 2.** In the Directories box, select a subdirectory.
- 3.** Choose OK.
- 4.** To close the dialog box, choose Cancel.

To change to the parent directory of the current directory:

- 1.** Choose File Open.
- 2.** In the Directories box, select "[..]".
- 3.** Choose OK.
- 4.** To close the dialog box, choose 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, Protel for Windows asks if you want to save changes.
- 2.** Choose Yes to save changes, No to discard changes, or Cancel to cancel the command. If the file hasnt 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.** Choose OK.

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

- 5.** Choose Yes to continue, choose No to enter another name, or choose Cancel to cancel the command and return to the PCB Window.

To view a list of files in a directory:

- 1.** Choose File Open.
- 2.** In the File Name box, type a file specification.
It can include drive, directory, and the wildcard characters "?" (to match a single character) and "*" (to match any number of characters).
- 3.** Choose OK. The Files box displays the names of all files matching the file specification you type.
- 4.** When you have finished viewing the list, choose Cancel.

To View all PCB files in a directory

- 1.** Choose File Open.
- 2.** In the File Name box, type *.PCB (asterisk, period, PCB).
- 3.** Choose OK.
- 4.** When you have finished viewing the list, choose Cancel.

To View all files in a directory:

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

Screen Regions

Control Menu Icon

Maximize Icon

Minimize Icon

Restore Icon

Scroll Bar

Status Bar

Title Bar

Tool Bar

Window Sizing Border

Control Menu Icon

Mouse users can click the Application Control Menu or PCB Window Control Menu icons to display commands that control the size and position of the windows.

The Control Menu icons are located at the left end of the title bar in the application and PCB windows.

See Also: **PCB Window Control Menu**
 Application Control Menu

Restore Icon

Mouse users can click the Restore icon to return a window to the size and location it had before it was sized.

Restoring an application window also restores any PCB Windows that are open within the application.

Clicking the Restore icon has the same effect as choosing the Control Restore command.

It does not affect changes made to a window with the Control Move or Control Size command.

Scroll Bar

The shaded bars along the right and bottom sides of a window.


Mouse users can use the scroll bars to move through a PCB Window or Help topic.

The position of the scroll box in the scroll bar indicates the part of the PCB Window ,list or topic currently displayed in the window.

To scroll quickly to another part of the PCB Window or topic, drag the box.

To scroll one quarter of a window size in a given direction, click the arrow at that end of the bar.

To scroll one full window size in a given direction, click in the bar on that side of the scroll box.

To scroll past the area of the PCB Window you are currently using and view the entire PCB Window, hold down  while you drag the scroll box.

Maximize Icon

Mouse users can click the Maximize icon to enlarge a window to its maximum size.

A maximized PCB Window window fills as much of the Protel for Windows window as possible and has no borders.

You can maximize the Protel for Windows window only if it is not full screen, or if it has been minimized to an icon.

Clicking the Maximize icon has the same effect as choosing the Control Maximize command.

To restore a maximized window to its former size, choose Control Restore.

With the mouse, click the Restore icon.

Minimize Icon

Mouse users can click the Minimize icon to shrink the Microsoft Protel for Windows window to a small icon.

As an icon, the application remains in memory but does not take up room on your desktop.

When you minimize the Protel for Windows window, any open PCB Windows are minimized with it.

Clicking the Minimize icon has the same effect as choosing the Control Minimize command.

To restore the Protel for Windows window, and any PCB Windows that were open when you minimized it, choose Control Restore.

With the mouse, click the Restore icon.

Status Bar

The status bar at the bottom of the screen displays information about the current activity or mode.

When a command is selected, the left side of the status bar briefly describes the chosen command.

The left side of the status bar also indicates the current cursor position.

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 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.

To increase memory

Use as many of the following procedures as practical. After each step below, you can try again to complete the task that requires more memory.

Quit any other applications you are running.

Choose File Save All.

Close all windows except the active window.

In the Options Menu, turn off Scroll Bars, Tool Bar, and status bar.

Quit, and then restart Windows and Protel for Windows.

As a last resort, quit Protel for Windows and Windows, then restart your computer. If possible, leave off networks, TSRs, and any other utilities you usually run.

Pad Types

In Protel for Windows, each pad has all of its own details, size shape etc, stored with it. To avoid having to enter this information each time we place a pad, we use pad types and the current pad type list. This is a list of pad types, each with a name (Round50 for example), which is kept in memory. We can save this list to a pad type file and reload it later on. This allows us to build a library of pad types. For example we may keep a list of SMT pads, for use in designing SMT boards, and a list of standard multilayer pads for standard boards.

In the Library Pads menu there are a series of commands to be used to Set-up, edit, load, save and generally control the use of pad types. When placing a pad we are using the current pad type (one of the pad types in the current list).

The pad type list, referred to in this section, is the list of pad types that are currently active. When the program starts up it attempts to load a pad type file called PFW.PAD. This contains a list of default pad types. To change the default list (the one loaded at the start) simply save your list to this file name.

Clear Choose Library Pads Clear to remove any details of pad types currently stored in memory. If you have made any changes to these, then save your pad type list first. No changes or deletions are made to any pad type files on you disk.

Delete If you wish to remove an unused pad type from the pad type list Choose Library Pads Delete and select the pad type to be deleted from the displayed list of all current pad types.

Edit See [Pad Type Edit](#)

Load Choose Library Pads Load to load a pad type file to the current list. Pad type files always have the extension .PAD. If you have made any changes to your current pad type list then save it before loading another list or your changes will be lost.

Merge If you wish, you can copy a single pad type, from another pad type file, into the current list. To do this, Choose Library Pads Merge. Enter the name of the pad type file (as in Loading a list of pad types), then, when prompted, enter the name of the pad type to merge from the other file. When you have entered a name, the pad type is copied from the pad type file on disk to the list of pad types in memory.

New To create a new pad type and add it to the current pad type list, Choose Library Pads New. The program will ask you for a new name. When you have entered a name, the program will move you to Edit Pad type with a set of default pad type parameters. **See also:** Editing a Pad type (above).

Save After you have changed the current pad type list (which is stored in memory) you will need to save it to disk to make the changes permanent. Choose Library Pads Save to do this. The file name you specify will be forced to extension PAD. If the file already exists you will be given the option to over write it.

Pad Type Edit

To modify an existing pad type Choose Library Pads Edit and select the pad type to be edited from the displayed list of all pad types in the current list. The pad type parameters which can be changed are

- Name** A 10 character identifier, ROUND100, for example.
- Hole Size** This is the size of the hole in the center of the pad type. For no hole (SMT or Edge connector) set this to zero.
- Shape** The options for shape are, rectangular, octagonal and rounded.
- X Size** This is the horizontal size of the pad type in mils (or millimetres). Any value up to 1000 can be used for this.
- Y Size** This is the vertical size of the pad type in mils (or millimetres). Any value up to 1000 can be used for this.
- Layer** This specifies whether the pad type is single layer (SMT or edge connector) or multilayer. Double sided edge connectors require two separate pads. For single layer pads, the actual layer (top or bottom) is not decided until the actual placement, where it takes on the current layer. Single layer pads can only be placed on the top or bottom layers.

Print

Introduction to Printing
Printer Setup
Generating Prints & Plots

Printing Introduction

- Windows Microsoft Windows provides a set of raster and postscript printer drivers. These range from Epson 9 pin dot matrix printers to Linotronic high resolution raster image processors (photo-typesetters).
- PFW Printing The Protel for Windows print routines will print all the layers (including DRC errors, rats nest etc), that you have currently got turned on. Tracks, pads, vias, arcs, fills, and text strings will be plotted in draft or final mode according to how you have them displayed on the screen. Choose **Zoom All** before running the printer routine.

Printing

When the printer is configured and you have the appropriate layers switched on or off, then Choose File Print to generate the print.

If you have selected the file output device, then you will be asked for a file name.

The program will now display a series of overlapping rectangles on the screen and will ask you to confirm the print. These rectangles represent the pages (or strips of pages for a continuous feed printer) that the board layout will be printed on. Their size reflects the scale and correction factors that you have set, and the page size of the printer type that you have selected. The board will be shown centered in the set of rectangles shown. If the Set-up is acceptable, then select Yes and the print will be generated. If you wish to change the scale (to fit it on one page for example), then select No and you will be returned to the printer Set-up dialogue box.

While the print is being generated, the status line will report the current completion. As each area on the board is printed, the screen display of that area will change Color.

You can interrupt or pause the print at any time. Choose any key and a menu will appear asking you whether you wish to stop the print or continue it.

Printer Setup

Scale Choose Scale and choose 1x or 2x or select Other and enter any scale you like.

Corrections These two values are multipliers for all co-ordinates sent to the printer. The X-correction is multiplied with all horizontal values and the y-correction is multiplied with all vertical values. Use these values to correct any linear errors in either axis of the printer. If you are only using the printer for check prints (not final artwork), then you can leave these set at 1x.

File Printer Setup dialog box

- Printer** Select the printer you want to use. Only those printers installed in Windows appear. Use the Control Panel to add printers.
- Setup** Displays a printer driver specific dialog box with options that change to match the printer. Every version contains at least the following options:
- Orientation** Portrait prints down the length of the paper. Landscape prints across the length of the paper.

Photoplotting

[Photoplotting Background](#)
[Gerber Photoplotter Language](#)
[Apertures](#)
[Photoplotter Setup](#)
[Photoplotting](#)

Photoplotting Background

All Photoplotters print images directly onto photographic film. In this way they are closer to the desired result since no photo-reduction or further photography is required.

Photoplotters fall into two classes :

Vector Plotters

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.

Raster Plotters

These work in much the same way as your screen, or a Laser printer. They construct a bit map image of the object to be printed. This bit map is then translated to the film by some form of optical scanner.

Gerber Photoplotter Language

It is a fact of evolution that nearly all photoplotters receive their commands in a vector based language developed by The Gerber Scientific Instrument Company. It is this fact which creates the few difficulties and limitations to be experienced by the photoplotting designer.

Apertures

Any given photoplotter supports a number of drawing tools or apertures. In the case of a vector plotter these apertures are real optical identities which are projected onto the film emulsion by the plotter's optical system. In the case of a raster plotter, the apertures are synthesized and represented by a bit map.

See Also [**Using Apertures**](#)
[**Aperture Library**](#)

Aperture Use

To make photoplotting as reliable and easy as possible, only use track and pad sizes and shapes for which there is a corresponding aperture. Configure your pad type list to match your aperture list.

It follows that if the designer familiarises himself with the aperture set supported by the target photoplotter, and tailors the choice of objects placed on a PCB design (what pad and track size he uses) accordingly, then the photoplotter will be able to faithfully reproduce precisely what the designer had in mind.

However, it is not always desirable to limit the range of choices to the family of apertures supported by an arbitrary photoplotter. Some system must be provided to allow the successful interpretation of any pad shape or track size that the designer feels he must use.

The program will select apertures (if they fit within set tolerances) and modify the design data base to match the final photoplot.

Photoplotter Setup

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.

Software Arc

Some photoplotters do not support a specific arc drawing command. Protel for Windows can generate a series of line segments to simulate an arc. Consult with your photoplot bureau first, and then turn this on or off as they suggest. The hardware arc is only used for arcs that are in multiples of 90 degrees. For small arc angle, the series of line segments is always used.

Tolerances

If Protel for Windows uses an aperture that does not match exactly, then the design data base will be changed. In this example, the 52 mil track will be replaced in the data base by a 50 mil track. If you do not want the changes, then save you file before plotting.

While the program is generating the photoplot, it will look for an aperture to match each item. For example, if it finds a 52 mil track on the board, but only has a 50 mil aperture, what should it do ?

The two tolerances control this. In the above example, the closest aperture is 2 mil smaller than track. If the minus tolerance is set to 2 mil or larger, then the 50 mil aperture can be used.

Generating Photoplots

The Protel for Windows photoplot routines will photoplot all the physical layers that you have currently got turned on. Tracks, pads, vias, arcs, fills, and text strings will always be photoplotted in final mode.

When the photoplotter is configured and you have the appropriate layers switched on or off, then Choose File Gerber Gerber Out to generate the plots.

NC Drilling

Introduction to NC-Drilling
Generating NC Drill files

NC-Drill Use

The use of numerically controlled (NC) drilling machines has provided advantages for both board manufacturers and board developers. By supplying the board maker with a control file which he can send directly to the drilling machine, to drill all the holes in the board, you can save the cost of manually programming the drill, and remove the danger of holes being missed on a complex board. There are however a number of dangers to be considered.

The golden rule is, if you are generating and using NC drill files, then your plots of the board must be very accurate, preferably photoplotted.

Firstly, NC drills are accurate. If your plot is not accurate, then the holes will not be drilled in the center of the pads. 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 is programmed from a CAD generated file, then the NC Drill will put the hole in exactly the correct position, up to 0.9mm from where it appears on the plot master.

NC Drill files

When creating NC files, Protel for Windows will generate three files types with the same name and different extensions.

You should not modify the report file. If the hole sizes are incorrect, then use the global editing facilities in Protel for Windows to change the hole sizes in the PCB design. This will insure that the PCB design file matches the actual board.

The control file (Extension DRL)

This is in the EIA character format (this is machine readable, not human readable) and is in the EXCELLON language. The data is specified in mils with trailing zero supChooseion. The drill tools are simply numbered from 1 up to 255. This is the file which is actually sent to the drilling machine.

The ASCII file (Extension TXT)

This file contains the same information as the EIA file described above but is stored in ASCII character format and hence is readable.

The report file(Extension DRR)

This file contains the list of drill numbers and tools sizes, hole counts, and travel lengths. All in inches and millimetres. Print this file out, and give it the board maker, together with the NC drill control files.

Special Files

- PFW.EXE** The main program file.
- PFW.INI** This is used to store all your current settings from one session to another. This file doesn't come with the program but is created the first time you run the program. If you want to have several default set-ups, for different kinds of PCB job, then copy this file to another file name and copy it back before running the program.
- PFW.PAD** This is the default set of pad type definitions. Protel for Windows will search for this at start up. If not found then no components will be able to be placed. To change the default set, simply save your set of pad type definition to the file PFW.PAD
- PFW.LIB** The default library of patterns for components on the PCB. Protel for Windows will search for this at start up. If not found then no components will be able to be placed until another library is specified.

File Types

file.PCB Used to store a PCB layout.

file.NET Net list file.

file.DRC Design rule check result file.

file.BOM Bill of material file.

file.AB0 to AB9 This is the auto back-up file used to store the layout you are working on.

Photoplotter output files

file.GTL	Gerber format top layer
file.GBL	Gerber format bottom layer
file.G1 to file.G16	Gerber format mid layer 1 to mid layer 16
file.GTO	Gerber format top overlay
file.GBO	Gerber format bottom overlay
file.GP1 - file.GP4	Gerber format internal power plane
file.GTS	Gerber format top solder mask
file.GBS	Gerber format bottom solder mask
file.GTP	Gerber format top paste mask
file.GBP	Gerber format bottom paste mask
file.GTA	Gerber format top placement layer
file.GBA	Gerber format bottom placement layer
file.GPM	Gerber format pad master
file.GKO	Gerber format keep out layer
file.GDD	Gerber format drill drawing
file.GDG	Gerber format drill guide
file.GM1 - file.GM4	Gerber format mechanical layers

Selection

The Protel for Windows sytem allows you to change the elements of your design in two ways:

Choose a command such as Move, then select an item to be moved, or;
Select one or more items then choose a command which acts on the selection such as Move, Clear, etc.

Selection is simply the process of choosing one or more items, upon which some action is to be performed. Under Protel for Windows, 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.

See Also: **[Selection for Autotrax Users](#)**
 [Selection and highlighting](#)
 [Making selections](#)
 [The selection hierarchy](#)
 [Select and De-select commands](#)

Selection for Autotrax Users

In Protel for Windows the functions of the 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 **Ctrl**+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 the reverse of the layer color. 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 or Paste. When you select connections or nets, other previously selected items stay selected until de-selected.

Making selections



Selections are made in three ways:

1. By **direct selection**, using +left mouse to add (or remove) individual items to the current selection;
2. By using the Edit Select and De-select commands to define a selection group;
3. By choosing an action, like the Move command, then selecting a single target item for that action.

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 make a select one item:

1. Use

 +


with the cursor position over an item;



The item will be re-drawn 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  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. Use

 +


over another item;



To release individual items from the selection,:

3. Hold down  and click on any selected item.

When released, the item will be re-drawn in its original (layer) color. Other selected items remain selected until they are either individually released using

 +


or until the Edit De-select All command is executed.

You can use the Zoom commands and hot key shortcuts  and , at any time when making or releasing selection.

The selection hierarchy

When making a direct selection using

Ctrl +

Ctrl,

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

Ctrl +

Ctrl

Select and De-select commands

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 set of options. 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 **Ctrl**+left mouse, described above.

Shortcut: press **Ctrl** to choose from the De-Select command options.

Select and De-select commands include:

- Select/Deselect Inside Area**
- Select/Deselect Outside Area**
- Select/Deselect Free Primitives**
- Select/Deselect All**
- Toggling the selection**

Select/Deselect Inside Area

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

To choose the items inside a selection rectangle.

1. Choose the Edit Select Inside Area command;
You will be prompted "Select First Corner."
2. Move the cursor then press **Ctrl** or left mouse to define the first corner of the selection rectangle;
The prompt changes to "Select Second Corner."
3. Move the cursor to enclose the selection area in the highlighted rectangle;
4. Press **Ctrl** or left mouse to complete the selection.
The newly selected primitives will be highlighted using the selection color. Any previously selected items will remain selected until de-selected.

Select/Deselect 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.

Select/Deselect All

This command selects everything placed in the Protel document window;

Select/Deselect 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.

Select/Deselect 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.

Toggling the selection status

The Toggle Selection command allows you to select or de-select specific primitives, by type. Using this option, rather than the

Ctrl+

Ctrl

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 Toggle Selection command;
2. Choose one of the available options: Arcs **Ctrl**, Components **Ctrl**, Fills **Ctrl**, Pads **Ctrl**, Strings **Ctrl**, Tracks **Ctrl**, or Vias **Ctrl**;
You will be prompted "Select (primitive)."
3. Click on the selected primitive type to add or remove the item from the current selection;
The prompt "Select (item)" will be continuously displayed.
4. Press esc to leave the Toggle Selection command.







Making Changes

The Change command is 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.


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

1. Choose Edit Change;
2. Choose the type of item to be changed: **Arc** , **Component** , **Fill** , **Pad** , **Track** , or **Via** ;

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

3. Position the cursor over the target item and press  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  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.

PCB will not prevent changes that violate specified design rules. A **Design Rule Check(DRC)** should always be performed before final artwork is generated.

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.

Virtually every editable attribute can be globally applied. 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.

All of these options (and more) are supported by the Change commands in Protel for Windows.

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.

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 two changeable attributes - Width and Layer, while a Pads have seven. For a detailed description of all items and the valid range for each attribute please see the section on PCB files in Appendix A.

There are four groups of options in the expanded dialog box:

Attributes

Attributes to Copy

Attributes to Match By

Edit Change Scope

Attributes

These are the changeable parameters of the item. For a Track, the changeable attributes are Width, and Layer. Any or all of the changeable attributes may be changed at each opening of the dialog box.

Attributes to 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.

Attributes to 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 value of the item being changed;
- Different Change all items which do not match the value of the item being changed;
- Any Change whether or not the attribute matches the item being edited.

Matching can be performed using mixed attributes. For example, track matching attributes include width, layer, and selection status. It is possible to make a global changes to currently selected tracks of any width, on the same layer, etc.

This provides feature provides comprehensive control over global changes.

Edit Change Scope

This option applies changes locally or globally.

This Item Only Local editing applies the changed attributes to the selected item (default setting);

Change All Matched Applies changes globally, using the Attributes To Match By and Attributes to Copy settings.

Changing arcs

Arc attributes which can be edited include:

Line Width	Arc line width can be any value between 1 - 255 mils (0.25 - 25.0 mm). 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.
Global Options	Click Options to view the global edit options for arcs.
Attributes To Copy	Attributes which can be copied to other arcs are Line Width and Layer.
Attributes To Match By	Attributes which determine the arcs to be globally edited include Line Width, Layer, and Selection.

Changing components

Component attributes which can be edited include:

Designator	Component designators and comments have a number of <u>editable attributes</u> :
Pattern	<p>The current physical pattern (or footprint) of the component can be changed to any other available pattern in an open component library.</p> <p>Click the Pattern button to scroll the selection bar through the current components.</p> <p>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.</p> <p>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 replacement would be cancelled.</p> <p>The ratsnest display would also be updated to reflect these unrouted nodes. A DIP16 can also be changed to a DIP14, providing there are no connections to pins 15 and 16 on the DIP16.</p> <p>These rules also apply to global pattern changes.</p>
Layer	<p>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.</p> <p>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 re-assigned 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.</p>
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 position is changed by the user. Once you move a component, its status is changed to Fixed In Place. This allows you to lock seed components into place and then re-run Auto Place.
Global Options	<p>Click Options to view the global edit options for components.</p> <p>Remember, component designators and comments have their own global options, described above.</p>

Component global options include:

Attributes To Copy Attributes which can be copied to other components are Pattern, Layer, and Place Status.

Attributes To Match By Attributes which determine the components to be globally edited include Pattern, Comment, Layer, Place Status, Selection, and Wildcard Name.

Wildcards The Wildcard Name field is used to apply changes to components that share designator prefixes (or some other common element). For example, to change all components whose designators start with "U" type U* in the Wildcard Name option. If you use this feature, components with designators U1, U2, etc will be changed.

Changing Designators and Comments

Component designator attributes which can be edited include:

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.

PCB stores the designators used each time you place a component and will automatically supply an 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.

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

See Also: [Component Text Comments](#)
[Component Text Designators](#)

Changing fills

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

Global Options Click Options to view the global edit options for fills.

Options Include:

Attributes To Copy Fill layer assignment can be copied to other fills .

Attributes To Match By Attributes which determine the fills to be globally edited include Layer and Selection.

Changing pads

Free pads or component pads can be individually and globally edited. Attributes which can be edited include:

Designator	<p>Pad designators are used to label pads with pin numbers when creating a new component.</p> <p>Free pads can also include a designator. The default value for free pad designators is empty.</p>
X Size	<p>This field describes the size of the pad on the horizontal (x) axis. Range is 1-1000 mils (.25-25.0 mm).</p>
Y Size	<p>This field describes the size of the pad on the vertical (y) axis. Range is 1-1000 mils (.25-25.0 mm).</p> <p>The X size and Y size can be changed independantly of each other to modify pad shapes.</p>
Hole Size	<p>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.</p>
Layer	<p>Pads can be assigned to any artwork layer. Multi-layer pads are used for through-hole components. SMD pads are typically assigned to either the Top layer or Bottom layer. Click the Layer button to scroll the selectin bar through the available layers to choose a new layer.</p>
Shape	<p>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. Typically, Pad 1 is assigned a unique shape for orientation during insertion onto a board during assembly.</p>
Pwr/Gnd	<p>This attribute is used to manually assign a through-hole (Multi-layer) pad's connection status to any of four internal power planes, if used. Pad's can be connected to a power plane by either a Direct or Relief connection. SMD pads can be Tagged To Plane for automatic connection during automatic or interactive routing.</p>
Global Options	<p>Click Options to view the global edit options for pads.</p>
Options Include:	
Attributes To Copy	<p>Attributes which an be copied to other pads are Designator, X size, Y size, Hole Size, Layer, Shape and Pwr/Gnd assignment.</p>
Attributes To Match By	<p>Global match attributes include Designator, X size, Y size, Hole Size, Component Pattern, Layer, Shape, Pwr/Gnd layer assignment, Selection status and Group.</p> <p>Group refers to pads that are part of a component. This option allows you to restrict changes to component pads, free pads or to</p>

allow changes to be made to any pad.

Changing strings

Free text strings can be changed both individually and globally. Editable text string attributes include:

- Text** The text content of the string. Strings can be up to 32 characters long and any alphanumeric character, including spaces.
- Height** Text size can be set in mils (.001 in) or mm. Range is 36-4000 mils. The stroke width and character width used to display/print the text is automatically proportioned to the height assigned.
- Layer** Text strings can be assigned to any layer.
- Font** Three special vector fonts are available: Default, Sans Serif and Serif. These fonts are designed to support vector plotting.
- Global Options** Click Options to view the global edit options for strings.

Options Include:

Attributes To Copy Copyable attributes include Text (contents of the string), Height, Layer or Font.

Attributes To Match By Global matching attributes for strings include Text (contents of the string), Height, Layer, Font and Selection status.

Changing tracks

Free text tracks and component text (designators and comments) 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 1÷1000 mils. The default width assignment is 12 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 also placed on other layers for non-electrical purposes.

For example, tracks are used on the Overlay layers to indicate the component outlines on the silkscreen artwork. Tracks are also used to indicate voids in layers which are plotted/printed in reverse, such as power planes, solder masks and paste masks.

Global Options Click Options to view the global edit options for tracks.

Options Include:

Attributes To Copy Copyable attributes include track Width and Layer .

Attributes To Match By Global matching attributes for tracks include track Width, Layer assignment and Selection status.

Changing vias

Vias are special purpose pads used to connect tracks between different layers. Vias can be placed through all layers or on any layer pair. Vias are always circular in shape, this is not an attribute which can be changed.

If a shape other than circular is required a free Pad should be used. Vias can be changed both individually and globally.

Editable via attributes include:

Diameter	Vias can be assigned any size in the range of 1-255 mils.
Hole Size	<p>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 1-255 mils.</p> <p>Via hole assignments should be determined in consultation with your board fabrication source. The diameter required</p>
Layer Pair	<p>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).</p> <p>It is possible to change layer pair assignments manually, but the designer should be aware that via layer re-assignment must be approached with great care, if unforeseen effects in the finished board artwork are to be avoided.</p>
Global Options	Click Options to view the global edit options for vias.
Options Include:	
Attributes To Copy	Copyable attributes include Diameter, Hole Size and Layer Pair.
Attributes To Match By	Global matching attributes for vias include Diameter, Hole Size, Layer Pair assignment and Selection status.

File Formats

Protel for Windows will load Autotrax, Tango PCB and Tango Series II and Protel PCB 3.12

See you manual for information.

absolute cursor

Display mode where the displayed cursor coordinates are measured from the absolute workspace origin (0,0 coordinates) or lower-left corner.

active layer

Any Board window layer which has been activated using the Options-Layers & Colors dialog box.

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.

arc

Circular or semi-circular design elements. PROTEL for WINDOWS displays arcs as a series of chords, or straight line segments.

Auto Place

PROTEL for WINDOWS options for automatically loading netlisted components from the current library into a pre-defined board outline.

autoroute

PROTEL for WINDOWS options for automatically (or interactively) routing the connections in a PCB.

Batch mode

Batch mode is an option under PROTEL for WINDOWS which allows you to select a number of plot types (or plot layers) to be generated in a single operation.

Bill of Materials

Or, BOM. A list of the components (including quantities) used in a PCB.

block

A group of components, tracks, etc., which once saved as a selection, can be imported into the current workfile.

Board layer

A special edit layer which holds alignment targets, trim marks, annotation or other mechanical details of your PCB, which you may wish to plot with selected layers.

Board window

The window where you display and edit your PROTEL for WINDOWS PCB file.

Bottom layer

Edit layer for the bottom (or 'solder side') of the PCB.

break

Conversion of a single track segment into two connected segments.

Check plot

A composite plot of multiple artwork layers, used to verify design rules and alignment of the PCB.

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 an PROTEL for WINDOWS 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.

copper

Any non-etched (conductive) portion of any layer of a printed circuit board. Also refers to Top layer of PCB.

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.

default

Program settings or options which remain selected until changed by the user. Most defaults are stored in a file called PROTEL for WINDOWS Prefs (ie. preferences).

de-select

Removing the highlighted (selected) condition, from an item in the Board window or a Tool. De-selection involves clicking elsewhere in the board window or selecting another tool.

Designator

Also called component label. The unique identifier assigned to each component in a circuit.

Design Rule Check

PROTEL for WINDOWS routines for checking a routed PCB file using a netlist to verify connections and user-defined clearance settings to check for clearance violations.

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.

fill

An area fill, or solid rectangular copper on a PCB layer, used for shielding and/or supply of current.

free pad

Any pad that does not belong to a library component. Free pads are identified by the default identifier '0'.

free text

Text placed in the Board window using the Text tool (not component text).

Gerber format

The RS-274 format is the standard file format adopted for coding photoplot files. A Gerber format file describes a photoplot 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.

grid

A system of visible and invisible points on the PROTEL for WINDOWS workspace used to locate a precise coordinate position.

Ground plane

Special PCB mid-layer supplying all ground pins.

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 PROTEL for WINDOWS.

highlight

A unique display color assigned to selected items on-screen to aid in identification or editing. PROTEL for WINDOWS uses the 'reverse' of the current color or grayscale shade to indicate selection.

imperial measure

Inch-based measurement system PROTEL for WINDOWS uses the mil (.001inch) as its default unit. Measurements are stored in imperial format regardless of the display mode.

Keep Out Layer

Special layer used to define a perimeter and 'no go' areas for auto component placement and autorouting.

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 of track routing on PCB layer pairs.

Match file

A text file (.MAT extension) that records the aperture assignments generated when creating a Gerber plot file.

metric measure

Metric-based measurement system using mm (millimeter) as the base unit of measure for PCB design and fabrication. PROTEL for WINDOWS stores all dimensions in imperial format, regardless of the display mode.

Mid layer

Any of four layers (other than the Top and Bottom layers) which can be used for routing the connections on a multi-layer PCB.

mil

Unit of imperial measure equal to .001 inch.

mm

Millimeter - unit of metric measure in PROTEL for WINDOWS.

minimum X, Y

The minimum X or Y coordinate of items in the PROTEL for WINDOWS workspace. This describes the left-most and bottom-most coordinates used in the file or plot.

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.

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.

offsets

Special corrections applied to the origin (0,0 coordinate) of a plot (or print), used to position the artwork within the plot area. Also, the additional X or Y axis spacing used in the Edit-Repeat placement option.

origin

Location of the 0,0 coordinate in either a PCB file or a plot. PROTEL for WINDOWS 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.

Overlay

A special layer of PCB artwork, also called the silkscreen layer. Overlays are used to identify components on the top or bottom of a PCB, and are provided as an aid to manufacturing and maintenance of the PCB.

package

The physical description 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 Board window. PROTEL for WINDOWS provides automatic panning when placing or moving selected items. The Zoom tool also pans the Board window.

Paste Mask

Special plot of SMD pads, used to define a mask for applying solder paste for 'hot re-flow' fabrication.

pre-route

Connection completed by the designer prior to using netlist-based autorouting options. PROTEL for WINDOWS processes these connections independently from autorouted connections.

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.

select

To position the cursor over an item and 'click' with the mouse button. Selected items are displayed in a highlighted color or pattern.

serial

Refers to RS-232C and RS-422 standard for data terminal equipment (DTE) communication. Defies comprehension.

signal layers

Layers available for routing PCB connections in PROTEL for WINDOWS, specifically the Top, Mid 104 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 PROTEL for WINDOWS workspace.

snap to

Special property of track placement in PROTEL for WINDOWS 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 PROTEL for WINDOWS using straight line chord segments. See 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 and vias 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 as well as user prompts.

string

Individual element of free text.

through hole

See Via. Also describes PCB technology where all component pins pass through the PCB.

Title block

Optional line of text which identifies the filename and plot layer.

track

Also called traces, used to carry current or signals on a PCB.

vector font

In PROTEL for WINDOWS, special font 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.

via

Or through hole, a special purpose pad, with a drilled (normally plated) hole, used to connect tracks on different layers. PROTEL for WINDOWS vias are multi-layer (occupy all board layers).

visible grid

An array of dots on the screen which provides a visual reference for positioning items accurately on-screen.

workspace

The complete PROTEL for WINDOWS editing environment including the Board window, Tool palette and Menu bar.

X, Y size

The difference between the minimum and maximum coordinates used on each axis of a PCB or plot, ie. the height and width of the board.

Messages

Auto Place cannot run. No un-locked components
Bad format in Aperture file
Bad format in Pad Type File
Clearance Error
Component Comment Different
Component Designator is already used
Component Designator Not Found
Component Missing From PCB
Component Not Found In Library
Component Pattern Different
Disk Is Write Protected
Draft Code Number Already Used
Drive Not Ready
Drive Write Error
Extra component in net file
Extra net in net file
Extra net on board
Extra Pin On Net
File Corruption Error
File Not Found
File Not Open
Film is too small for plot
I/O Error Number
Invalid Drive
Invalid File Access
Library File Already Exists
Library Name is Already Used
Net Broken Into Multiple Sub-nets
Net Name Not Found
New component cannot be matched with old
No Aperture Space (1000 Limit)
No Apertures Loaded
No Components In Library
No Components Placed
No Holes In PCB
No Horizontal Route Layer Specified
No Libraries Have Been Opened
No Nets Loaded
No Outline On Keepout Layer
No Pad Type Space (200 Limit)
No Pad Types Loaded
No Pins Have Been Selected
No Pins On Power Plane
No Printer Selected
No Selected Primitives For Library Component
No Vertical Route Layer Specified
Nothing Has Been Selected
Nothing to plot
Out Of Memory
Pad Not Found
Pad Type Name Already Used
Pad Type Not Found
Page is too small for plot

Path Not Found
Pin Missing From PCB
Rectangle With No Area Specified
Sector Not Found
Selected Items Will Be Placed Outside PCB Area
String Not Found
This Library is Full
Too Many Files Open
Unrecognised Library File Format
Unrecognised PCB File Format

Auto Place cannot run. No un-locked components

Autoplace can only move components which have been set to " Free to move" using the edit change component command. If there are non free, then the placer will do nothing.

Bad format in Aperture file

The aperture file which you have tried to load or merge from (extension APT) contains text which does not conform to the Protel aperture file format. See the manual for more details.

Bad format in Pad Type File

The pad type file which you have tried to load or merge from (extension PAD) contains text which does not conform to the protel PAD file format. See the manual for more details. These files can also be edited with a standard text editor.

Clearance Error

Only occurs in DRC files.

The two listed items are closer together than is allowed using the current clearance setting. Either move the items or change the clearance setting using Netlist Clearance.

Component Comment Different

Only occurs in DRC files.

The comment of the component with the listed designator, does not match that was found in the DRC netlist.

Component Designator is already used

Components must always have unique names (designators). If you specify a name which is already used then you will get this message. For example, if you are placing a capacitor and type C21 and that designator is used. For this example try typing C? and the program will find you the lowest number with prefix C that you can use.

Component Designator Not Found

You will get this message if you enter a component designator that is not used in the current design. Type `?` to get a list to select from, or use the Information Components command to display a list.

Component Missing From PCB

Only occurs in DRC files.

The comment of the component with the listed designator, does not match that was found in the DRC netlist.

Component Not Found In Library

While placing a component or editing a component pattern, you have typed in a component name in the library (DIP21 for example) that does not exist. Use the Library Components command to browse through the library.

Component Pattern Different

Only occurs in DRC files.

A component which is listed in the netlist file was not found on the board.

Disk Is Write Protected

Remove the write protection tab or get a new disk.

Draft Code Number Already Used

While editing an aperture file, you have tried to create a new photoplotter aperture with the same draft code as one already created. Either delete the original or use a different draft code.

Drive Not Ready

If this is a floppy, the door is probably open. If it occurs with a hard drive, you may have a system hardware/software fault.

Drive Write Error

Most likely cause is running out of disk space.

Extra component in net file

Only occurs in DRC files.

DRC has found a component in the netlist which does not appera on the board.

Extra net in net file

Only occurs in DRC files.

DRC has found a net in the netlist file which does not appear on the board.

Extra net on board

Only occurs in DRC files.

DRC has found a net on the board which does not appear in the netlist file.

Extra Pin On Net

Only occurs in DRC files.

DRC has found a Pin on a net on the board which does not appear on the same net in the netlist file.

File Corruption Error

The PCB design file which you have tried to load from (extension PCB) contains text which does not conform to the Protel PCB file format.

This error indicates that the program recognised the file as a valid PCB file, but then while actually loading it, some part of it was invalid. If you have been editing the file manually (with a text editor) then see the manual for more information or contact Protel Technical support. Or, if the file is a foreign format, Tango for example, it may contain elements that are not recognised by the Protel loader.

Protel AUTOTRAX and EASYTRAX, Tango PCB, Tango Series II and Protel PCB 3.12 files are loaded automatically.

File Not Found

The file name selected could not be found.

File Not Open

The file chosen could not be opened. Check the name and try again.

Film is too small for plot

With the film and border size you have selected, the current board will not fit on the film.

I/O Error Number

Contact Protel Tech support with details of your actions and the I/O error number.

Exit the program, exit windows and try again.

Invalid Drive

You have specified a drive which doesnt exist.

Invalid File Access

Either a network sharing error or some sort of file corruption.

Exit the program, exit windows and try again.

Library File Already Exists

When creating a new library, you must use a name that is not already used.

Library Name is Already Used

Component patterns in the same library must always have unique names. If you specify a name which is already used then you will get this message.

Net Broken Into Multiple Sub-nets

Only occurs in DRC files.

DRC has found a net on the PCB which is not fully routed.

The partial routed section of the net are listed.

Net Name Not Found

You will get this message if you enter a net name that is not used in the current design. Type ? to get a list to select from, or use the Information Nets command to display a list.

New component cannot be matched with old

When editing a component pattern, pins with connection going to the them could not be matched with pins on the new pattern. For example, if you change a DIP16 to DIP14 and there is a connection to pin 16, then the program cannot resolve the connection. If you really want to make the change, you must either delete and replace the component, or clear the netlist and reload it after editing the pattern.

No Aperture Space (1000 Limit)

You can only have 1000 apertures loaded at one time.

No Apertures Loaded

You have tried to photoplot or edit an aperture without having loaded an aperture file. See the manual for more details.

No Components In Library

There are no component patterns stored in the current library. The library was probably just created. Select another library.

No Components Placed

You have selected Information Components, with no components placed on the current design.

No Holes In PCB

If you try to generate an NC drill file when there are no through hole items in the current design, you will get this message.

No Horizontal Route Layer Specified

To run the autorouter, you must select a horizontal layer and a vertical layer, or alternatively, select a single layer for both. See the chapter on automatic routing for more details.

No Libraries Have Been Opened

You have tried to list a library or place a component, when there is no library selected. Select a library file and try again.

No Nets Loaded

You have tried to list a net list or autoroute, when no nets have been loaded. Use the Netlist Nets Get Nets command to load a netlist. See the chapter on netlists for more details.

No Outline On Keepout Layer

Before autoplacing components, you must specify some board outline and keep outs for the program to work within.

No Pad Type Space (200 Limit)

You can only have 200 different pad types loaded at one time.

No Pad Types Loaded

You have tried to edit, list or delete a pad type (or place a pad), when there are none loaded. The default PFW.PAD file should be loaded automatically when the program is started. If not, then make sure this file is the same directory as PFW.EXE, or in directory that appears on the DOS path.

No Pins Have Been Selected

You have selected Information Selected Pins with no pins currently selected.

No Pins On Power Plane

You have selected Information Power Plane 1,2,3,4 with no pins currently connected to the chosen plane. Set the power plane net names with the Netlist Power Plane command, and reload your netlist to make these connections, or alternatively, manually edit the power plane attribute of these pads with Edit Change Pad.

No Printer Selected

You must selected a windows printer in File Plot/Print before choosing Generate Prints.

Use Window Control Panel to install printers.

No Selected Primitives For Library Component

You have used the Library Components Add command with nothing selected on the PCB. Place some items, selected them, and run the command again.

No Vertical Route Layer Specified

To run the autorouter, you must select a horizontal layer and a vertical layer, or alternatively, select a single layer for both. See the manual chapter on automatic routing for more details.

Nothing Has Been Selected

Use the select/deselect commands or SHIFT+Left Mouse to select some items.

Nothing to plot

Not items were found on the board.

Out Of Memory

While placing items, loading a file or general editing, the program has run out of memory. If you have done some deletions then save, close and reload you file to clear the Undo stack. If none of these options work, then you need to add some more memory.

Pad Not Found

In the Jump Pad command, the pad designator that you specified was not found anywhere on the component that you have selected. See the chapter on finding and jumping to items for more details.

Pad Type Name Already Used

While editing an aperture file (See the chapter on aperture files), you have tried to create a new photoplotter aperture with the same draft code as one already created. Either delete the original or use a different draft code.

Pad Type Not Found

You have tried to merge a pad type from a pad type file. The pad type name specified was not found in that file.

Page is too small for plot

With the current printer, scale and border size you have selected, the current board will not fit on the page.

Path Not Found

You have specified a path which doesnt exist.

Pin Missing From PCB

Only occurs in DRC files.

DRC has found a Pin on a net in the netlist file, which does not appear on the board.

Rectangle With No Area Specified

While selecting an area for Zoom Point, Zoom Window or Select Inside, you have specified either the horizontal or vertical dimension of the rectangle to be zero.

Sector Not Found

Hardware read/write error, usually on hard drives.

Selected Items Will Be Placed Outside PCB Area

You cannot move or paste a selection outside the 0,0 32000,32000 workspace.

String Not Found

In the Jump String command, the text string that you specified was not found anywhere on the board. See the chapter on finding and jumping to items for more details.

This Library is Full

Each library can hold 500 component patterns. If you try to store more than this in a library, you will get this message. Create a new library. There is no limit on the number of libraries that you can have open at one time.

Too Many Files Open

Check you files statement in C:\Config.sys

Set to at least 30.

Unrecognised Library File Format

The library file which you have tried to load or merge from (extension LIB) is not saved in a valid Protel format. It may not be the right file (a Protel Schematic file for example) or it may have been damaged. If it is damaged, then you will have to resort to your floppy back-up (which, of course, you always keep safe).

Unrecognised PCB File Format

Protel for windows will load Protel AUTOTRAX and EASYTRAX, Tango PCB, Tango Series II and Protel PCB 3.12 files automatically.

Any other PCB file formats will give this error.

