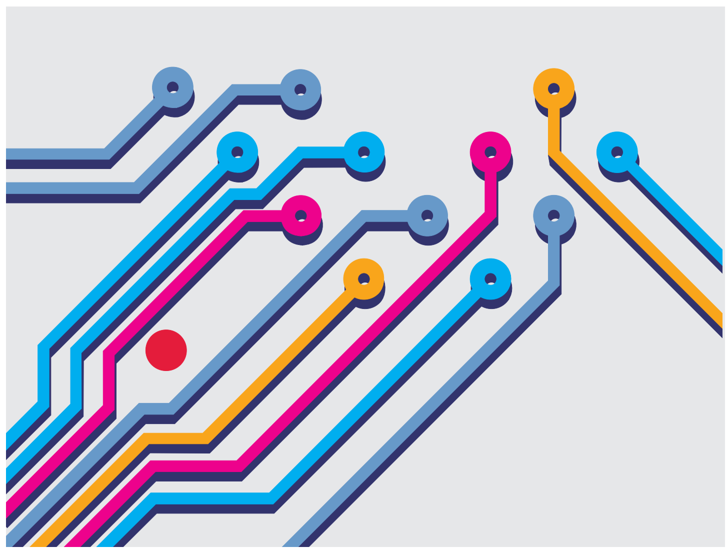
Advanced PCB ™

Printed Circuit Board Design System for Windows



Professional 32-bit PCB design system for Windows

Options: ***Advanced PCB design automation & productivity tools***

Advanced Place, intelligent component auto placement

Advanced Route, 16 layer rip-up / retry autorouting

Protel Technology

On-line Reference

8/93

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

Sales: (800) 544-4186
Tech support: (800) 827-2910
Facsimile: (408) 243-8544

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

Sales: 1800 03 0949
Business: 002 73 0100
Facsimile: 002 73 0944

Product of USA

Contents

File menu

Edit menu

Library menu

Netlist menu

Auto menu

Commands

File menu

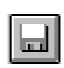
New

Shortcut f, n

Summary This command Displays a new document window with the title "PCB_#". file.

Procedure Choose File-New. A new, empty document window is opened and becomes the current window.

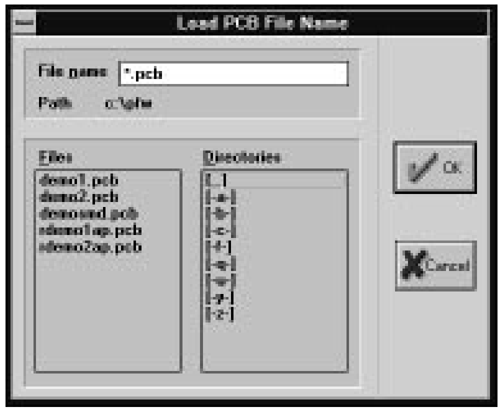
Open

Shortcut f, o or 

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

Procedure Choose File-Open. The file may be of any type supported by Advanced PCB: Advanced PCB binary, Advanced PCB

ASCII, Autotrax ASCII, PADS-PCB (.ASC), PADS-2000 (.ASC), PCAD (PDIF 5/6) or Tango Series II.



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 SPACEBAR. This allows you to move directly to another menu command or Tool button without waiting for the entire screen redraw to be completed.

Close

Shortcut f, c

Summary Closes the current document window.

Procedure First, 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

Restore Backup

Shortcut f, k

Summary Overwrites the current board window with the last created autobackup version of the file (if any). Auto backups are labeled: (filename).AB0, .AB1, etc., up to .AB9, then back through .AB0, .AB1, etc. again. Restore Backup uses the creation date for the autobackup file, not the numeric extension, to determine the restored version.

Procedure First, choose a document window, then choose File-Restore Backup.

See also File-Exit, File-Save, File-Save As

Warning You cannot Undo this command. This command overwrites the current PCB document window contents with the restored backup version. Make sure that you save the current workfile with a new name as a safeguard before using the Restore Backup command.

Save

Shortcut f, s or 

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

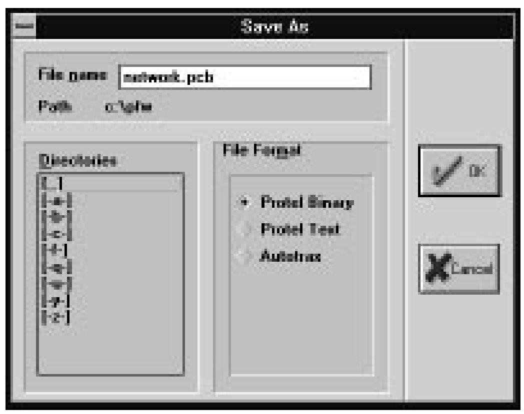
Procedure First, choose a document window, then choose File-Save.

See also Save As, Save All

Save As

Shortcut f, a

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



Procedure First, choose a document window, then choose File-Save As. Three file formats are available:

a. Binary: the default Advanced PCB file format.

b. Text: user-editable ASCII version of the Advanced PCB file format.

c. Autotrax: ASCII format for Protel Autotrax PCB layout system.

Warning Autotrax does not support all Advanced PCB data and some Advanced PCB information may be lost if you save your file in this format. Please consult your Autotrax documentation prior to generating Autotrax files.

See also File-Save, File-Save All

Save All

Shortcut f, l

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, File-Save As

Shape Based Router-Export Design

Summary Exports the current PCB workfile to the Advanced SB Route (or Specctra SP-10) application for batch processing.

Procedure Choose the File-Shape Based Router-Export Design command.

Shape Based Router-Import Routes

Summary Imports an autorouted PCB workfile from the Advanced SB Route (or Specctra SP-10) application.

Procedure Choose the File-Shape Based Router-Import Routes command.

Shape Based Router-Run

Summary Launches the Advanced SB Router (or Specctra SP-10) application.

Procedure Choose the File-Shape Based Router-Run command. A Files dialog box opens. Choose the appropriate directory and the router .EXE file. Double-click or press OK to start the router application.

Gerber-Gerber In

Shortcut f, g, i

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. If the plots are generated with the Center Plots on Film option disabled (default is enabled), plots will be directly superimposed over the (previously opened) PCB file.

Procedure Open a new document window or choose an existing document window for the Gerber file. Open a matching aperture table for the Gerber file using the Library-Apertures command. This aperture table must include matches for each primitive to be "stroked,", "flashed" or "painted" in the Gerber file. If no aperture file is loaded, primitives will be displayed by simulating a 1 mil round aperture.

1. Choose the Gerber-Gerber In command.

A Load File Name dialog box opens, listing files with the extension G.**. Gerber files are typically identified by this format, e.g. (filename).GTL for a "Top layer" gerber.

2. Place the selection bar over the desired file name and double-click, or click OK.

The Gerber data will be interpreted, then displayed on the current layer. As the Gerber file is loaded, all the primitives will be placed on the current layer on the Advanced PCB workspace. Strokes will be converted to tracks, and flashes will be converted to pads. There are some exceptions to this rule:

If a horizontal or vertical stroke was generated using a square or rectangular aperture then the PCB file will contain an area fill of equivalent size.

If a rectangular aperture has been used to draw a stroke that is neither horizontal nor vertical then it will be converted to a standard track. This can produce some errors when loading Gerber files. The only place where non-vertical/horizontal strokes are used is when filling octagonal pad.

Note The Gerber loader recognizes Gerber 2:3, 2:4 and 2:5 numeric formats. You have the option to specify the numeric format when generating Gerber files.

The "File-Gerber-Gerber In" and "File-Gerber-Batch Load" commands do not incorporate file integrity checking (performed automatically whenever loading PCB files in all formats). This enables Gerber files to be viewed without modification. Saving non-Protel Gerber files as PCB files may cause problems unless a few precautions are followed.

If the workspace appears empty after opening a Gerber file, the data may exist beyond the legal boundaries (the Advanced PCB environment allows data within X:0–99999.999 and Y:0–99999.999 coordinates). This situation can be confirmed by viewing the "PCB Information" dialog box (Choose "Info/Board Status..."). "Board Dimensions" beyond the 0 to 99999.999 range and quantities greater than zero for "Physical Items on PCB" are indications that data exists beyond the workspace boundaries.

Data can be moved within the borders of the workspace by using the following method:

1. Choose Edit-Select-All.

2. Choose Edit-Move-Move Selection.
3. Designate a Reference Point near the upper right corner of the workspace.
4. Place the selection somewhere near the lower right corner of the workspace.
5. Choose the Zoom-All command to view the file.

If this procedure was successful the screen will redraw with the data in view. If not, try moving the selection in another direction.

6. Save the document using the text format then open this file.

Integrity checking will take place as the text format file is opened. Integrity checking ensures that invalid data can not cause application or system errors.

Warning When regenerating edited Gerber files, Solder Mask expansions, Paste Mask expansions and Thermal Relief expansions are reapplied to the current data. Be sure that you reset these values to zero before regenerating the Gerbers.

See also Library-Apertures

Gerber-Batch Load

Shortcut f, g, b

Summary Loads all Gerber files in the current directory with the format (filename).G** into the current document window.

Procedure First, choose a document window. Open a matching aperture table for the file, using the Library-Apertures command. This aperture table must include matches for each primitive to be “stroked,” “flashed” or “painted” in the Gerber file.

1. Choose the File-Gerber-Batch Load command.

A Load File Name dialog box opens, listing files with the extension G.**. Gerber files are typically identified by this format, e.g. (filename).GTL for a “Top layer” gerber.

2. Place the selection bar over any layer file and double-click, or click OK.

The Gerber data will be interpreted, then assigned to the available design layers, according to the extension for each layer file, e.g. .GTL will be assigned to the Top (component side) layer, .GBL to the bottom (solder side) layer, etc. If no aperture file is loaded, primitives will be displayed by simulating a 1 mil round aperture. Layers are assigned to each plot file, as follows:

Top Overlay	.GTO	Bottom Layer	.GBL
Top layer	.GTL	Power Plane 1, etc.	.GP1
Mid layer 1, etc.	.GM1	Top Solder Mask	.GTS
Mech. layer 1, etc.	.GF1	Bottom Solder Mask	.GBS
Drill Drawing	.GDD	Pad Master	.GPM
Drill Guide	.GDG	Keep Out layer	.GKO

As the Gerber file is loaded, all the primitives will be placed on the current layer on the Advanced PCB work-space. Strokes will be converted to tracks, and flashes will be converted to pads. There are some exceptions to this rule:

If a horizontal or vertical stroke was generated using a square or rectangular aperture then the PCB file will contain an area fill of equivalent size.

If a rectangular aperture has been used to draw a stroke that is neither horizontal nor vertical then it will be converted to a standard track. This can produce some errors when loading Gerber files. The only place where non-vertical/horizontal strokes are used is when filling octagonal pads.

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). If the plots are generated with the Center Plots on Film option disabled (default is enabled), plots will be directly superimposed over the (previously opened) PCB file.

Note The Gerber loader recognizes Gerber 2:3, 2:4 and 2:5 numeric formats. You have the option to specify the numeric format when generating Gerber files.

Warning See the notes regarding file integrity in the preceding section on the File-Gerber In command.

See also File-Gerber-Gerber In, Library-Apertures

Gerber-Gerber Output

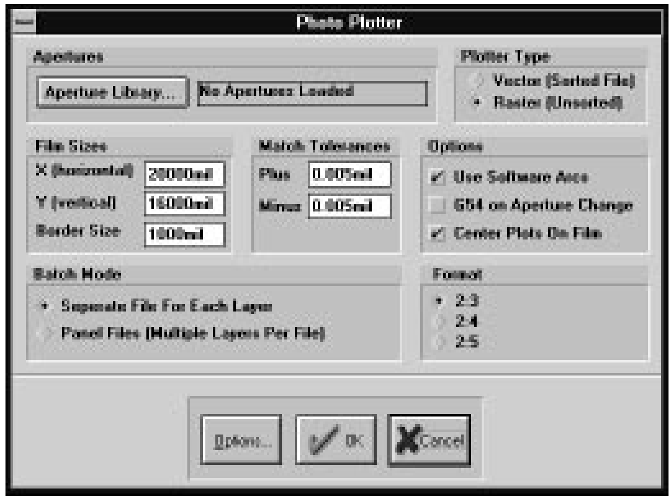
Shortcut f, g, g

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. Output in the following Gerber coordinate formats is supported: 2:3, 2:4 and 2:5.

Procedure First, choose a document window. Load the appropriate aperture file for the target plotter or generate apertures automatically for the current PCB.

1. Choose the File-Gerber-Gerber Out command.

The Photoplotter dialog box opens, presenting the following options:



Apertures	Click Aperture Library to open the Apertures dialog box. Choose from any pre-defined aperture files, identified by the extension .APT, edit the current aperture list or generate a new aperture file from the current PCB.
Plotter Type	This option defines the sorting of the Gerber file.
Film Sizes	Specifies the film format and minimum border for the target plotter. The default is 20 x 16 inches. Your bureau will tell you the available sizes, but be careful to specify the x and y values in the proper orientation.
Border Size	Specifies the border size for each film. This will define the border around each plot, so there will be a space at least equal to twice the border size between panelized plots.

The system will spread the plots out to equalize the borders around all the plots if there is more space available.

Match Tolerances

Specifies the tolerances applied to aperture assignment. This system allows use of apertures which are not an exact match with the primitive, but are within a specified range. For example, a tolerance of -2 mils allows an aperture 2 mils smaller than the primitive to be used to “flash” the primitive. A match tolerance of +2 mils would allow an aperture 2 mils larger than the target primitive, etc. Matches that fall outside the specified tolerance are achieved by either “stroking” or “painting” the desired shape where the available apertures permit. However, some photoplotters restrict the use of individual apertures to either “flash” or “stroke.” Other plotters allow less restrictive use.

➡ Tolerances should be applied with care, due to the possible compromise of design integrety when accepting inexact matches. It is strongly recommended that you consult with your plotting bureau for specific information regarding plotting tolerances and restrictions of apertures before generating Gerber files. It is also recommended that you make a careful inspection of your Gerber files prior to submission to your plotting bureau.

Options

Use Software Arcs

Some photoplotters do not support the Gerber arc drawing command, where the arc is generated by the plotter (“hardware arcs”). The system can generate a series of short line segments to draw the arcs – referred to as “software arcs.” Hardware arcs are preferable, if supported by the target plotter. Consult with your photoplot bureau. Hardware arcs are used for arcs that are in multiples of 90 degrees only – as the Gerber language can only describe arcs of one-to-four quadrants. For any small arc angle (less than 90 degrees), software arcs are automatically generated, regardless of whether or not Use Software Arcs is selected.

G54 on Aperture Change

Some early model photoplotters require a specific G54 command to be inserted in the control code before every aperture change command. Consult with your photoplot bureau to determine this requirement.

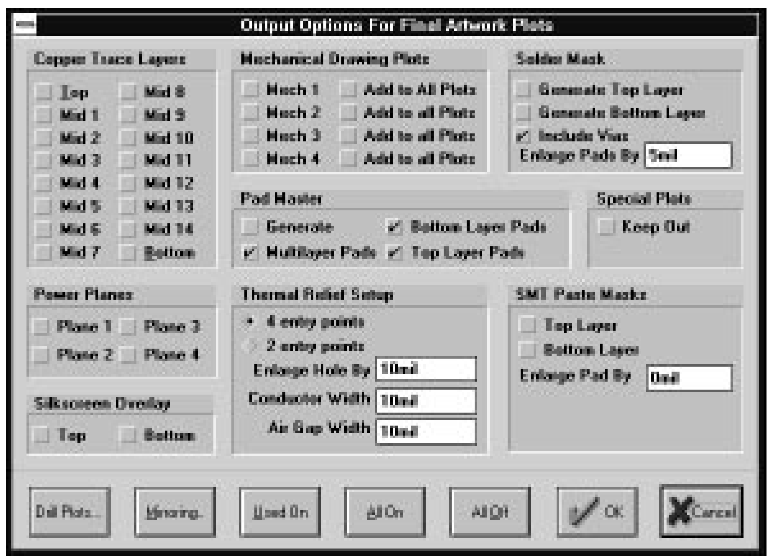
Center Plots on Film: Places the file image at the center of each film sheet.

Batch Mode

Options include Separate File for Each Layer (one plot per film) or Panel Files (panelizes multiple layers on a single sheet, using the border size as a minimum layer-to-layer clearance. If there is insufficient film space for your layers (per your Border setting above) the system will generate as many files as required for the plots you have specified. The first file will be called <filename>.P01, the next one <filename>.P02, etc.

Format — Specifies the numeric format for the Gerber file. 2:3 is the default.

The Options button opens the Output Options for Final Artwork Plots dialog box.



Options for this dialog box include:

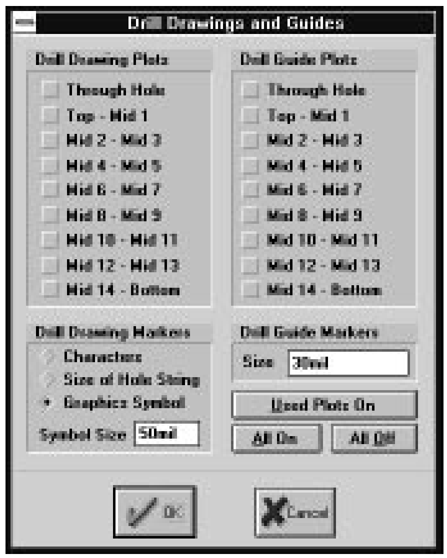
Copper Trace Layers — Specifies signal layers to be included in the plot file(s).

Power Planes — Specifies power plane layers to be included in the plot file(s).

Mechanical Drawing	Specifies mechanical layers to be plotted alone or to be assigned to all plots in the current batch.
Solder Mask	Specifies solder mask plot(s) with the option to include vias. User can also specify expansion of pads (and vias, if enabled). Expansion is in all directions by the specifed value. For example, round pads and vias will be expanded radially by the entered value.
Pad Master	Specifies special Pad Master plot, with options to include multilayer, top layer and/or bottom layer pads in the plot.
Keep Out	Includes the Keep Out layer in the batch plot.
Thermal Relief Setup	Specifies thermal reliefs and the number of entry points and enlargement (annular, not diameter) on internal Power Plane layer plots. The width of the entry point conductors and air gap can also be specified. Note: method of connection of power pins to internal planes is pre-assigned, on a net-by-net basis, in the PCB editor. See Netlist-Power Planes for more information.
SMT Paste Masks	Specifies paste masks to be generated as plot layers. Includes a pad enlargement (pad will be enlarged in all directions by the value specified).

Used On	Selects all layers in the current PCB file for plotting in the batch.
All On	Selects all available system layers for plotting in the batch.
All Off	Deselects all system layers.

The Drill Plots option opens the Drill Drawings and Guides dialog box.



Drill Plots options include:

Drill Drawing Plots	Specifies layers to be included in the Drill Drawing plot file.
Drill Guide	Specifies layers to be included in the Drill Guide plot file.
Drill Drawing Markers	Selects the type and size of hole location markers to be plotted.
Drill Guide Markers	Specifies the size of the hole drilling targets in the Drill Guide plot.

Used Plots On	Selects all layers in the current PCB file for Drill Guide/Drill Drawing plots.
All On	Selects all available systsme layers for inclusion in Drill Guide/Drill Drawing plots.
All Off	Deselects all system layers.

The Mirroring option opens the Flip Layer on Artwork dialog box. Any layers selected in this dialog box will be plotted in a flipped orientation.

2. When all options have been specified, click OK to generate the Gerber plot file(s).

Extensions are automatically applied to Gerber output files, using a unique identifier for each layer (see list under File-Gerber-Batch In). If you are generating panelized files, the system will generate as many files as required to fit all the plot layers specified. The files will be automatically labeled (filename).P01, .PO2 and so on. If you have specified separate films for each file, then specific file name extensions will be generated such as .GTL for Gerber top layer, etc.

Note An option in the Photoplotter setup dialog box allows the user to Center Plots on Film. Users should be aware that choosing this option does not change the N/C drill coordinates for the file. If N/C drill file and plot file coordinates must match, do not choose this option. If you do not choose this option, The File-Gerber-In command will superimpose the Gerber layer(s) directly over the (open) PCB file.

See also Options-Layers, Options-Display, Library-Apertures

Print-Composite

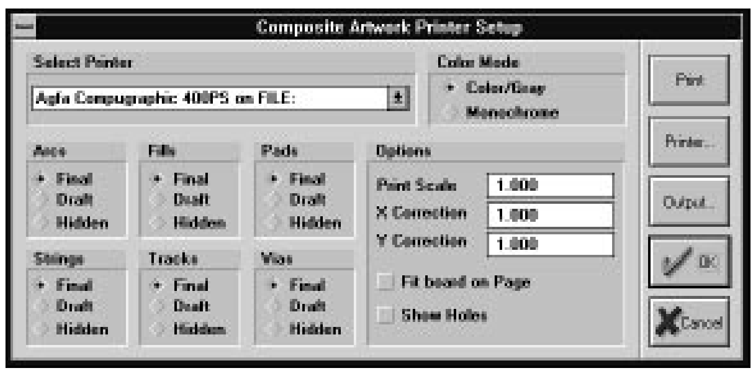
Shortcut f, p, c

Summary Generates Printer (or plotter, when using Windows raster plotter drivers) output. Multiple layer composite (or "check") prints with final or draft (outline) primitive rendering is supported.

Procedure Choose a document window to print (plot).

1. Choose the File-Print-Composite command.

The Composite Artwork Printer Setup dialog box opens.



Options include:

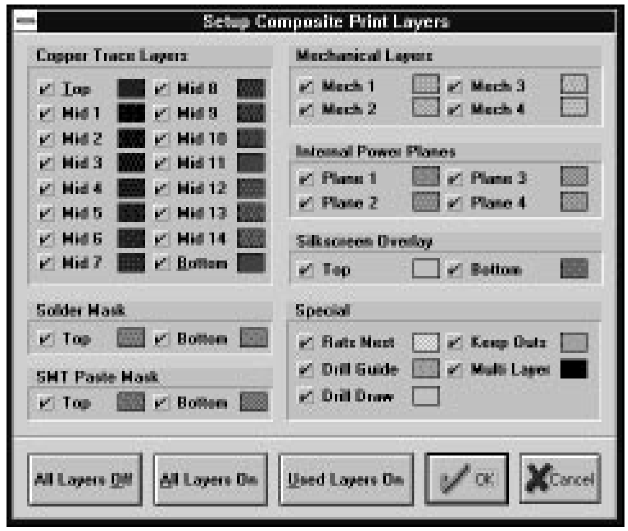
Select Printer	Choose from the available devices. All installed Windows printer drivers are supported. To install additional device drivers, see you *Microsoft Windows Users Guide* for details.
Color Mode	Selects either Color/Gray rendering (for devices that support this option) or Monochrome (black only) printing.

Primitives	Assigns a print mode for each primitive type: Arcs, Fills, Pads, Strings, Tracks, Vias. Final mode prints the object as a solid. Draft prints an outline of each primitive. Hidden object types will not print.
Print Scale	Assigns a scaling factor to the print. Range is from .250 to 4.000 times, assuming sufficient printer memory and capacity to support the selection. Default is 1.0.
X, Y Correction	Provides a correction factor for either axis. Allows the user to fine tune the scaling of the print to enhance accuracy. Default is 1.000.
Fit Board on Page	Automatically scales the board image to fit onto the page size specified for the selected device.
Show Holes	Prints pads and vias with holes. When this option is disabled, pad and vias print without holes.

Pressing the Print button sends the print job directly to the selected device. If printing to File, you will be prompted to supply a filename. As the print is generated, a dialog box reports progress.

Pressing the Printer button opens the Windows dialog box for the selected device. Depending upon the device, options will include paper source, paper size, page orientation, number of copies, etc. For accurate scaling, use the Composite Artwork Setup dialog box, not the device Options dialog box to adjust print scale.

Pressing the Output button opens the Setup Composite Print Layers dialog box.



You can include all of these layers in a composite print. Click to choose the layers, then click OK.

After all settings are complete click the Print button to print the current document window or click OK to close the dialog box and save the settings without printing at this time.

See also File-Plot command

Print-Final Artwork

Shortcut f, p or 

Summary Generate Printer or (plotter, when using Windows raster plotter drivers) output. Single layer prints with final quality and draft (outline) primitive rendering is supported.

Procedure Choose a document window to print (plot).

1. Choose the File-Print-Final Artwork command.

The Final Artwork Printer Setup dialog box opens.

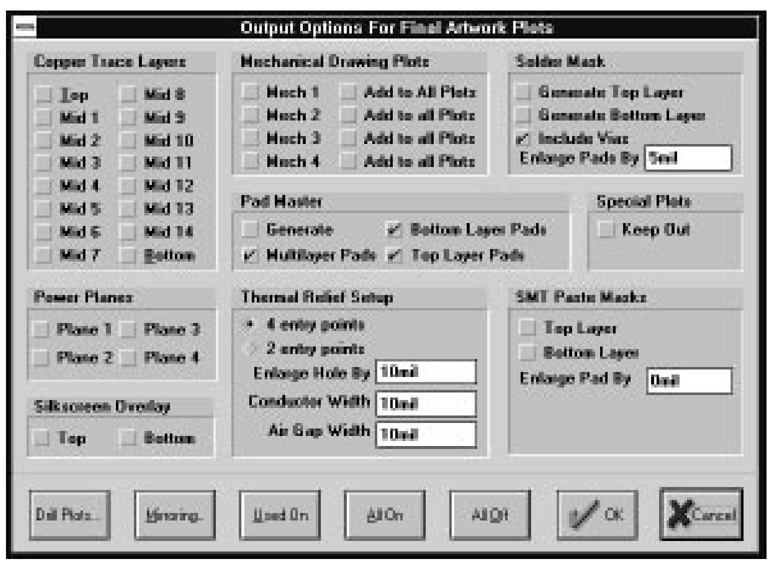
Choose from the following options and settings:

Select Printer	Choose from the available devices. All installed Windows printer drivers are supported. To install additional device drivers, see you *Microsoft Windows Users Guide* for details.
Batch Mode	Specifies either Separate Page for each Layer (one sheet per Layer) or Panels (fits as many layers on each sheet as possible, per the Border Between Prints setting).
Border Between Prints	Specifies the minimum border around each image. Applies to both Separate Page for Each Layer prints and Panels of multiple layers per page.

Show Holes	Prints pads and vias with holes. When this option is disabled, pad and vias print without holes.
Fit Layer on Page	Automatically scales the layer image to fit onto the page size specified for the selected device.
Print Scale	Assigns a scaling factor to the print. Range is from .100 to 10.000, assuming sufficient printer memory and capacity. Default is 1.000.
X, Y Correction	Provides a correction factor for either axis. Allows the user to fine tune the scaling of the print to enhance accuracy. Default is 1.000.

Click the Print button to send the print job directly to the selected device.

The Options button opens the Output Options for Final Artwork Plots dialog box.



Options for this dialog box include:

Copper Trace Layers	Specifies signal layers to be included in the print file(s).
Power Planes	Specifies power plane layers to be included in the print file(s).
Mechanical Drawing	Specifies mechanical layers to be printed alone or to be assigned to all prints in the current batch.
Solder Mask	Specifies solder mask print(s) with the option to include vias. User can also specify expansion of pads (and vias, if enabled). Expansion is in all directions by the specifed value. For example, round pads and vias will be expanded radially by the entered value.
Pad Master	Specifies special Pad Master print, with options to include multilayer, top layer and/or bottom layer pads in the print.
Keep Out	Includes the Keep Out layer in the batch print.
Thermal Relief Setup	Specifies thermal reliefs and the number of entry points and enlargement (annular, not diameter) on internal Power Plane layer print. The width of the entry point conductors and air gap can also be specified. Note: method of connection of power pins to internal planes is pre-assigned, on a net-by-net basis, in the PCB editor. See Netlist-Power Planes for more information.

SMT Paste Masks	Specifies paste masks to be generated as plot layers. Includes a pad enlargement (pad will be enlarged in all directions by the value specified).
Used On	Selects all layers in the current PCB file for plotting in the batch.
All On	Selects all available system layers for plotting in the batch.
All Off	Deselects all system layers.

The Drill Plots option opens the Drill Drawings and Guides dialog box.

Options include:

Drill Drawing Plots	Specifies layers to be included in the Drill Drawing plot file.
Drill Guide	Specifies layers to be included in the Drill Guide plot file.

Drill Drawing Markers	Selects the type and size of hole location markers to be plotted.
Drill Guide Markers	Specifies the size of the hole drilling targets in the Drill Guide plot.
Used Plots On	Selects all layers in the current PCB file for Drill Guide/Drill Drawing plots.
All On	Selects all available systsme layers for inclusion in Drill Guide/Drill Drawing plots.
All Off	Deselects all system layers.

The Mirroring option opens the Flip Layer on Artwork dialog box. Any layers selected in this dialog box will be plotted in a flipped orientation.

2. When all options have been specified, click the Print button to send the file directly to the specified device.

After these settings are complete click the Print button.

See also File-Plot command

Pen Plot-Composite

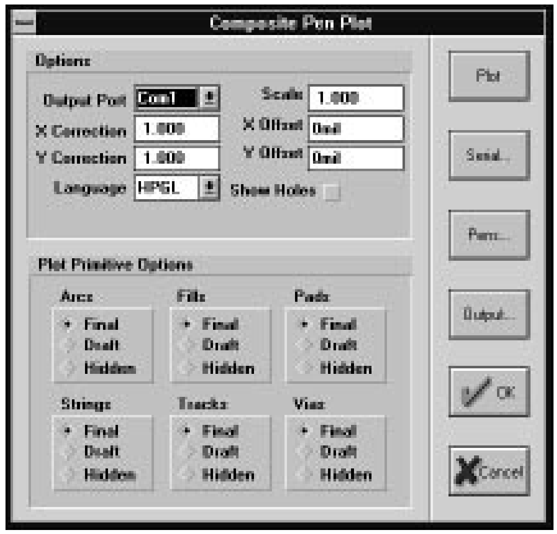
Shortcut f, t

Summary Generate vector pen plotter output for HP-GL and DM-PL compatible devices. This command bypasses the standard Windows device drivers and uses a special plot routine optimized for precision pen plotting. Multiple layer composite, final quality and draft (outline) modes are supported. Pens can be assigned to each layer in composite (e.g. color) check plots.

Procedure Choose a document window to plot.

1. Choose the File-Pen Plot-Composite command.

The Composite Pen Plot dialog box opens.



The following options and settings are displayed:

Output Port	Choose from the available devices installed on your system. See you *Microsoft Windows Users Guide* for details.
Scale	Assigns a scaling factor to the print. Range is from .100 to 10.000 times, assuming sufficient plotter capacity to support the selection. Default is 1.000.
X, Y Correction	Provides a correction factor for either axis. Allows the user to fine tune the scaling of the plot to enhance accuracy. Default is 1.000.
X, Y Offset	Provides positioning control for the plot image on the sheet area. The

default 0 mil setting positions the plot at the plotter origin.

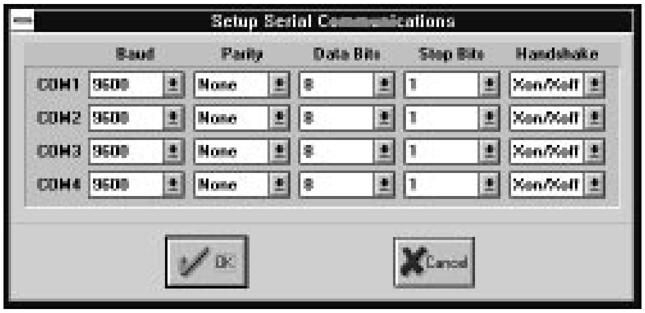
Language — Selects either HP-GL (Hewlett Packard and compatible plotters) or DM-PL (Houston Instruments or compatible plotters).

Show Holes — Plots pads and vias with holes. When this option is disabled, pad and vias print without holes.

Primitives — Assigns a plot mode for each primitive type: Arcs, Fills, Pads, Strings, Tracks, Vias. Final mode plots the object as a solid. Draft mode plots an outline of each primitive. Hidden object types will not plot.

Pressing the Plot button sends the current plot job directly to the specified plotter. As the plot is processed a dialog box reports completion progress.

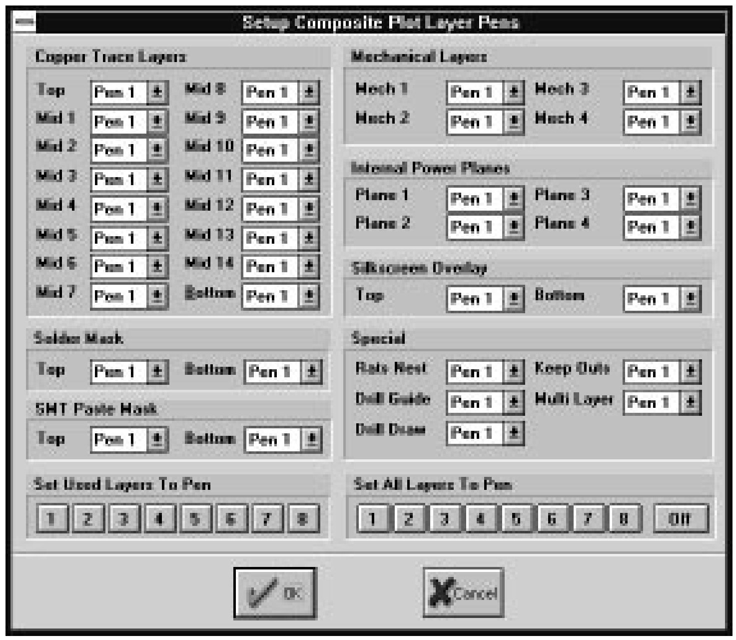
Pressing the Serial button opens the Setup Serial Communications dialog box.



These options specify communication for the available devices. See your system and plotter documentation for details.

Pressing the Pens button opens the Plotter Pens dialog box for the selected device. Options include the speed and nib thickness (in mils or mm) for each of up to 8 pens.

Pressing the Output button opens the Setup Composite Print Layers dialog box.



Choose from the available options to assign pens (and therefore colors) to various layers and other special items.

After these settings are complete click the Plot button.

See also File-Print command

Pen Plot-Final Artwork

Shortcut f, t

Summary Generate vector pen plotter output for HP-GL and DM-PL compatible devices. This command bypasses the standard

Windows device drivers and uses a special plot routine optimized for precision.

Procedure Choose a document window to plot.

1. Choose the File-Pen Plot-Final Artwork command.

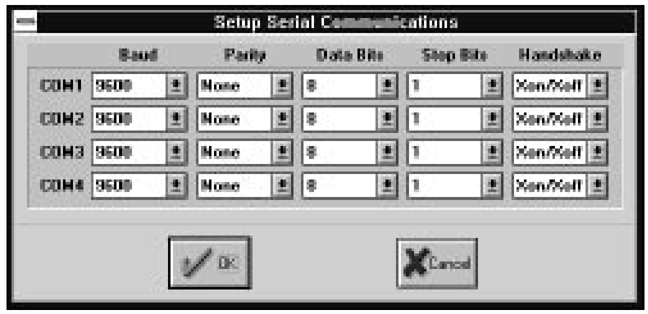
The Final Artwork Pen Plot dialog box opens.

The following options and settings are available:

Output Port	Choose from the available devices installed on your system. See you *Microsoft Windows Users Guide* for details.
Scale	Assigns a scaling factor to the print. Range is from .100 to 10.000 times, assuming sufficient plotter capacity to support the selection. Default is 1.000.
X, Y Correction	Provides a correction factor for either axis. Allows the user to fine tune the scaling of the plot to enhance accuracy. Default is 1.000.
X, Y Offset	Provides positioning control for the plot image on the sheet area. The default 0 mil setting positions the plot at the plotter origin.
Language	Selects either HP-GL (Hewlett Packard compatible plotters) or DM-PL (Houston Instruments compatible plotter).
Show Holes	Plots pads and vias with holes. When this option is disabled, pad and vias print without holes.

Wait Between Sheets When Plotting (if enabled) pauses the plot program as each sheet is completed. This allows you to re-load paper, re-fill pens, etc.

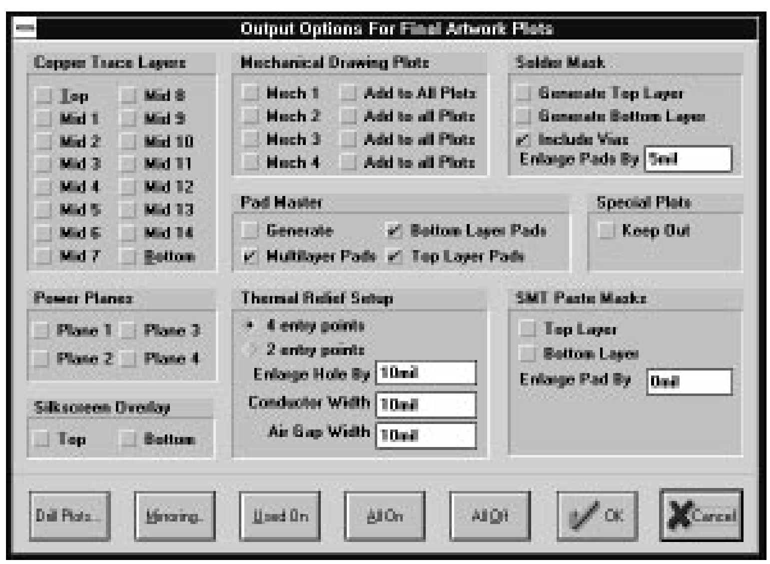
Pressing the Serial button opens the Setup Serial Communications dialog box.



These options setup plotter communication. See your system and plotter documentation for details.

Pressing the Pens button opens the Plotter Pens dialog box for the selected device. Options include the speed and nib thickness (in mils or mm) for each of up to 8 pens.

Click on the Output button to open the Output Options for Final Artwork Plots dialog box.

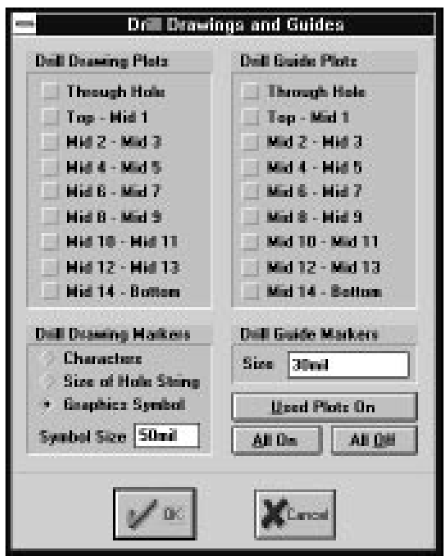


Options for this dialog box include:

Copper Trace Layers	Specifies signal layers to be included in the plot file(s).
Power Planes	Specifies power plane layers to be included in the plot file(s).
Mechanical Drawing	Specifies mechanical layers to be plotted alone or to be assigned to all plots in the current batch.
Solder Mask	Specifies solder mask plot(s) with the option to include vias. User can also specify expansion of pads (and vias, if enabled). Expansion is in all directions by the specifed value. For example, round pads and vias will be expanded radially by the entered value.
Pad Master	Specifies special Pad Master plot, with options to include multilayer, top layer and/or bottom layer pads in the plot.
Keep Out	Includes the Keep Out layer in the batch plot.
Thermal Relief Setup	Specifies thermal reliefs and the number of entry points and enlargement (annular, not diameter) on internal Power Plane layer plots. The width of the entry point conductors and air gap can also be specified. Note: method of connection of power pins to internal planes is pre-assigned, on a net-by-net basis, in the PCB editor. See Netlist-Power Planes for more information.

SMT Paste Masks	Specifies paste masks to be generated as plot layers. Includes a pad enlargement (pad will be enlarged in all directions by the value specified).
Used On	Selects all layers in the current PCB file for plotting in the batch.
All On	Selects all available system layers for plotting in the batch.
All Off	De-selects all system layers.

Drill Plots opens the Drill Drawings and Guides dialog box.



Options include:

Drill Drawing Plots	Specifies layers to be included in the Drill Drawing plot file.
Drill Guide	Specifies layers to be included in the Drill Guide plot file.

Drill Drawing Markers	Selects the type and size of hole location markers to be plotted.
Drill Guide Markers	Specifies the size of the hole drilling targets in the Drill Guide plot.
Used Plots On	Selects all layers in the current PCB file for Drill Guide/Drill Drawing plots.
All On	Selects all available systsme layers for inclusion in Drill Guide/Drill Drawing plots.
All Off	Deselects all system layers.

The Mirroring option opens the Flip Layer on Artwork dialog box. Any layers selected in this dialog box will be plotted in a flipped orientation.

2. When all options have been specified, click OK to generate the plot file(s).

See also File-Print command

NC Drill

Shortcut f, d

Summary Generates binary and ASCII NC Drill files.

Procedure First, choose a document window, then choose File-NC Drill.

DXF

Shortcut f, f

Summary Generates a DXF format file.

Procedure Load a .PCB file and choose File-DXF. A composite of all board layers is generated as a DXF file. Tracks will be rendered as individual polyline segments.

Export Selection

Shortcut f, e

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

Procedure First, choose a document window and use Edit-Select and Edit-De-Select to select the primitives and components to include in the PCB file. Choose File-Export Selection.

Any selection can be saved as a Advanced PCB binary file which can be opened in a new document window at any time. This provides a convenient way to store modular design elements for future use. 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

Reports-Bill Of Material

Shortcut f, r, b

Summary Generates an ASCII parts list file.

Procedure Choose File-Reports-Bill of Materials. Two options are available: standard Protel format or CSV (comma separated values) format. CSV format is compatible with most database and spreadsheet applications. Both formats are output in ASCII text.

Reports-Board Specifications

Shortcut f, r, s

Summary Generates an ASCII report file that lists board primitives, components, hole counts, copper areas and layers used.

Procedure Choose File-Reports-Board Specifications and choose either the Protel or CSV format.

Reports-Netlist Status

Shortcut f, r, n

Summary Generates an ASCII file that lists routed/unrouted nets.

Procedure Choose File-Reports-Netlist-Status and choose either the Protel or CSV format.

Reports-Pick And Place

Shortcut f, r, p

Summary Generates an ASCII report file (.PIK) that lists components by designator, pattern, center coordinates, status (layer, etc.) and rotation. Format is described on page 224.

Procedure First load all libraries used to create the board. Then choose File-Reports-Pick And Place. Finally, choose either the Protel or CSV (comma separated values) format.

Re-Annotate

Summary Re-labels board component designators and generates a .WAS file, used to back-annotate component designators in Advanced Schematic.

Procedure First, choose a document window and then the File-Re-Annotate command. Three options are provided: X then Y, Y then X and Name from Position. Each option re-labels component designators using a positional scheme based on the component reference point coordinates.

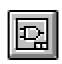
ReAnnotation order includes five options:

...Ascending...	Annotates from bottom-to-top.
...Descending...	Annotates from top-to-bottom.

...X Then... Y	Annotates horizontal rows, then vertical rows.
...Y Then... X	Annotates vertical rows, then horizontal rows.
Name from Position	Annotates with abbreviated coordinates, derived from the position of the component reference (usually the center of pad #1. For example, a coordinate of X:1500.000, Y:3900.000 results in a new designator "U015_039".

When you choose this command an ASCII text ".WAS" file is generated, listing initial and re-annotated designator values. Advanced Schematic reads the .WAS file (Back Annotate command) and updates all sheet part designators.

Run Schematic Capture

Shortcut f, h or 

Summary Launches the Advanced Schematic application.

Procedure Choose File-Run Schematic Capture command. If the Advanced Schematic (SCHEDIT.EXE) file is in the current DOS PATH statement, the application will be launched.

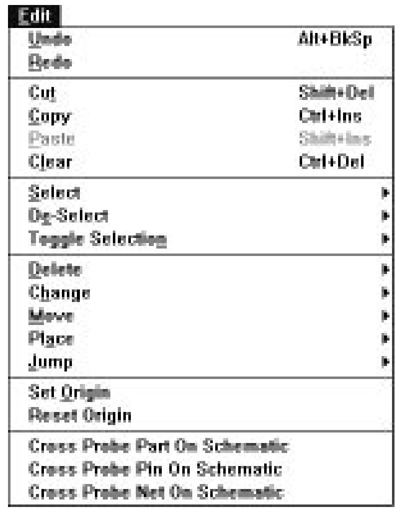
Exit

Shortcut f, x

Summary Quits the current session of Advanced PCB.

Procedure Choose File-Exit. If any document window has been modified since saving, the Confirm dialog box will be displayed, prompting the user to save the changes and exit (Yes), discard the changes and exit (No) or Cancel the Exit command.

Edit menu



Undo

Shortcut ALT+BACKSPACE or

Summary Restores the contents of the current document window to its state prior to the previous command. Multiple level Undo is supported, back to the previous File-Open or File-Save As command.

Note The Undo-Redo stack can consume a large proportion of the available memory, degrading system performance. Choose the File-Save AS command to clear this stack and performance will return to normal.

Procedure Choose Edit-Undo

See also Edit-Redo

Redo

Shortcut ALT, r or

Summary Restores the previous changes made by the Edit-Undo command. Multi-level Undo-Redo is supported back to the previous File-Open or File-Save As command.

Procedure Choose Edit-Redo.

See also Edit-Undo

Cut

Shortcut SHIFT+DEL or

Summary Removes the selected items form the current document window and places them into the Advanced PCB clipboard.

Procedure Select the objects that will be removed. Use Edit-De-Select (or SHIFT+LEFT MOUSE) to de-select objects that are not to 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 ENTER.

See also Edit-Paste, Edit-Select, Edit-De-Select, Edit-Toggle Selection

Copy

Shortcut CTRL+INS

Summary Copies all selected items from the current document window to the Advanced PCB clipboard.

Procedure Select the objects that will be copied. Use Edit-De-Select to de-select objects that are not to 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 ENTER.

See also Edit-Paste, Edit-Select, Edit-De-Select, Edit-Toggle Selection

Paste

Shortcut SHIFT+INS or 

Summary Copies the contents of the Advanced PCB clipboard into the current document window.

Procedure First, 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

Clear

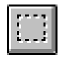
Shortcut CTRL+del

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

Procedure First, choose a document window. Use Edit-De-Select to de-select objects that should not be cleared. Use Edit-Select to select the objects that will be cleared. Choose Edit-Clear.

See also Edit-Select, Edit-De-Select, Edit-Toggle Selection

Select-Inside Area

Shortcut s, i or 

Summary Selects primitives and components within a rectangular region.

Procedure First, choose a document window.

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 ENTER 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 ENTER 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.

See also Edit-De-Select, Edit-Toggle Selection

All primitives or components that fall completely inside the rectangle will be included in the current selection and will be displayed using the color assigned to selections in the Setup Layers dialog box (Options Layers command).

Select-Outside Area

Shortcut s, o

Summary Selects primitives and components outside a rectangular region.

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

All primitives or components that fall completely inside the rectangle will be included in the current selection and will be displayed using the color assigned to selections in the Setup Layers dialog box (Options Layers command).

See also Edit-De-Select, Edit-Toggle Selection

Select-All

Shortcut s, a

Summary Adds all primitives and components within the current document window to the current selection.

Procedure First, choose a document window, then choose Edit-Select-All.

See also Edit-De-Select, Edit-Toggle Selection

Select-Free Primitives

Shortcut s, f

Summary Selects all free primitives within the current document window.

Procedure First, choose a document window, then choose Edit-Select-Free Primitives. Free primitives include pads, vias, tracks, area fills, arcs and text which are not associated with placed component library footprints. By removing all other primitives, this procedure provides a convenient method for stripping a layout back to its “placed components” condition.

See also Edit-De-Select, Edit-Toggle Selection

Select-All On Layer

Shortcut s, l

Summary Adds all items on the current layer (within the current document window) to the current selection.

Procedure First, choose a document window, then use Current-Layer to choose the layer that will be added to the current selection. Choose Edit-Select-All On Layer.

See also Edit-De-Select, Edit-Toggle Selection

Select-Physical Net

Shortcut s, n

Summary Adds all primitives on a physical net to the current selection.

Procedure To use this feature, first, choose a document window, then:

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

You will be prompted “Select Pad.”

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

All primitives in physical contact with the net will be added to the current selection and be displayed in the Selection layer color.

See also Edit-De-Select, Edit-Toggle Selection

Select-Physical Connection

Shortcut s, c

Summary Adds all primitives that are part of a physical connection to the current selection.

Procedure First, choose a document window. Choose Select-Physical Connection, then choose any non-pad primitive that is included a physical connection.

See also Edit-De-Select, Edit-Toggle Selection

Select-Hole Size

Shortcut s, h

Summary Adds all pads/vias of a designated hole diameter to the current selection.

Procedure First, choose a document window. Choose Select-Hole Size then type the target hole size into the Hole Size dialog box. All pads or vias with this hole size will be selected.

See also Edit-De-Select

Select-Off Grid Pads

Shortcut s, g

Summary Adds all component or free pads which are off the snap grid to the current selection.

Procedure First, choose a document window. Choose Select-Off Grid Pads. All off-grid pads will be selected and all on-grid pads will be displayed in draft mode, to aid in identification. This feature is provided to facilitate correction of any off-grid pads which can prevent successful autorouting.

See also Edit-De-Select, Edit-Toggle Selection

De-Select-Inside Area

Summary Removes all primitives and components within a rectangular region from the current selection.

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

See also Edit-Select, Edit-Toggle Selection

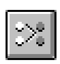
De-Select-Outside Area

Summary Removes all primitives and components outside a rectangular region from the current selection.

Procedure First, 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 deselected.

See also Edit-Select, Edit-Toggle Selection

De-Select-All

Shortcut x, a or 

Summary Removes all primitives and components within the current document window from the current selection.

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

See also Edit-Select, Edit-Toggle Selection

De-Select-Free Primitives

Summary Removes all free primitives within the current document window from the current selection.

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

De-Select-All on Layer

Summary Removes all primitives and components on the current layer within the current document window from the current selection.

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

See also Edit-Select, Edit-Toggle Selection

Toggle Selection-Arcs

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

Procedure First, choose a document window, then choose Edit-Toggle Selection-Arcs.

See also Edit-Select, Edit-De-Select

Toggle Selection-Components

Summary Selects components that were not previously selected within the document window, and deselects components that were previously selected within the document window.

Procedure First, choose a document window, then choose Edit-Toggle Selection-Components.

See also Edit-Select, Edit-De-Select

Toggle Selection-Fills

Summary Selects fills that were not previously selected within the document window, and deselects fills that were previously selected within the document window.

Procedure First, choose a document window, then choose Edit-Toggle Selection-Fills.

See also Edit-Select, Edit-De-Select

Toggle Selection-Pads

Summary Selects pads that were not previously selected within the document window, and deselects pads that were previously selected within the document window.

Procedure First, choose a document window, then choose Edit-Toggle Selection-Pads.

See also Edit-Select, Edit-De-Select

Toggle Selection-Strings

Summary Selects text that was not previously selected within the document window, and deselects text that was previously selected within the document window.

Procedure First, choose a document window, then choose Edit-Toggle Selection-Strings.

See also Edit-Select, Edit-De-Select

Toggle Selection-Tracks

Summary Selects tracks that were not previously selected within the document window, and deselects tracks that were previously selected within the document window.

Procedure First, choose a document window, then choose Edit-Toggle Selection-Tracks.

Toggle Selection-Vias

Summary Selects vias that were not previously selected within the document window, and deselects vias that were previously selected within the document window.

Procedure First, choose a document window, then choose Edit-Toggle Selection-Vias.

See also Edit-Select, Edit-De-Select

Delete-Arc

Shortcut d, a

Summary Removes an arc from the current document window.

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

See also Current-Layer

Delete-Component

Shortcut d, c

Summary Removes a component from the current document window.

Procedure First, choose a document window. If the component is not on the current layer and is below other components, choose the layer that contains the component. Choose Edit-Delete-Component and choose the component.

See also Current-Layer

Delete-Fill

Shortcut d, f

Summary Removes a fill from the current document window.

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

See also Current-Layer

Delete-Pad

Shortcut d, p

Summary Removes a pad from the current document window.

Procedure First, choose a document window. If the pad is single layer and not on the current layer or if it is below other pads, then use Current-Layer command to choose the layer that contains the pad. Now choose Edit-Delete-Pad and choose the pad.

Delete String

Shortcut d, s

Summary Removes text from the current document window.

Procedure First, choose a document window. If the string is not on the current layer and is below other strings, then use Current-

Layer command to choose the layer that contains the string. Now choose Edit-Delete-String and choose the string.

See also Current-Layer

Delete-Track

Shortcut d, t

Summary Removes a track from the current document window.

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

See also Current-Layer

Delete-Via

Shortcut d, v

Summary Removes a via from the current document window.

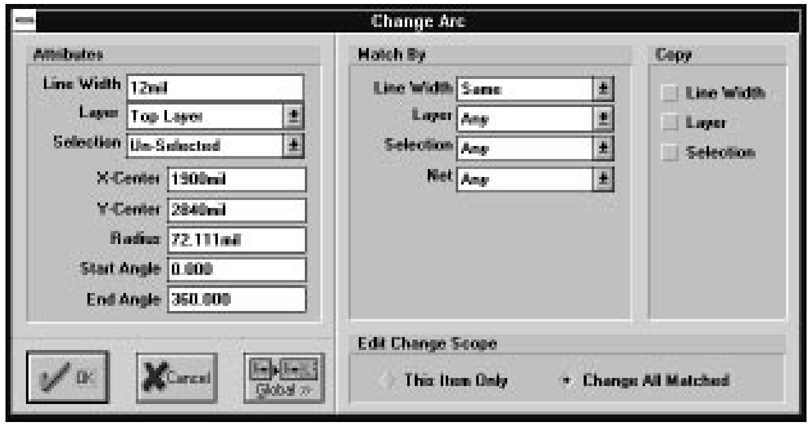
Procedure First, choose a document window. If the via is not on the current layer and is below other vias, then use Current-Layer command to choose the layer that contains the via. Now choose Edit-Delete-Via and choose the via.

See also Current-Layer

Change-Arc

Shortcut e, a or double-click on any arc segment.

Summary Opens the Change Arc dialog box. Attribute changes can be applied globally to other arcs.



Procedure Choose a document window.

1. Choose the Edit-Change-Arc command.

The status line will prompt “Select Arc.”

2. Choose an arc by positioning the cursor over the desired arc and clicking LEFT MOUSE.

The Change Arc dialog box opens. If the arc is not on the current layer and is hidden below other arcs, you may need to first use Current-Layer command to choose the layer that contains the arc. Zoom-in (shortcut: PGUP) if necessary to position the cursor accurately.

3. Make the desired changes in the Attributes fields. Editable attributes for arcs include:

Line Width — The width of the line segment used to stroke the arc.

Layer — The current layer assignment of the arc. Arcs can be placed on any board layer.

Selection — Indicates whether the arc is included in the current selection. Selected items can be manipulated using standard Windows commands, like Copy, Cut, Paste and Clear

	or moved as a group using the Edit-Move-Move Selection.
X, Y Center	Coordinates of the arc center, using the current unit of measure (mils or mm).
Radius	Distance from center coordinate to the center of the arc stroke. This value remains constant as the arc line width changes.
Start/End Angle	Radial coordinates of the arc. Range is 0.000–360.000 degrees.

The Global button is used to apply attribute changes to other arcs.

The Match By fields define which attributes are used to match with other arcs when applying global changes. Same indicates that the changes will be applied to all arcs that match the current attribute. Different applies changes to all arcs that do not match the current attribute value. Any applies the global changes to all arcs in the document window.

The Copy buttons are used to specify which globally editable changes are applied to the arcs defined using the Match By criteria.

Edit Change Scope fields specify whether the changes are applied only the current arc or to all arcs defined using the Match By criteria.

4. When all options have be set, click OK or press ENTER to apply the changes to the current item.

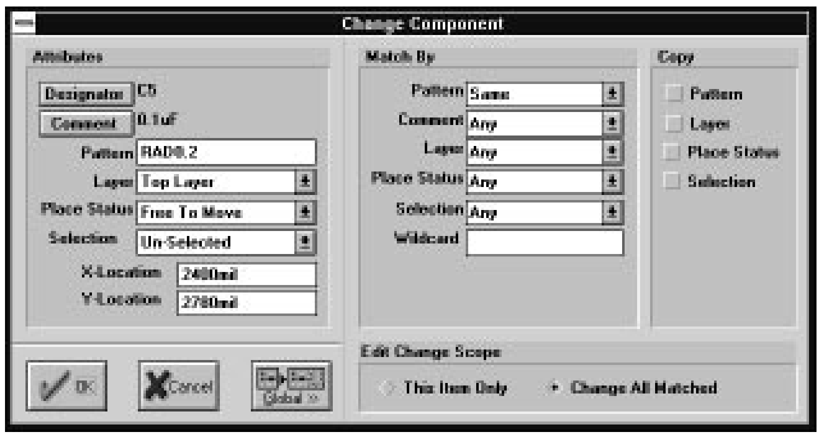
If global changes have been specified, you will be prompted to confirm those changes. As the changes are applied to the board, progress is indicated (percentage completion) to the left of the status line. If the Online DRC option is enabled (Options-Preferences dialog box) you will be prompted: “Do you want a clearance check?” If you click YES, a clearance check will be run on the board during the global edit.

Press ESC at any time to abort this process. If you make a mistake, just use the Edit-Undo command to remove the effect of these changes.

Change-Component

Shortcut e, c or double-click on any component.

Summary Opens the Change Component dialog box. Attribute changes can be applied globally to other components.



Procedure Choose a document window.

1. Choose the Edit-Change-Component command.

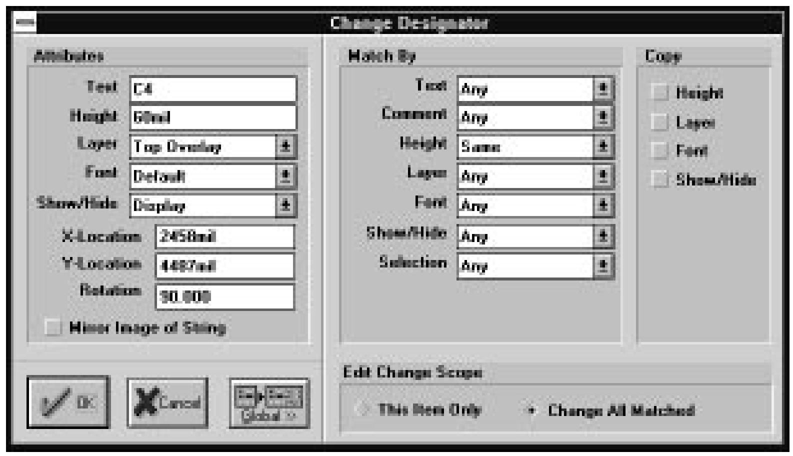
The status line will prompt “Select Component.”

2. Choose a component by positioning the cursor over the desired component and clicking LEFT MOUSE.

The Change Component dialog box opens. If the component is on the Bottom layer, you may need to first use Current-Layer command to choose the Bottom layer. Zoom-in (shortcut: PGUP) if necessary to position the cursor accurately.

3. Make the desired changes in the Attributes fields. Editable attributes for components include:

Designator — Click the Designator button to Open the Change Designator dialog box.



Designators can be individually edited and you can apply changes globally to other component designators in the board. Attributes include:

Text — The actual label or name of the component. Wildcard characters can be used to globally edit these strings.

Height — The text size in mils or mm. Text line stroke width is automatically proportioned to the text height.

Layer — The layer used to display and print or plot the designator. Any layer can be used, but typically designators will be placed on either the Top Overlay (or silkscreen) layer or on the Top (copper) layer, assuming a component placed on the Top (Component side) layer.

Font	Any of the three standard Advanced PCB fonts can be used to display the designator: Default, Sans Serif or Serif.
Show/Hide	Designators can be displayed or hidden. This does not effect printing or plotting.
X, Y Location	Coordinates of the designator reference point (lower left boundary of the text string).
Rotation	Designators can be rotated. Range is 0.001–360.00 degrees.
Mirror Image	Flips the designator along its x axis.

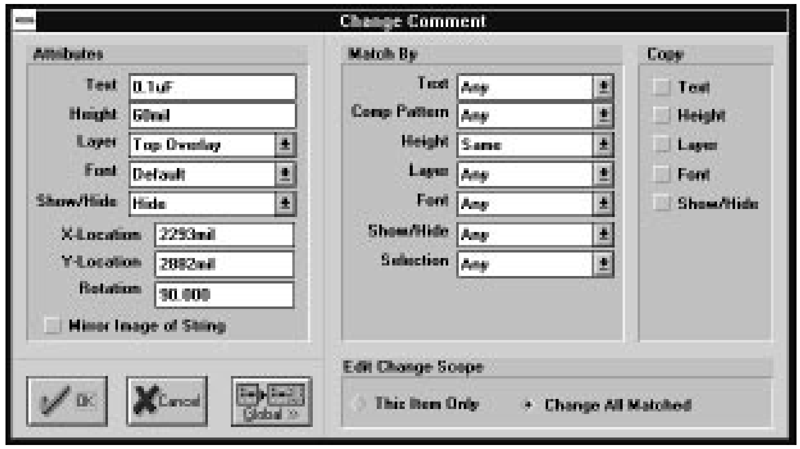
Match By fields define which attributes are used to match with other designators when applying global changes. Same indicates that the changes will be applied to all designators that match the current attribute. Different applies changes to all designators that do not match the current attribute value. Any applies the global changes to all designators in the document window.

The Copy buttons are used to specify which globally editable changes are applied to the designators defined using the Match By criteria.

Edit Change Scope specifies whether the changes are applied only the current designator or to all designators defined using the Match By criteria.

Once the desired options have been selected, click ok or press ENTER to return back to the Change Component dialog box. Additional component attributes include:

Comment	Click the Comment button to Open the Change Comment dialog box.



Comments can be individually edited and you can apply changes globally to other component comments in the board. Attributes include:

Text	The actual label or name of the component. Wildcard characters can be used to globally edit these strings.
Height	The text size in mils or mm. Text line stroke width is automatically proportioned to the text height.
Layer	The layer used to display and print or plot the comment. Any layer can be used, but typically comments will be placed on either the Top Overlay (or silkscreen) layer or on the Top (copper) layer, assuming a component placed on the Top (Component side) layer.
Font	Any of the three standard Advanced PCB fonts can be used to display the comment: Default, Sans Serif or Serif.
Show/Hide	Comments can be displayed or hidden. This does not effect printing or plotting.

X, Y Location	Coordinates of the comment reference point (lower left boundary of the text string).
Rotation	Comments can be rotated. Range is 0.001–360.00 degrees.
Mirror Image	Flips the comment along its x axis.

Match By fields define which attributes are used to match with other comments when applying global changes. Same indicates that the changes will be applied to all comments that match the current attribute. Different applies changes to all comments that do not match the current attribute value. Any applies the global changes to all comments in the document window.

The Copy buttons are used to specify which globally editable changes are applied to the comments defined using the Match By criteria.

Edit Change Scope specifies whether the changes are applied only the current comment or to all comments defined using the Match By criteria.

Once the desired options have been selected, click ok or press ENTER to return back to the Change Component dialog box. Other editable component attributes include:

Pattern	This field is the name of the component type in the Advanced PCB component footprint library. When this field is changed, Advanced PCB will search the current library for a matching pattern and, if found, the new pattern will be substituted in the PCB. Any connections which match the new component pin designators will be maintained. This option should be used with caution, as the connectivity of the board can be effected by these changes.

Layer	The current layer assignment of the component. Components can be placed on the Top or Bottom layers.
Place Status	Components can be designated either Free to Move or Locked in Place for auto placement. Locked in Place components will not be effected the Advanced PCB auto place or Advanced Place passes.
Selection	Indicates whether the component is included in the current selection. Selected items can be manipulated using standard Windows commands, like Copy, Cut, Paste and Clear or moved as a group using the Edit-Move-Move Selection.
X, Y Location	Coordinates of the component center, using the current unit of measure (mils or mm).

The Global button is used to apply attribute changes to other components.

Match By fields define which attributes are used to match with other components when applying global changes. Same indicates that the changes will be applied to all components that match the current attribute. Different applies changes to all components that do not match the current attribute value. Any applies the global changes to all components in the document window.

The Copy buttons are used to specify which globally editable changes are applied to the components defined using the Match By criteria.

Edit Change Scope specifies whether the changes are applied only the current component or to all components defined using the Match By criteria.

4. When all options have be set, click OK or press ENTER to apply the changes to the current item.

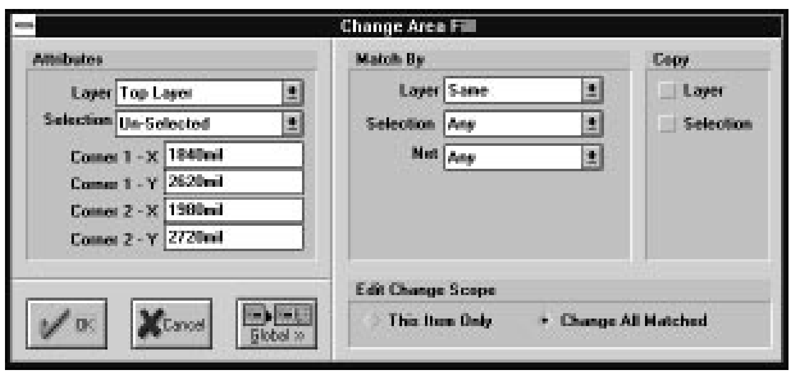
If global changes have been specified, you will be prompted to confirm those changes. As the changes are applied to the board, progress is indicated (percentage completion) to the left of the status line.

Press ESC at any time to abort this process. If you make a mistake, just use the Edit-Undo command to remove the effect of these changes.

Change-Fill

Shortcut e, f or double-click on any area fill.

Summary Opens the Change Fill dialog box. Attribute changes can be applied globally to other fills.



Procedure Choose a document window.

1. Choose the Edit-Change-Fill command.

The status line will prompt “Select Fill.”

2. Choose a fill by positioning the cursor over the desired fill and clicking LEFT MOUSE.

The Change Fill dialog box opens. If the fill is not on the current layer and is hidden below other fills, you may need to first use Current-Layer command to choose the layer that contains the fill. Zoom-in (shortcut: PGUP) if necessary to position the cursor accurately.

3. Make the desired changes in the Attributes fields.

Editable attributes for fills include:

Layer	The current layer assignment of the fill. Fills can be placed on any board layer.
Selection	Indicates whether the fill is included in the current selection. Selected items can be manipulated using standard Windows commands, like Copy, Cut, Paste and Clear or moved as a group using the Edit-Move-Move Selection.
Corner 1, (2) X, Y	Coordinates of the two opposite corners which define the fill, using the current unit of measure (mils or mm).

The Global button is used to apply attribute changes to other fills.

Match By fields define which attributes are used to match with other fills when applying global changes. Same indicates that the changes will be applied to all fills that match the current attribute. Different applies changes to all fills that do not match the current attribute value. Any applies the global changes to all fills in the document window.

The Copy buttons are used to specify which globally editable changes are applied to fills defined using Match By criteria.

Edit Change Scope specifies whether the changes are applied only the current fill or to all fills defined using the Match By criteria.

4. When all options have be set, click OK or press ENTER to apply the changes to the current item.

If global changes have been specified, you will be prompted to confirm those changes. As the changes are applied to the board, progress is indicated (percentage completion) to the left of the status line. If the Online DRC option is enabled (Options-Preferences dialog box) you will be prompted: “Do you want a clearance check?” If you click YES, a clearance check will be run on the board during the global edit.

Press ESC at any time to abort this process. If you make a mistake, just use the Edit-Undo command to remove the effect of these changes.

Change-Pad

Shortcut e, p or double-click on any component (or free) pad.

Summary Opens the Change Pad dialog box. Attribute changes can be applied globally to other pads. Two basic types are pads can be included in a design. *Component pads* are stored as part of a library component. *Free pads* are pads which are placed directly in the PCB. These pads are typically used to indicate a test point or a mechanical hole. A component pad can be edited individually, just like a free pad, however the changes will effect only the instance of the pad placed in the board, not the stored library entity. The ability to change size, hole size and shape attributes for Top, Mid 1–14 and Bottom layers allows the user to define ”pad stacks” using Multi-layer pads.

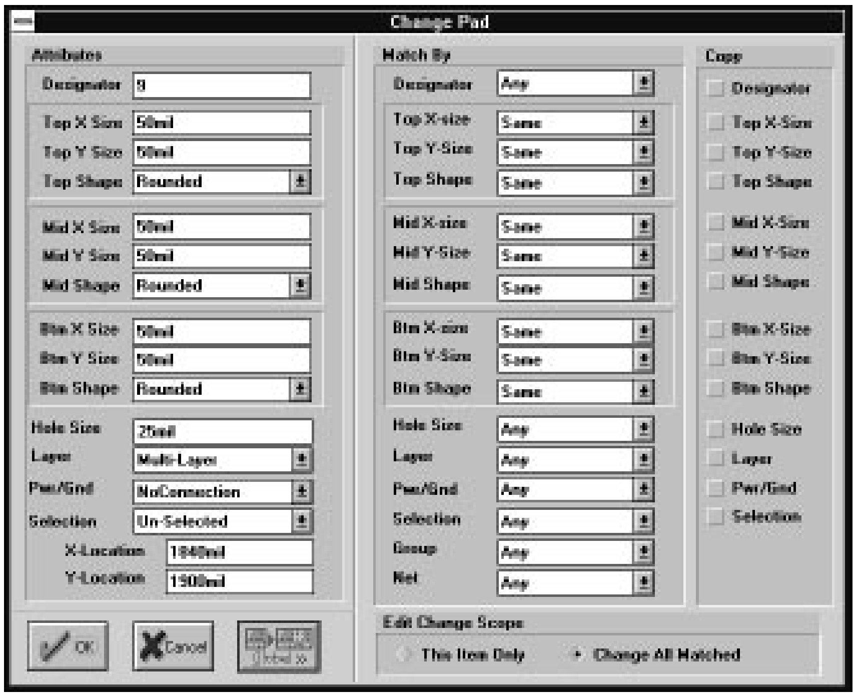
Procedure Choose a document window.

1. Choose the Edit-Change-Pad command.

The status line will prompt “Select Pad.”

2. Choose a pad by positioning the cursor over the desired pad and clicking LEFT MOUSE.

The Change Pad dialog box opens. If the pad is not on the current layer and is hidden below other pads, you may need to first use Current-Layer command to choose the layer that contains the pad. Zoom-in (shortcut: PGUP) if necessary to position the cursor accurately.



3. Make the desired changes in the Attributes fields. Editable attributes for pads include:

Designator	The pad identifier, typically a component pin number. Designators can be globally edited using wildcards to match other pad designators.
X, Y Size	Size of the pad on the Top layer, Mid layers 1–14 or Bottom layer, using the current unit of measure (mils or mm). Range is 1.000–500.000 mils.

Hole Size	The diameter of the pad hole, using the current unit of measure (mils or mm). Range is 0–500.000 mils. Zero hole size indicates "no hole."
Layer	The current layer assignment of the pad. Pads can occupy the Multi layer (with size and shape attributes for Top, Mid 1–14 and Bottom layers) or be placed as a single layer pad on any board layer.
Shape	Three pad shapes are provided: Rounded, Rectangular, Octagonal. These can be independently assigned to Top, Mid 1-14 and Bottom layers.
Pwr/Gnd	Pads can be assigned directly to any of four internal power planes and will be automatically "connected" to these planes when artwork is generated. Options are:

No Connection
Pad is not connected to any internal plane layer.

Direct to Plane (1–4)
Component pin will be in direct contact with the specified internal plane layer (0 clearance around the pin hole on layer).

Relief to Plane (1–4)
Component pin will be connected via a thermal relief to the specified internal plane layer.

Tagged to Plane (1–4)
SMD pad will be connected to the specified internal plane layer using a "tag" which is a short length of track and a special via type to connect to the layer.

Selection	Pad selection status. Selected items can be manipulated using standard Windows commands, like Copy, Cut, Paste and Clear or moved as a group using the Edit-Move-Move Selection command.
X, Y Location	Coordinates of the pad center, using the current unit of measure (mils or mm).

The Global button is used to apply attribute changes to other pads.

Match By fields define which attributes are used to match with other pads when applying global changes. Same indicates that the changes will be applied to all pads that match the current attribute. Different applies changes to all pads that do not match the current attribute value. Any applies the global changes to all pads in the document window.

The Copy buttons are used to specify which globally editable changes are applied to the pads defined using the Match By criteria.

Edit Change Scope specifies whether the changes are applied only the current pad or to all pads defined using Match By.

4. When all options have be set, click OK or press ENTER to apply the changes to the current item.

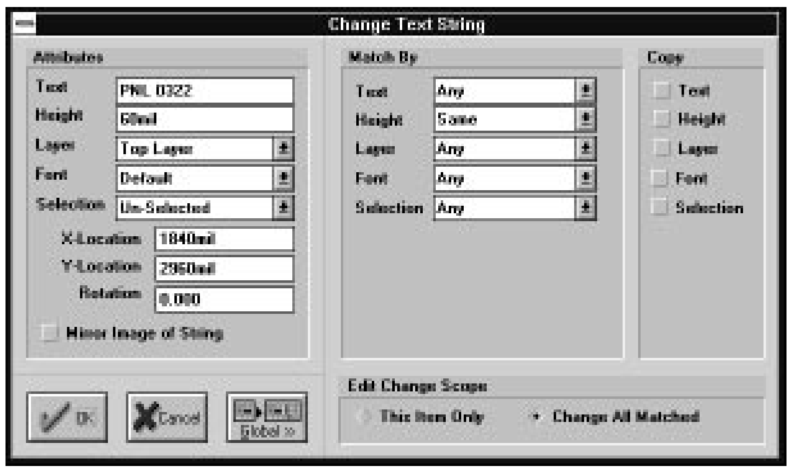
If global changes have been specified, you will be prompted to confirm those changes. As the changes are applied to the board, progress is indicated (percentage completion) to the left of the status line. If the Online DRC option is enabled (Options-Preferences dialog box) you will be prompted: "Do you want a clearance check?" If you click YES, a clearance check will be run on the board during the global edit.

Press ESC at any time to abort this process. If you make a mistake, just use the Edit-Undo command to remove the effect of these changes.

Change-String

Shortcut e, s or double-click on any free text string.

Summary Opens the Change Text String dialog box. Attribute changes can be applied globally to other text.



Procedure Choose a document window.

1. Choose the Edit-Change-String command.

The status line will prompt "Select String."

2. Choose a string by positioning the cursor over the desired string and clicking LEFT MOUSE.

The Change Text String dialog box opens. If the string is not on the current layer and is hidden below other strings, you may need to first use Current-Layer command to choose the layer that contains the string. Zoom-in (shortcut: PGUP) if necessary to position the cursor accurately.

3. Make the desired changes in the Attributes fields. Editable attributes for strings include:

Text	The actual label or name of the component. Wildcard characters can be used to globally edit these strings.
Height	The text size in mils or mm. Text line stroke width is automatically proportioned to the text height.
Layer	The layer used to display and print or plot the string. Any board layer can be used.
Font	Any of the three standard Advanced PCB fonts can be used to display the string: Default, Sans Serif or Serif.
X,Y Location	Coordinates of the string reference point (lower left boundary of the text string).
Rotation	Strings can be rotated. Range is 0.001–360.000 degrees.
Mirror Image	Flips the string along its x axis.

The Global button is used to apply attribute changes to other strings.

Match By fields define which attributes are used to match with other strings when applying global changes. Same indicates that the changes will be applied to all strings that match the current attribute. Different applies changes to all strings that do not match the current attribute value. Any applies the global changes to all strings in the document.

The Copy buttons are used to specify which globally editable changes are applied to the strings defined using the Match By criteria.

Edit Change Scope specifies whether the changes are applied only the current string or to all strings defined using the Match By criteria.

4. When all options have be set, click OK or press ENTER to apply the changes to the current item.

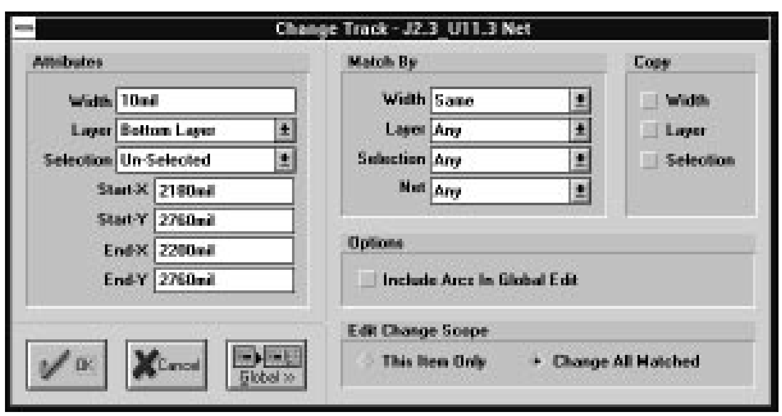
If global changes have been specified, you will be prompted to confirm those changes. As the changes are applied to the board, progress is indicated (percentage completion) to the left of the status line. If the Online DRC option is enabled (Options-Preferences dialog box) you will be prompted: “Do you want a clearance check?” If you click YES, a clearance check will be run on the board during the global edit.

Press ESC at any time to abort this process. If you make a mistake, just use the Edit-Undo command to remove the effect of these changes.

Change-Track

Shortcut e, t or double-click on any track.

Summary Opens the Change Track dialog box. Attribute changes can be applied globally to other tracks.



Procedure

Choose a document window.

1. Choose the Edit-Change-Track command.

The status line will prompt “Select Track.”

2. Choose an track by positioning the cursor over the desired track and clicking LEFT MOUSE.

The Change Track dialog box opens. If the track is not on the current layer and is hidden below other tracks, you may need to first use Current-Layer command to choose the layer that contains the track. Zoom-in (shortcut: PGUP) if necessary to position the cursor accurately.

3. Make the desired changes in the Attributes fields. Editable attributes for tracks include:

Width	The width of the track, using the current unit of measure (mils or mm). Range is .001–9999.999 mils.
Layer	The current layer assignment of the track. Tracks can be placed on any board layer.
Selection	Indicates whether the track is included in the current selection. Selected items can be manipulated using standard Windows commands, like Copy, Cut, Paste and Clear or moved as a group using the Edit-Move-Move Selection.
Start, End (X, Y)	Coordinates of the two ends which define each individual track segment, using the current unit of measure (mils or mm).

A special option, Include Arcs in Global Edits, applies global track changes to all matching arcs on the PCB, whether or not these arcs are connected to other tracks.

The Global button is used to apply attribute changes to other tracks.

Match By fields define which attributes are used to match with other tracks when applying global changes. Same indicates that the changes will be applied to all tracks that match the current attribute. Different applies changes to all tracks that do not match the current attribute value. Any applies the global changes to all tracks in the document window.

The Copy buttons are used to specify which globally editable changes are applied to the tracks defined using the Match By criteria.

Edit Change Scope specifies whether the changes are applied only the current track or to all tracks defined using the Match By criteria.

4. When all options have be set, click OK or press ENTER to apply the changes to the current item.

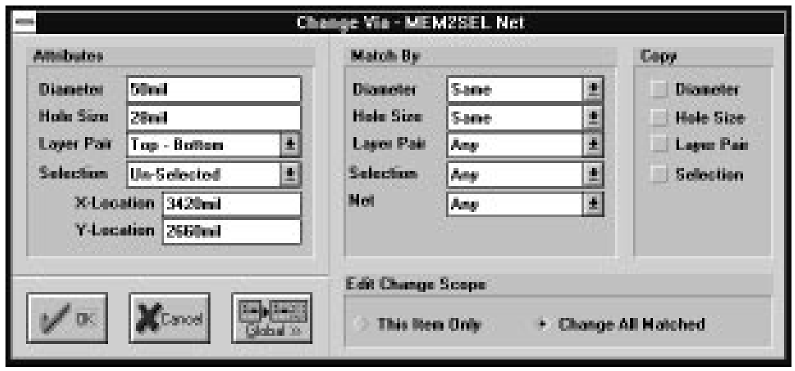
If global changes have been specified, you will be prompted to confirm those changes. As the changes are applied to the board, progress is indicated (percentage completion) to the left of the status line. If the Online DRC option is enabled (Options-Preferences dialog box) you will be prompted: “Do you want a clearance check?” If you click YES, a clearance check will be run on the board during the global edit.

Press ESC at any time to abort this process. If you make a mistake, just use the Edit-Undo command to remove the effect of these changes.

Change-Via

Shortcut e, v or double-click on any via.

Summary Opens the Change via dialog box. Attribute changes can be applied globally to other vias. Two types of vias can be present in a PCB. Free vias are placed manually by the user. These vias do not have any special connectivity to tracks, which allow the vias and tracks to be moved or dragged as a unit. Automatically placed vias (added during track placement or routing) remain connected to their track segments when the vias or tracks are moved. Identical editing features are provided for both free and automatically placed vias.



Procedure Choose a document window.

1. Choose the Edit-Change-Via command.

The status line will prompt “Select Via.”

2. Choose an via by positioning the cursor over the desired via and clicking LEFT MOUSE.

The Change Via dialog box opens. If the via is not on the current layer and is hidden below other vias, you may need to first use Current-Layer command to choose the layer that contains the via. Zoom-in (shortcut: PGUP) if necessary to position the cursor accurately. If you are using the double-click shortcut to edit the via and accidentally choose a track instead, zoom-in and move the cursor toward the outer edge of the via and try again.

3. Make the desired changes in the Attributes fields. Editable attributes for vias include:

Diameter	Size of the via, using the current unit of measure (mils or mm). Range is 0.001–500.000 mils.
Hole Size	The diameter of the via hole, using the current unit of measure (mils or mm). Range is 0–500.000 mils. Zero hole size indicates “no hole.”
Layer Pair	The current layer assignment of the via. Vias can be multi-layer (through all board layers), blind Top-to-Mid layer 1 or Bottom-to-Mid layer 14) or buried (any internal layer pair), depending upon the layer pair assignment.
Selection	Indicates whether the via is included in the current selection. Selected items can be manipulated using standard Windows commands, like Copy, Cut, Paste and Clear or moved as a group using the Edit-Move-Move Selection.
X-, Y-Location	Coordinates of the via center, using the current unit of measure (mils or mm).

The Global button is used to apply attribute changes to other vias.

Match By fields define which attributes are used to match with other vias when applying global changes. Same indicates that the changes will be applied to all vias that match the current attribute. Different applies changes to all vias that do not match the current attribute value. Any applies the global changes to all vias in the document window.

The Copy buttons are used to specify which globally editable changes are applied to the vias defined using the Match By criteria.

Edit Change Scope specifies whether the changes are applied to the current via or all vias (Match By criteria).

4. When all options have be set, click OK or press ENTER to apply the changes to the current item.

If global changes have been specified, you will be prompted to confirm those changes. As the changes are applied to the board, progress is indicated (percentage completion) to the left of the status line. If the Online DRC option is enabled (Options-Preferences dialog box) you will be prompted: "Do you want a clearance check?" If you click YES, a clearance check will be run on the board during the global edit.

Press ESC at any time to abort this process. If you make a mistake, just use the Edit-Undo command to remove the effect of these changes.

Change-Repour Polygon

Shortcut e, g

Summary Opens the Place Polygon Plane dialog box. New design rules can be applied to the selected polygon and the polygon will be repoured around any new obstacles.

Procedure First, choose a document window. If the polygon is not on the current layer and is below other vias, then use Current-Layer command to choose the layer that contains the polygon. Now choose Edit-Change-Repour Polygon and choose the polygon.

See also Edit-Place-Polygon Plane

Change-Edit Polygon Vertices

Shortcut e, y

Summary Graphically edits the perimeter of a placed polygon plane.

Procedure First, choose a document window. If the polygon is not on the current layer and is below other vias, then use Current-Layer command to choose the layer that contains the polygon.

1. Choose Edit-Edit Polygon Vertices and click LEFT MOUSE anywhere inside the polygon.

The perimeter tracks of the polygon will be highlighted.

2. Click on any perimeter track to choose the vertex nearest to the current cursor position.
3. Move the vertex to a new position and click a second time.
4. Click again to choose a new vertex or press ESC or click LEFT MOUSE to open the Place Polygon Plane dialog box.

You can add new vertices to the polygon by holding down INSERT while you click LEFT MOUSE over a perimeter track segment. To remove a vertex, press DELETE while it is being moved.

5. Redefine the polygon design rules (if desired) then click OK to redraw the polygon with the new vertices and design rules.
6. Choose another polygon the edit or press ESC again to cancel the command.

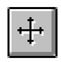
Change-Convert Selection to Fills

Shortcut e, n

Summary Recalculates selection area and replaces track segments with area fills. Using this command can simplify polygon planes or other filled areas by replacing small tracks, etc. with larger fills.

Procedure First, choose a document window. Use the Edit-Select-Physical Net to add a polygon or other primitives to the current selection. Now choose Edit-Change-Convert Selection to Fills and click inside any continuous selection. It will take a few moments to recalculate the area and convert small primitives to fills. Small or irregular areas may convert less efficiently, resulting in a greater number of overlapping fills.

Move-Move Selection

Shortcut m, m or 

Summary Moves all selected items within a document window.

Procedure First, 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 the Options-Preferences command), click at this location then choose Edit-Move-Move Selection. If the cursor is locked, you can use the m, m keyboard shortcut.

While moving a selection, you can also rotate it around the cursor and flip it along the x and y axis using the following keys:

SPACEBAR	Rotates the component clockwise. Rotation angle is specified under Options-Preferences.
X	Flip the string horizontally.
Y	Flip the string vertically.

Note Moving a selection will not drag non-selected tracks which are connected to selected items, whether the Drag Tracks option (Options-Preferences command) is enabled or not.

Move-Flip Selection

Shortcut m, i

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

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

Move-Rotate Selection

Shortcut m, o

Summary Rotates all selected items within the current document window. Items are rotated relative to a user-defined reference point, to .001 degree accuracy.

Procedure First, choose a document window. Use Edit-Select and Edit-De-select as needed, then choose Edit-Move-Rotate Selection. You will be prompted to choose a reference point, which can be any location in the document window. A dialog box allows you to enter the amount of rotation from 0.001 degree to 360.000 degrees. The selection will be rotated from the center of the reference point.

Note Rotation of selections has some important limitations which the user should be aware of prior to using this command. Rectangular component pads and area fills will have their position rotated, but will be distorted so that their sides remain parallel to the x or y axis. If components are to be included in rotations (other than 90 degree), first globally change all pad shapes to “round” to avoid this problem.

See also Edit-Select, Edit-De-Select

Move-Break Track

Shortcut m, b or press CTRL+SHIFT and click on a track segment.

Summary Creates a vertex (or break) in a track segment at the current cursor position. This vertex may be dragged to a new location when the break is formed.

Procedure First, choose a document window. If the track is not on the current layer and is below other tracks, use Current-Layer command to choose the layer that contains the track.

1. Choose Edit-Move-Break Track.

You will be prompted “Select track.”

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

The track will be broken and you will be free to move the vertex to a new location. The two attached track segments will rubberband to accommodate the move.

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

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

See also Current-Layer

Move-Drag End

Shortcut m, d or press CTRL and click at either end of a track.

Summary Moves either end point of a track segment to a new location without disturbing other connected items.

Procedure

First, choose a document window. If the track is not on the current layer and is below other tracks, use Current-Layer command to choose the layer that contains the track.

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 ENTER.

The cursor will jump to the nearest end of the track segment. The selected segment and any intersecting segments connected to the end will be highlighted.

The prompt “Dragging track end” will be displayed on the Status line.

3. Drag the track end 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

Current-Layer

Move-Whole Track

Shortcut

m, t or press CTRL + LEFT MOUSE at the middle of a track segment a track segment.

Summary

Moves a track segment without changing its orientation.

Procedure First, choose a document window. If the track is not on the current layer and is below other tracks, use Current-Layer command to choose the layer that contains the track.

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 ENTER or click LEFT MOUSE again.

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

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

See also Current-Layer

Move-Re-Route

Shortcut m, r

Summary Adds/moves segments to the current track. Similar to the Move-Break Track command, however this sequence will continue to generate additional vertices as you move the current segment and click LEFT MOUSE.

Procedure First, choose a document window. If the track is not on the current layer and is below other tracks, use Current-Layer command to choose the layer that contains the track.

1. Choose Edit-Move-Re-Route.

You will be prompted "Select Track."

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

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

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

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

4. Continue to move and press ENTER or click LEFT MOUSE to re-route the track segment-by-segment.

5. When finished, press ESC or click 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 click RIGHT MOUSE.

You can change layers at any time during this operation with the *, + or - keys. If you specify two consecutive track segments, and change layers between, then a via will be place 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.

See also Current-Layer

Move-Arc

Shortcut m, a or press CTRL and click on an arc segment.

Summary Moves an arc to a new location.

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

See also Current-Layer

Move-Component

Shortcut m, c or press CTRL and click on a component.

Summary Moves a component to a new location.

Procedure First, choose a document window. If the component is not on the current layer and is below other components, use Current-Layer command to choose the layer that contains the component. 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 Bottom Layer.

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

SPACEBAR	Rotates the component clockwise. Rotation angle is specified under Options-Preferences.
X	Flip the component horizontally.
Y	Flip the component vertically.

See also Current-Layer

Move-Fill

Shortcut m, f or press CTRL and click on a fill.

Summary Moves a fill to a new location.

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

Note A placed fill can be resized during the move command by clicking along its edges or corners.

See also Current-Layer

Move-Pad

Shortcut m, p or press CTRL and click on a free pad.

Summary Moves a pad to a new location.

Procedure First, choose a document window. If the pad is a single layer free pad and not on the current layer and is below other pads, use Current-Layer command to choose the layer that contains the pad. Now choose Edit-Move-Pad, then choose the pad.

See also Current-Layer

Move-String

Shortcut m, s or press CTRL and click on a free text string.

Summary Moves text to a new location.

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

While moving a text string, you can also rotate it around the cursor and flip it along the x and y axis using the following keys:

SPACEBAR	Rotates the string clockwise. Rotation angle is specified under Options-Preferences.
X	Flip the string horizontally.
Y	Flip the string vertically.

See also Current-Layer

Move-Via

Shortcut m, v or press CTRL and click on a via.

Summary Moves a via to a new location.

Procedure First, choose a document window. If the via is not on the current layer and is below other vias, use Current-Layer command to choose the layer that contains the via. Now choose Edit-Move-Via, then choose the via. Auto placed vias will drag connected track segments, when moved.

See also Current-Layer

Place-Arc (Center)

Shortcut p, a or 

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

Procedure First, choose a document window and use the Current-Layer command to choose a layer for placement.

To place an arc on the current layer, using the arc center as the positional reference:

1. Choose Edit-Place-Arc (Center) (shortcut: p, a).

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

2. Position the cursor where the center of the arc is to be placed and press ENTER or LEFT MOUSE once.

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

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

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

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

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

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

When using the Arc tool button, the default placement method (Center or Edge) is determined by the option last used under the Edit-Place-Arc command. PressTAB to change the current (default) track width during placement.

See also Current-Layer

Place-Arc (Edge)

Shortcut p, r

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

Procedure First, choose a document window and use the Current-Layer command to choose a layer for placement.

To place an arc on the current layer using the arc edge as the positional reference:

1. Choose Edit-Place-Arc (Edge) (shortcut: p, r).

The prompt “Select Arc Edge” is displayed on the Status line. Press TAB to change the default line width. You can change layers at any time during this operation by pressing * (to toggle active signal layers); + or – (to toggle up and down through all active layers).

2. Position the cursor where the edge of the arc is to be placed and press ENTER or LEFT MOUSE once.

The Status line will now display the Radius and Start Angle of the arc. As you move the mouse or use the cursor key array, a highlighted arc will be displayed. Note that you can now rotate the arc around the starting point while you change both the radius and starting angle.

3. Now, position the cursor to establish the desired radius and starting point, then press ENTER or LEFT MOUSE a second time.

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

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

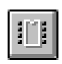
If you are drawing a 360 degree arc, you can click without moving the cursor (Start and End Angle values will be the same). As you define the end point, note that the radius does not change, regardless of the distance of the cursor from the arc edge.

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

When using the Arc tool button, the default placement method (Center or Edge) is determined by the option last used under the Edit Place Arc command. Press TAB to change the current (default) track width during placement.

See also Current-Layer

Place-Component

Shortcut p, c or 

Summary Retrieves a library component from any open footprint library for placement in the current document window. During placement the component may be moved, rotated, or mirrored.

Procedure First, choose a document window. To place a component in the workspace:

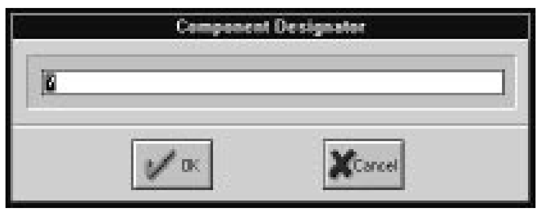
1. Choose Edit-Place-Component (shortcut: press p, c or click the Component tool button).

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

2. If you know the component package description (DIP14, for example), type it into the window and Click OK or press ENTER. If you are not sure of the package description, type *?* to list the components in all currently opened libraries.

3. Move the selection bar through the available components using the mouse or cursor keys.

4. Press ENTER or LEFT MOUSE when the correct component name is highlighted.



The Component Designator dialog box opens, displaying a "?" if no components of this pattern have been previously placed. If you just type *?* then the system will use the lowest letter (starting at A) with the lowest number (starting at 1). If components have been previously placed, the next available designator will be displayed.

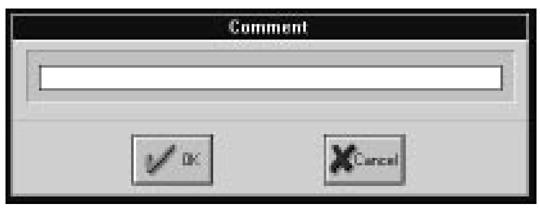
The program will now prompt for the component designator. This is the unique label for each instance of a component, "U1" for example. Note that this must be different from all other component designators current used in the design. If not, then an error message will appear. You do not have to type any designator. If you press OK, the default designator "A1" will be automatically assigned to the first component, "A2" to the next, and so on.

Designators can be up to 8 characters long and any alphanumeric character. It is suggested that you follow the letter-number format (A1, U12, etc.). This allows the auto numbering system to work reliably.

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 *u?* or *c?*. The program will then supply the next available number for this prefix.

For example, if the last designator starting with the letter “U” is “U12” and you enter *U?*, then the program will assign “U13” to the component. Remember, if you just enter ? then the program will use the lowest letter (starting at A) with the lowest number (starting at 1).

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



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

Comments, such as a component value or part number, are optional. Comments can be up to 32 alphanumeric characters, including spaces. A default comment will be supplied if a component of the same type has already been placed (assuming a comment was included, during placement).

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

6. Press ENTER or click OK to accept the default comment (if any) or type in a new comment or leave the comment field empty, then click OK or press ENTER.

7. Move the component into the desired position.

You can change Zoom and Snap Grid settings to facilitate this task.

While placing a component, you can also rotate it around the cursor and flip it along the x and y axis using the following keys:

SPACEBAR	Rotates the component clockwise. Rotation angle is specified under Options-Preferences.
X	Flip the component horizontally.
Y	Flip the component vertically.

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

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

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

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

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

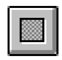
10. Type a new component type or a *?* to browse the available component types in the current libraries. Click Cancel or press ESC to exit component placement.

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

Note When placing an SMD component, the current layer should be either the Top or Bottom layer. Press * to toggle the active signal layers. Through hole components are placed on the top layer unless the Bottom layer is the current layer.

See also Library-Components

Place-Fill

Shortcut p, f or 

Summary Places a rectangular solid fill region in the current layer.

Procedure First, choose a document window and use the Current-Layer command to choose a layer for placement.

To place an area fill on the current layer:

1. Choose Edit Place Fill (shortcut: p, f or click the Fill tool button).

The prompt "Select First Corner" is displayed on the Status line.

2. Position the cursor where the edge of the fill is to be placed and press ENTER or LEFT MOUSE once.

The Status line will now display "Select Second Corner." As you move the mouse or use the cursor key array, a highlighted fill will be displayed.

3. Position the cursor to establish the fill shape, then press ENTER or LEFT MOUSE.

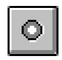
The prompt "Select First Corner" is displayed again on the Status line. Note that the fill shape is constrained by the current snap grid.

4. Start a new fill or press ESC or RIGHT MOUSE to exit this command.

Fill shape and coordinates can be edited after placement using the Edit-Change-Fill or Move shortcut (CTRL + LEFT MOUSE).

See also Edit-Change-Fill, Current-Layer

Place-Pad

Shortcut p, p or 

Summary Places a free pad. Pad types include multi-layer (through hole) and single layer (SMD). Multi-layer pad size, hole size and shape attributes can be assigned individually for Top, Mid 1–14 and Bottom layers to create “pad stacks.” Pad attributes are defined using the Current-Pad command. The pad may be moved and rotated during placement.

Procedure First, choose a document window and if placing a single layer pad, use the Current-Layer command to choose a layer for placement.

To place a free pad:

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

The prompt “Place Pad” is displayed on the Status line. Press TAB to access the Pad Types dialog box while placing pads. If placing SMD pads, you can change layers at any time during this operation by pressing * (to toggle active signal layers); + or – (to toggle up and down through all active layers).

2. When placing a pad, the cursor position defines the pad center. Move the cursor to position the pad.

3. Press ENTER or LEFT MOUSE to place one pad.

A pad will be redrawn at the selected position. Note that a highlighted pad is still displayed under the cursor and that “Place Pad” is still displayed on the Status line.

4. Press ENTER or LEFT MOUSE again to place another pad or press ESC or RIGHT MOUSE to leave this command.

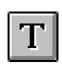
Shortcut: to place one or more (default) pads, click the Pad tool button.

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

Note Use the Edit-Place-Array command to place multiple pads, when creating a custom component, etc. Place-Array allows you to precisely pre-define the spacing of the pad row, and will automatically increment the pad number fields, if desired.

See also Current-Layer, Edit-Change-Pad, Edit-Place-Array

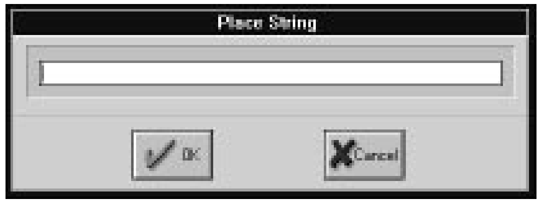
Place-String

Shortcut p, s or 

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

Procedure First, choose a document window and use the Current-Layer command to choose a layer for placement.

1. Choose Edit Place String (shortcut: p, s or click the String tool button).



The Place String dialog box opens. If a string has been placed during the current session that string will be displayed in the text line. You can change layers at any time during this operation by pressing * (to toggle active signal layers); + or – (to toggle up and down through all active layers).

Press TAB to change the current (default) string height during placement. String stroke (line) width is automatically proportioned to the string height.

2. Type a string or press ENTER or LEFT MOUSE to accept the current string.

The prompt “Moving String” is displayed on the Status line.

While placing a text string, you can also rotate it around the cursor and flip it along the x and y axis using the following keys:

SPACEBAR	Rotates the string clockwise. Rotation angle is specified under Options-Preferences.
X	Flip the string horizontally.
Y	Flip the string vertically.

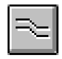
3. Position the string, click LEFT MOUSE or press ENTER.

The Place String dialog box opens.

4. Click OK or press ENTER or LEFT MOUSE again to place another string or press ESC or RIGHT MOUSE to leave the Place String command.

See also Current-Layer, Edit-Place-Component

Place-Track

Shortcut p, t or 

Summary Places tracks (or traces) on any layer. The default track width (.001–9999.999 mils) is predefined using the Current-Track command.

Procedure Choose the document window and use the Current-Layer command to choose a layer for placement.

To place tracks on the current layer:

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

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

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

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

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

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

4. Move the cursor to continue with a new highlighted track segment, which is extended from the existing

track. Click LEFT MOUSE or press ENTER again to define this segment.

If you make a mistake, you can press BACKSPACE to remove the last track segment. You can also press ESC or RIGHT MOUSE to “cancel” the current segment currently being placed.

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

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

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

Track placement mode

Track placement modes which can be selected from the Options Track Mode commands. You can toggle through the Track Mode options by pressing SPACEBAR anytime during track placement. Options include:

Any Angle
Track placed at any angle (.001–360.000 degrees).

90/90 Line
Track placed horizontal or vertically only.

45/90 Line
Constrains track angle to 0, 45, 90, 135, 180, 225, 270 or 315 degrees.

90 Arc/Line
Constrains track angle and arcs to orthogonal horizontal or vertical orientation.

See also Current-Track, Current-Layer

Place-Via

Shortcut p, v or 

Summary Places a free via on the current layer:

The via can be moved during placement, its attributes are predefined using the Current-Via command. Vias can be multi-layer (passing from the Top layer to the Bottom layer through all other layers) or confined to any two (fabricated) layer pairs – known as blind or buried vias.

Procedure First, choose a document window and use the Current-Layer command to choose a layer for placement.

To place a free via:

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

The prompt “Place Via” is displayed on the Status line. Press TAB to change the default size while placing vias.

2. When placing a via, the cursor position defines the via center. Move the cursor to position the via.
3. Press ENTER or LEFT MOUSE.

A via will be redrawn in the position selected. Note that a via is displayed under the cursor and that “Place Via” is still displayed on the Status line.

4. Press ENTER or LEFT MOUSE again to place another via or press ESC or RIGHT MOUSE to leave the Place Via command.

Possible via layer assignments include:

Layer pairs			Via type
Top layer	to	Bottom layer	(multilayer)
Top layer	to	Mid layer 1	(blind)
Mid layer 2	to	Mid layer 3	(buried)
Mid layer 4	to	Mid layer 5	(buried)
Mid layer 6	to	Mid layer 7	(buried)
Mid layer 8	to	Mid layer 9	(buried)
Mid layer 10	to	Mid layer 11	(buried)
Mid layer 12	to	Mid layer 13	(buried)
Mid layer 14	to	Bottom layer	(blind)

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

See also Current-Via, Current-Layer

Place-Polygon Plane

Shortcut p, g

Summary Creates a solid or lattice plane of copper on any signal layer. Clearances are maintained around all existing primitives and optional thermal ties can be made to pads on a defined net.

Procedure Choose the document window and use the Current-Layer command to choose a layer for placement.

1. Choose Edit-Place-Polygon Plane.

The Place Polygon Plane dialog box opens. Options include:

Connect to Net — When this option is enabled, the plane will automatically tie to any pads that are assigned to the same net. Pads will be connected with “thermal relief” style track segments, whether or not existing track connections to the pad are present.

Remove Dead Copper — This option is available only when Connect to Net has been enabled. When enabled, Remove Dead Copper deletes all “islands” of copper that do not touch the designated physical net for the polygon.

Warning If this option is enabled (along with Connect to Net) and no part of the designated net is inside the polygon perimeter (and on the same layer), the entire polygon will be regarded as “dead” and be removed, after it is generated.

Hatching Lines — Specifies the method used to fill the polygon. Choose from Horizontal, Vertical or both. If the Fills option is selected, a polygon will be regenerated with area fills replacing as many track segments as possible.

	This option will speed Gerber screen redraw, file generation and plotting.
Surround Pads with Octagons	Surrounds pads, vias or thick track segments (tracks which are at least 3x the hatching line width) using octagons, rather than complex arcs. This option will speed Gerber screen redraw, file generation and plotting.

Any same layer text strings inside the polygon will be cleanly outlined within a cleared rectangle.

2. Once all options are selected, click Place to close the dialog box.

A Net Name dialog box opens. Assign the polygon to a net in the current workfile and the message “Select Start Point On Polygon” will be displayed on the Status line.

3. Click at the starting point of the polygon.

“Line Length: 0” is displayed on the Status line. As you move the cursor, the perimeter of the plane is indicated by a highlighted line and the length of each segment is displayed. The perimeter is drawn with track segments. All track placement modes (orthogonal 90/90, orthogonal 45/90, any angle or 45/90 with curves) is supported. Press SPACEBAR to change track placement mode on-the-fly.

4. Click at each corner of the polygon until the perimeter of the polygon is defined. Press ESC or RIGHT MOUSE to “close” the perimeter of the plane.

Any remaining opening in the polygon will automatically close and the copper pour area will be generated.

Note The Fills/Tracks option will substitute area fills for parallel track segments in solid (not lattice) planes. This will replace tract segments with fill areas that can be plotted with a larger aperture, for more efficient plotting.

Place-Outline Selected Items

Shortcut p, u

Summary Places an outline of tracks and arc around a selection.

Procedure First, choose a document window and add the desired items to the current selection.

Choose Edit-Place-Outline Selected Items. Tracks and arcs of the current default width will be placed as an outline around the selection. Current primitive-to-primitive clearances will be observed, but nets are not regarded. Use Netlist-Design Rule Check to validate net integrity after using this command.

Place-Coordinate String

Shortcut p, o

Summary Places text that indicates the coordinates. These markers include a point marker (small cross of two tracks) and the x, y coordinates of the position. When you place coordinate strings, the default font type and height are used.

Procedure First, choose a document window and use the Current-Layer command to choose a layer for placement.

To place a coordinate string:

1. Choose Edit-Place-Coordinate String (shortcut: p, o).

The prompt "Select Reference Point" will be displayed.

2. Move the cursor to the desired position.
3. Press ENTER or LEFT MOUSE to place the coordinate.

The prompt "Select Reference Point" will again be displayed.

4. Place another string or press ESC or LEFT MOUSE to leave this command.

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

See also Current-Layer

Place-Dimension

Shortcut p, d

Summary Places a dimension line and dimension string between any two user-specified coordinates. The current (default) string height is used for the dimension.

Procedure First, choose a document window and use the Current-Layer command to choose a layer for placement.

To place a dimension:

1. Choose Edit-Place-Dimension (shortcut: p, d)

The prompt "Select Measure Start Point" will be displayed on the status line.

2. Position the cursor and press ENTER of click LEFT MOUSE once.

The prompt will change to "Select Measure End Point" and a highlighted line will extend from the staring point of the dimension.

3. Extend the highlighted line in any direction.

Use the coordinate display on the Status line, zoom or grid commands, as required.

4. Press ENTER or LEFT MOUSE to set the end point for the dimension.

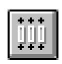
The dimension will be generated and the prompt will change back to “Select Measure End Point.”

5. Place another dimension or press ESC or RIGHT MOUSE to leave this command.

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

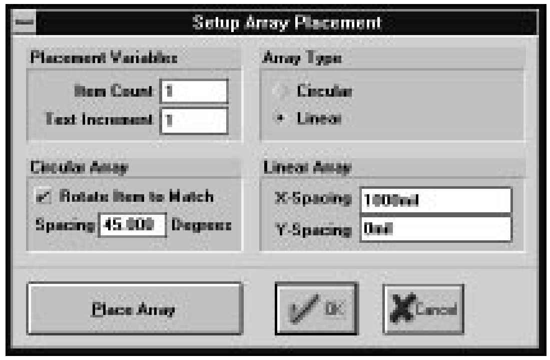
See also Current-Layer

Place-Array

Shortcut p, y or 

Summary Generates a repeated placement of the contents of the clipboard. The placement can be defined as linear (to 0.001 mil accuracy) or circular (to 0.001 degree accuracy). Each placement within the array may be individually rotated.

Procedure First, choose a document window and select the primitives or components that will be placed in the array. Then choose Edit-Copy or Edit-Cut to place the selection in the clipboard. Finally, choose Edit-Place-Array, set the dialog box options and use the OK button.



Place Array dialog box options include:

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.
Rotate Items to Match	Rotates individual items radially in circular arrays. Rectangular pads and area fills cannot be rotated in less than 90 degree increments. To rotate components, first change all pad shapes to round. Polygon planes can be substituted for rectangular fills in circular arrays, if desired.
Spacing	Circular arrays can have the rotation of the selection defined to .001 degree. Positive (counter-clockwise) or negative (clockwise) rotation can be specified.
X-offset	The horizontal distance between the next placed and the last placed item. Values can be positive or negative. Indicate negative entries by typing “-” (dash), then the digits (e.g. -3.325mils).
Y-offset	The vertical distance between the next placed and the last placed item. Values can be positive or negative. Indicate

negative entries by typing "-" (dash), then the digits (e.g. -3.325mils).

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".

See also Edit-Selection, Edit-Copy, Edit-Cut

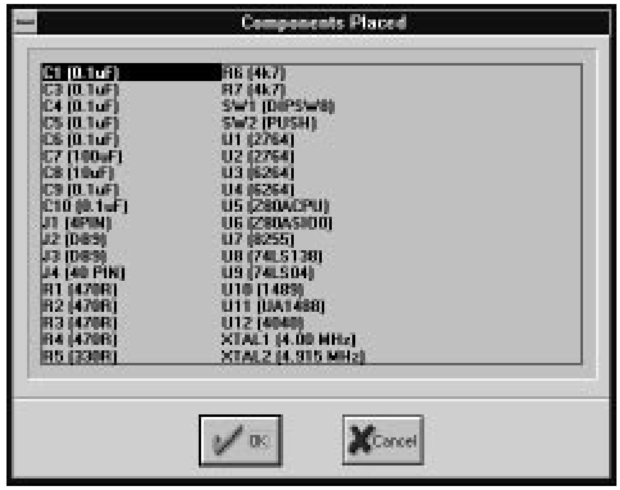
Jump-Component

Shortcut ALT, e, f, c

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

Procedure First, choose a document window. Choose the Edit-Jump-Component command to open the Component Designator dialog box.

1. Type the designator in the Jump dialog box and click OK. If you do not know the designator, type *?* and press ENTER or click LEFT MOUSE to scan the board for all placed components.



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

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

Jump-Net

Shortcut ALT, e, f, n

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

Procedure First, choose a document window.

1. Type the net name in the Jump Net dialog box and click OK. If you do not know the net, type *?* and press ENTER or click LEFT MOUSE to scan the board for all nets.

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

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

Jump-Pad

Shortcut ALT, e, f, p

Summary Moves the cursor to a pad.

Procedure Choose a document window, then Edit-Jump-Pad.

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

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

Jump-String

Shortcut ALT, e, f, s

Summary Moves the cursor to the text.

Procedure First, choose a document window.

1. Type the target string in the Jump to String dialog box and click OK.

The cursor will jump to the named string. This option works with free text strings only – not component text strings. The system will perform three searches:

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

b. If no match is found, then a string with same characters and length but ignoring case.

c. If still not found then for a string with same characters but possibly 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-Absolute Origin

Shortcut j, a

Summary Moves the cursor to the lower-left corner of the work space (default 0,0 coordinate).

Procedure First, choose a document window, then choose Edit-Jump-Absolute Origin.

Jump-Current Origin

Shortcut j, o

Summary Moves the cursor to the relative origin (0,0 coordinate) defined using the Edit-Set Origin command.

Procedure First, choose a document window, then choose Edit-Jump-Current-Origin.

See also Edit-Set Origin

Jump-New Location

Shortcut j, l

Summary Moves the cursor to new x and y coordinates.

Procedure First, choose a document window, then choose Edit-Jump-New Location and follow the status line prompts to complete the action.

Jump-Design Rule Violation

Shortcut j, d

Summary Moves the cursor to new x and y coordinates.

Procedure First, choose a document window, then choose Edit-Jump-Design Rule Violation and follow the status line prompts to complete the action. Violations are marked by highlighted primitive outlines. Correct the clearance violation by moving any offending primitive(s). Choose this command again to jump to another violation. As you repeat the command, the system will continue to jump to the next error until all violations are corrected. When no violations are present the system will "beep" when the command is used.

Set Origin

Summary Defines a location as the new origin and resets its coordinates to 0,0.

Procedure First, choose a document window.

To set a relative origin at the current cursor position:

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

The status line will now display X:00000 Y:00000 mils (or X:0000 Y:0000 mm) at the current cursor position.

To restore the absolute origin (extreme lower left corner of the workspace) choose the Edit Jump Absolute Origin command (shortcut: j, a) to return the cursor to the absolute origin. Then, choose the Edit Set Origin command again.

See also Edit-Jump-Current-Origin

Reset Origin

Summary Restores default (absolute) 0,0 origin at the extreme lower-left corner of the document window.

Procedure First, choose a document window. Then, choose Edit-Reset Origin.

See also Edit-Set Origin

Cross Probe Part On Schematic

Summary Moves focus to the corresponding Advanced Schematic document window and centers view on cross-probed part.

Procedure First, choose a document window. Then, choose Edit-Cross Probe Part On Schematic. You will be prompted to select a component. Move the cursor to a PCB component and click LEFT MOUSE.

See also Edit-Cross Probe Pin On Schematic

Cross Probe Pin On Schematic

Summary Moves focus to the corresponding Advanced Schematic document window and centers view on cross-probed component pin.

Procedure First, choose a document window. Then, choose Edit-Cross Probe Pin On Schematic. You will be prompted to select a component pad. Move the cursor to a PCB component pad and click LEFT MOUSE.

See also Edit-Cross Probe Part On Schematic

Cross Probe Net On Schematic

Summary Moves focus to the corresponding Advanced Schematic document window and highlights a cross-probed net.

Procedure First, choose a document window. Then, choose Edit-Cross Probe Net On Schematic. You will be prompted to select a connection pad. Move the cursor to a PCB component pad and click LEFT MOUSE. The schematic window will be moved to the top of the display and the corresponding schematic net will be highlighted.

See also Edit-Cross Probe Part On Schematic

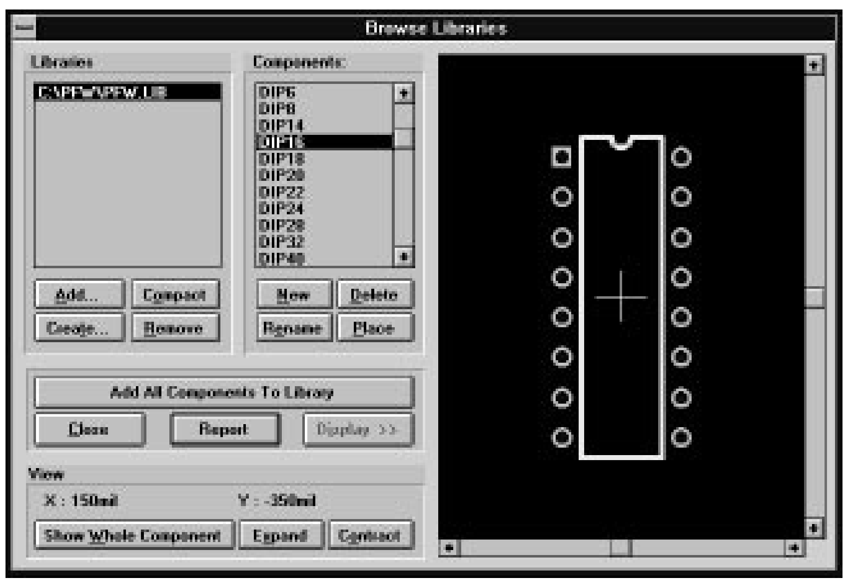
Library menu

Components

Shortcut l, c or 

Summary Opens the Browse Libraries dialog box, which allows browsing, creation, management and placing of components.

Procedure Choose Library-Components.



Browse Libraries dialog box options include:

Add Adds an additional library to the current list of open libraries.

Create	Creates a new (empty) library, and adds that library to the current list of open libraries.
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 re-sort the library and remove any fragmentation.
Remove	Removes a library from the current (open) library list.
New	Adds all currently selected primitives in the PCB document window to the selected library as a new componentt. Each library can have up to 500 components.
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).
Place	Places the selected component in the current PCB document.
Add All	(Components to Library) Copies all components in the current PCB document window to the selected destination library. Each library can hold up to 500 components.
Close	Closes the Browse Libraries dialog box.
Report	Generates a text file listing of all components in the selected library.
Display	Choosing this command will expand the dialog box. A window for viewing the components is shown together with zoom control buttons and scroll bars. The x and y coordinates displayed are the position of the

cross hair in the center of the view window relative the (stored) reference point of the component. Display options include:

Show Whole	(Component) Resets the browse window zoom level to display all primitives for the current component.
Expand	Zooms up one level to show more of the current component.
Contract	Zooms down one level show more component detail.

See also Edit-Place-Component

Un-Group

Shortcut l, u

Summary Returns a component to its free primitives.

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

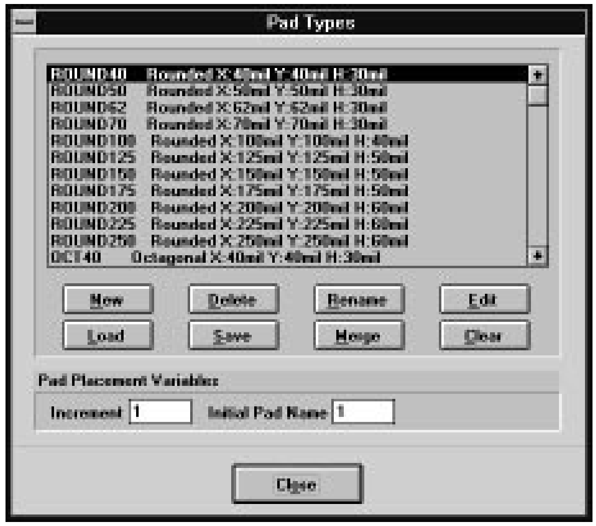
See also Current-Layer

Pad Types

Shortcut l, p

Summary Allows generation and management of pad libraries.

Procedure Choose Library-Pad Types.



This dialog box lists the current Pad Type file (.PAD) and displays the following parameters: Pad Name (e.g. ROUND40), Shape (e.g. Rounded), x size, y size and hole diameter.

Pad types dialog box options include:

New	Add a new pad type to the current list.
Delete	Remove the highlighted pad type from the current list.
Rename	Assign a new name (to 10 characters, spaces are not allowed) to the highlighted pad type.
Edit	Modify the attributes of the highlighted pad type. Editable attributes include: Name (1–10 characters), x Size, Y Size (to 5000.000 mils), Hole Size (diameter, to 5000.000), Layer Type (Multi-layer or Single layer) and Shape (Rounded, Rectangular or Octagonal). Pad size, hole size and shape can be defined independently for Top, Mid 1–14 and Bottom layers to create “pad stacks.” Global editing

of the pad type file is supported for all attributes. The list is not permanently updated until it is saved.

Load	Opens any existing pad type list.
Save	Saves any changes made to the current (open) pad type list.
Merge	Adds the current pad list to another existing list.
Clear	Clears all pad descriptions from the current list.

Pad Placement Variables allow the user to define the automatic increments and initial values for sequential pad placement.

Note When Advanced PCB is started, the library files PFW.PAD (containing pad descriptions) and PFW.LIB (standard footprint library) are automatically loaded. You can rename a custom library as PFW.LIB, and this will become the (automatically loaded) default. If you choose to do this, remember to first save the standard library under another name, or it may be overwritten and lost. Library and pad files from previous versions of Advanced PCB are automatically converted to 32-bit format when loaded and saved in the current version.

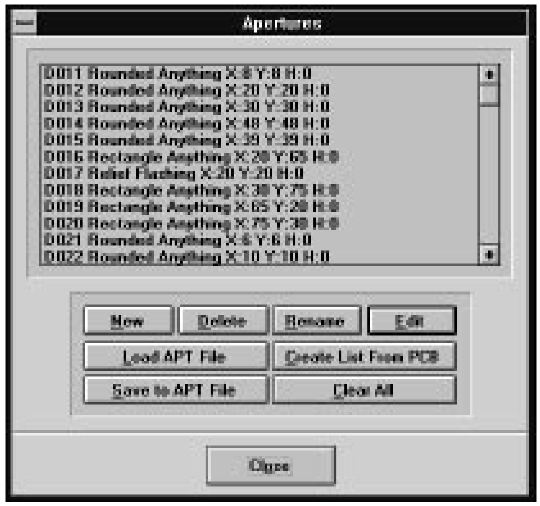
See also Current-Pad Type

Apertures

Shortcut l, a

Summary Allows generation and management of aperture tables, used to generate Gerber (RS-274) plot files.

Procedure First, choose a document window, then choose Library-Apertures.

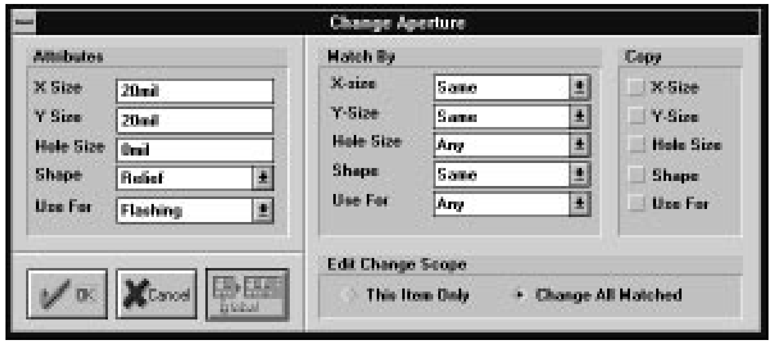


Apertures dialog box options include:

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.

The Edit option opens the Edit Aperture dialog box, allowing you to define all attributes:



Apertures listed in the table include the following parameter fields:

Draft Code	Identifies the aperture. It can be any code from D04 to D999.
Hole Size	Diameter, for “donut” style apertures. Zero (“0”) means no hole in aperture.
Shape	Options include Rounded, Rectangular, Octagonal or (thermal) Relief shape.
X Size	Horizontal dimension (to 9999 mils).
Y Size	Vertical dimension (to 9999 mils).
Use	Specifies whether the aperture can be used to draw, flash or for both uses.

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.

Note A special aperture must be provided for any file when holes are defined for any Pads or Vias in the PCB file. This is because Advanced PCB needs to provide an aperture for stroking Drill Drawing symbols and/or text. The aperture is required even if you are not generating a Drill Drawing plot. If no holes are in the file (no vias, all pad holes defined as zero, for example), the aperture is not required.

If using automatic aperture generation, see below. The required apertures for Drill Drawing symbols and other text is automatically provided with the stroke width being proportional to the text height. For example, the default 50 mil Drill Drawing symbol size yields a 7 mil aperture for stroking the symbols. 60 mil text height yields 8 mils stroke width. Users should take care in defining text sizes for Silkscreen (or Overlay) layers, as very fine strokes may not be successfully screened onto the board.

Warning If Advanced PCB is used to automatically generate an aperture table, this should be done after setting all output options and Gerber setup parameters.

Autotrax Moire and Target pad and aperture types

Autotrax board moire or target pads will be converted to arcs and tracks when loaded. These items will be painted when generating Gerber plots and not be converted back into special moire or target-type apertures.

Netlist menu

Load

Shortcut n, l

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

Procedure First, choose a document window, use the Netlist-Power Planes command to assign nets to the plane layers as needed.

To load a Protel format netlist:

Choose the Netlist-Load command.

If you have a netlist already loaded, you will be prompted "Netlist Already Loaded, Confirm Load."

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

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

After the components are loaded all nets are loaded and optimized using the (current default) strategy, i.e. shortest overall connection distance. Optimization orders connections in the net for various connection strategies.

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

Number of components loaded.

Missing patterns, e.g. netlist components which cannot be matched to the PCB patterns in the current libraries.

Existing components, i.e. previously placed components whose designators match one or more netlist components.

Nets loaded, i.e. the total number of nets in the netlist.

Errors, i.e. missing pins or missing components in the loaded netlist.

Missing Components or Pins

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

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

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

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

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

Note When a new netlist loaded, Advanced PCB compares the existing internal netlist connections (if any) with the nodes described in the new netlist. Ratsnested connections will be updated to reflect any changes. If a connection has been changed in the new netlist, the physical connection (tracks, etc.) in the PCB will be removed and a logical (ratsnest) connection will be created from the updated netlist.

New nets are optimized on the basis of the Method field for each individual net (Netlist-Edit Net command) . The Auto-Unroute commands will not work for any unreconciled connections.

See also Library-Components, Netlist-Clear, Netlist-Power Planes, Netlist-Optimize, Netlist-Edit Net

Clear

Shortcut n, c

Summary Removes the netlist from memory.

Procedure First, choose a document Window, then choose Netlist-Clear.

Note Any PCB file saved when a netlist is loaded, will be saved with the netlist attached to the PCB data. Clearing the netlist allows you to Save the PCB file without this information attached, which makes the file smaller. If you intend to make future changes to the design, you should either retain the attached netlist or keep a copy of the netlist (in its final form) as a separate (.NET) file.

See also Netlist-Load, Netlist-Generate

Optimize-Net

Shortcut n, o, n

Summary Reorders all connections on a net. Each net includes an optimization method field which can be assigned using the Netlist-Edit Net command. Optimization sorts the connections within each net per the assigned method.

Procedure First, 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.

Optimization methods include:

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 minimization 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. Minimization is performed, within the limits of the daisy chain topology, to reduce the total connection length.

Start/End Daisy Chain	Using this method, the connections in the net will appear as nodes strung along one wire which threads around the board. Each node will have two connections, except the first and last connection, which will have a single connection. Minimization is performed within this topology to reduce the total connection length. The first and last nodes in the net (from the order of nodes in the original netlist) 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, as in high speed circuits.
Star Point	This method orders connections into a star configuration, with minimization of each node radiating from the central node.

When editing individual nets, optimization method can be globally applied to other nets in the current netlist.

See also Netlist-Edit Net

Optimize-On Component

Shortcut n, o, o

Summary Re-orders the netlist connections on a Component. The sorting method is predefined using the Netlist-Edit Net command.

Procedure First, choose a document window. If the component is not on the current layer and is behind other components, use the current layer command to choose the layer that contains the component. Now choose Netlist-Optimize-On Component and choose the component.

See also Netlist-Optimize-Net, Netlist-Edit Net, Current-Layer

Optimize-All

Shortcut n, o, a

Summary Reorders all the netlist connections within the current document window. The sorting method is predefined using the Netlist-Edit Net command.

Procedure First, choose a document window, then choose Netlist-Optimize-All.

See also Netlist-Optimize-Net, Netlist-Edit Net

Show Connections-Net

Shortcut n, s, n

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

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

See also Options-Layers

Show Connections-On Component

Shortcut n, s, o

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

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

See also Options-Layers

Show Connections-All

Shortcut n, s, a

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

Procedure First, choose a document window, then turn on the Rats Nest layer using the Options-Layers command. Choose Netlist-Show Connections-All.

See also Options-Layers

Hide Connections-Connection

Shortcut n, h, c

Summary Turns off the display of a ratsnest connection.

Procedure First, choose a document window, then choose Netlist-Hide Connections-Connection and choose a ratsnest connection.

Hide Connections-Net

Shortcut n, h, n

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

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

Hide Connections-On Component

Shortcut n, h, o

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

Procedure First, choose a document window. If the component is not on the current layer and is behind other components, use the current layer command to choose the layer that contains the component. Now choose Netlist-Hide Connections-On Component and choose a component.

See also Current-Layer

Hide Connections-All

Shortcut n, h, a

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

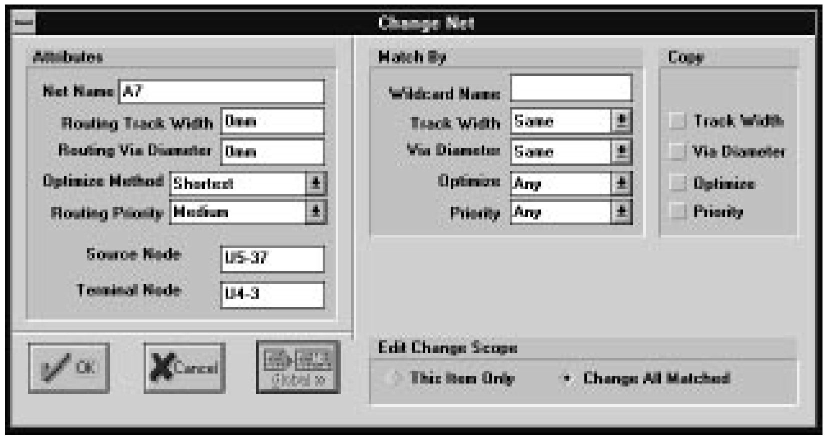
Procedure First, choose a document window, then choose Netlist-Hide Connections-All.

Edit Net

Shortcut n, e or e, n

Summary Modifies the attributes assigned to a net. These include Net Name, Routing Track Width, Routing Via Diameter, Optimize Method, and Routing Priority. Any attribute changes can by applied globally to all other nets in the current PCB document.

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



Change Net dialog box options include:

Net Name	Label currently assigned to the net. Net labels can be up to 20 alphanumeric characters long (no spaces are allowed).
Routing Track Width	Assigns track width used to autoroute this net. Overrides the default global routing width set for all nets in the Setup Autorouter dialog box.
Routing Via Diameter	Assigns the via diameter used to autoroute this net. Overrides the default global routing via diameter assigned in the Setup Autorouter dialog box. The current default via hole size is used when routing.
Optimize Method	Assigns the method used to sort connections within the net for auto placement and autorouting. See Netlist-Optimize for a description of each option.
Routing Priority	Assigns a priority for autorouting this net. Options include Highest, High, Medium, Low or Lowest.

Source Node Terminal Node	Assigns specific net nodes to be used to identify either end of a Daisy Chain topology.

See also Options-Layer, Netlist-Show

Identify

Shortcut n, i

Summary Displays the net name for a chosen ratsnest connection.

Procedure First, choose a document window then turn on the Rats Nest layer using the Options-Layers command. Display the ratsnest using the Netlist-Show command.

To identify the net associated with a physical connection:

1. Choose Netlist Identify (shortcut: n, i).

The prompt “Select Connection” will be displayed on the Status line.

2. Position the cursor over the ratsnest connection and press ENTER or LEFT MOUSE.

The name of the net will be displayed.

3. Click OK.

You will then be prompted “Select Connection” again.

4. Select another connection or press ESC or RIGHT MOUSE to finish identifying connections.

See also Options-Layers, Netlist-Show

Length

Shortcut n, n

Summary Displays the total connection distance for the netlist loaded in the current document window.

Procedure First, choose a document window, then choose Netlist-Length.

Note The length calculated is the total Manhattan (Cartesian x + y) distance of all connections in the board. This feature provides feedback, when moving components around, to find the optimum placement for minimizing routing distance.

If you have changed the layout since the last optimization, re-optimize the nets before checking the length. This step will update the internal connection list which is used to calculate the connection distance.

Add Nodes

Shortcut n, a

Summary Interactively adds pads to the current netlist and displays an updated ratsnest connection. Existing connections to the specified pad from other nets are removed. The new connection is optimized per the method assigned to the net.

Procedure First, choose a document window. A netlist must be loaded to use this feature.

1. Choose the Netlist-Add Nodes command.

The Net Name dialog box opens, displaying “?” in the net field.

2. Type in the desired net name or click OK to choose a net name from all currently loaded nets.

The status line will prompt: “Select Pad to be added to Net.”

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

The Net Name dialog box will re-open, displaying the last used net name. Continue to add new nodes, nominate another net and add new nodes or press ESC to exit this command.

Note After adding nodes, it may be necessary to use the Zoom-Redraw command (shortcut: END or Redraw button) to display the updated ratsnest.

See also Netlist- Delete Nodes, Netlist-Add Net

Delete Nodes

Shortcut n, l

Summary Interactively removes pads from the current netlist. Ratsnest connections to the specified pad from the net are removed.

Procedure First, choose a document window. A netlist must be loaded to use this feature.

1. Choose the Netlist-Delete Nodes command.

The status line prompts: “Select Pad to be removed from Net.”

2. Position the cursor over the target pad and click LEFT MOUSE or press ENTER.

The status line will again prompt: “Select Pad to be removed from Net.”

Continue to delete nodes or press ESC to exit this command.

Note After deleting nodes, it may be necessary to use the Zoom-Redraw command (shortcut: END or Redraw button) to display the updated ratsnest.

See also Netlist-Add Nodes, Netlist-Add Net

Add Nets

Summary Adds a new net to the current netlist. A new internal netlist is created if no netlist is loaded.

Procedure First, choose a document window.

1. Choose the Netlist-Add Nets command.

The Net Name dialog box opens, displaying “?” in the net field.

2. Type in the new net name and click OK or press ENTER.

The Net Name dialog box will re-open, displaying the last used net name. You can continue to add new net names or press ESC to exit this command. You can now connect nodes to this new net using the Netlist-Add Nodes command.

Note After adding nodes, it may be necessary to use the Zoom-Redraw command (shortcut: END or Redraw button) to display the updated ratsnest.

See also Netlist-Add Nodes, Netlist-Delete Nodes

Run ECO File

Summary Reads-in an .ECO (Engineering Change Order) file and updates the current loaded netlist and PCB file.

Procedure First, choose a document window, then choose Netlist-Run ECO File. All files with the extension .ECO will be listed. Choose an an ECO file and click ok or press ENTER to update the PCB.

The following changes are supported:

Add Node	A node is added to an existing net in the PCB. Any connection from that node to another net is removed. The optimization method for the net is

	applied. The ratsnest display is updated to reflect the change.
Delete Node	A node is removed from an existing net in the PCB. The ratsnest display is updated to reflect the change.
Rename Net	The name field for the net is updated to relect the new net name. If net names are displayed at the pads which represent each net node, these will be updated.
Add Component	A new component is added to the PCB and the display is updated to reflect the new pattern and connections. If the component type (pattern) is not in the current PCB library, it will be reported as a Missing Pattern.
Delete Component	An existing component and its connections are removed from the PCB. The display is updated to reflect this change.
Change Component Pattern	The Component Type field is changed to the new type and the displayed component pattern is updated to relect the change.
Rename Component	The Component Designator field is updated and the displayed designator (unless hidden) is updated.
Join Net	Two existing nets are combined, a new connection is displayed and the redundant net name is converted (where displayed on pads) to reflect this change.

Split Net	An existing net is broken into two individual nets. The ratsnest is updated to reflect this change and the new net name is added to the netlist.

Note You can use the Edit-Undo command after the Netlist-Run ECO File command, if necessary.

Export

Shortcut n, x

Summary Produces a netlist from the current netlist, loaded in memory.

Procedure First, choose a document window, then choose Netlist-Export. You will be prompted to supply a name for the netlist file (default is (filename).NET).

Generate

Shortcut n, g

Summary Produces a netlist from the current document window.

Procedure First, choose a document window, then choose Netlist-Generate.

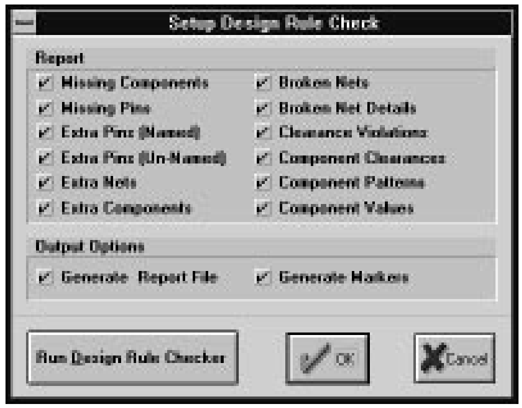
Design Rule Check

Shortcut n, d

Summary Opens the Design Rule Check dialog box allowing the user to choose DRC options and run a check of the current PCB document. Design rule check include both physical clearance checks and netlist-to-connectivity checks. A text file report and/or error display in the PCB file will be generated.

Procedure

First, choose a document window. Make sure that the correct netlist is loaded for the current workfile. Use the Netlist-Clearances command to define the physical design rules. Choose Netlist-Design Rule Check, set the report options and press OK.



User can select from the following DRC options:

Missing Components	Reports netlisted component that are not present in the PCB (by designator).
Missing Pins	Reports netlisted pins that are not present in the PCB (by component designator and pin number).
Extra Pins (Named)	Reports named pins in the PCB which are not included in the netlist.
Extra Pins (Un-Named)	Reports unnamed pins (with empty designator field) in the PCB which are not included in the netlist. Supports use of Pads to indicate mechanical holes, etc., without generating errors.

Extra Nets	Reports physical connections or nets in the PCB which are not included in the netlist.
Extra Components	Reports components (by designator) in the PCB which are not included in the netlist.
Broken Nets	Reports any nets (by name) which appear as two unconnected nets in the PCB.
Broken Net Details	Reports each unconnected subnet for broken (or partially routed) nets.
Clearance Violations	Reports locations of clearance violations.
Component Clearances	Reports components (by designator) which violate clearance rules or the Large Component/Small Component clearances defined under the Auto-Auto Place command options.
Component Patterns	Reports differences between the netlist component pattern fields and the PCB pattern.
Component Values	Reports differences between the netlist and PCB component value fields.

The Output options include: Generate Report File (ASCII text report) and Generate Markers (highlighted primitives on the DRC Errors layer). Use the Options-Layers command to assign colors or to toggle the DRC Errors layer on/off.

See also Netlist-Clearances, Netlist-Reset DRC Error Markers

Clearance Check

Shortcut n, k

Summary Runs a physical clearance check on the current PCB document. Violations will be highlighted on the DRC errors layer (use Options-Layers command to setup the DRC Errors layer). Correcting a violation will remove the display highlight on the DRC Errors layer. Use the Edit-Jump-Design Rule Violation command to sequentially display each clearance violation, until all violations are cleared.

Procedure Choose Netlist-Clearance Check.

See also Netlist-Design Rules, Netlist-Reset DRC Error Markers. Edit-Jump-Design Rule Violation

Reset DRC Error Markers

Summary Clears all highlighted errors displayed on the DRC errors layer.

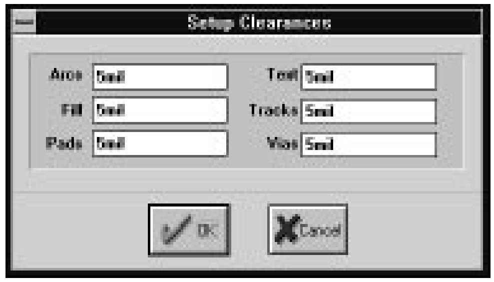
Procedure Choose Netlist-Reset DRC Error Markers.

See also Netlist-Clearances

Clearances

Shortcut n, r

Summary Assigns clearances for all primitives: arcs, fills, pads, text strings, tracks and vias. Clearances define minimum distance between primitives for design rule checking, for autorouting and for generating polygon planes.



Procedure First, choose a document window, then choose Netlist-Clearances. Type the desired minimum clearance for each primitive in the dialog box then click OK.

➡ Clearances in this system are additive – settings for two primitives are combined to define the total air gap.

See also Netlist-Design Rule Check, Netlist-Clearance Check

Power Planes

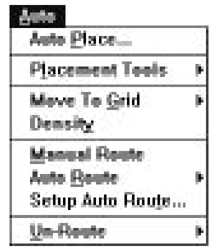
Shortcut n, w

Summary Assigns nets to intermal plane layers. Pins on these nets are automatically assigned as thermal relief connections during any netlist load procedure.

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

See also Netlist-Load

Auto menu

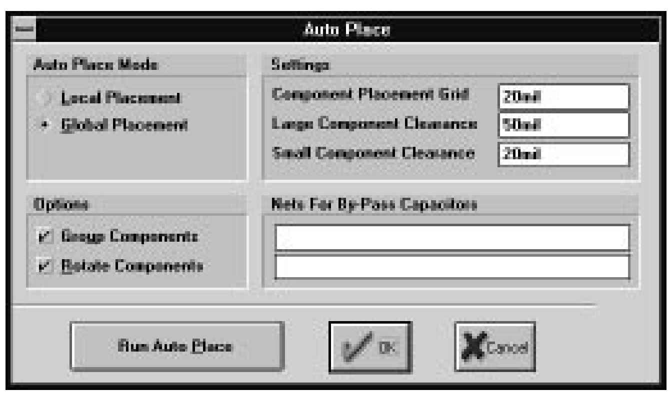


Auto Place

Shortcut a, p

Summary Provides access to auto place (Advanced PCB) and Advanced Place options. Auto placement positions netlisted components inside a predefined board area.

Procedure First, choose a document window. Use the Netlist-Load command to load the netlist and retrieve the components. Define the board outline with tracks placed on the Keep-out layer. Define "no-go" areas with rectangular area fills placed on the Keep-out layer.



When you choose Auto-Auto Place (shortcut: a, p) the Auto Place Setup dialog box opens. Choose from the following option:

Auto Place Mode options

Local Placement	This option is available for Advanced PCB and provides quick “pre-placement” of components with components evenly spread within the available space.
Global Placement	This is the Advanced Place option which uses a global optimization strategy to improve autorouter performance for large digital layouts.
Group Components	This Advanced Place option attempts to arrange components based on the optimization method, grid and clearance selections specified by the user. When the Group Components option is enabled, the system looks for component groupings based on the connections present in the netlist. Advanced Place will attempt to group components which are heavily connected together. These groups are then treated as individual components during placement.
Rotate Components	When disabled, components are placed in their current orientation. If this option is selected, Auto Place will rotate selected components in increments of 90 degrees for the best fit to the board and connection result, based on the optimization strategy assigned to each net. The default setting is “disabled.” It is recommended that you try both options with difficult placements,

	to see which gives the best result. Carefully evaluate fabrication and assembly considerations before using this option.
Settings	Grid and Clearance setup define the positioning options for placement. During placement, the reference point (usually the center of pad 1 or A1) of the component will be placed on the grid.

Large components (internally defined as having fourteen or more pins) and small components (thirteen or less pins) can be assigned different minimum clearance values.

The Large Component Clearance defines the clearance between two large components. The Small Component Clearance value defines the clearance between two small components, or between a small and large component.

Nets for By-Pass Capacitors	Bypass components are assigned to ICs on the basis of the connections in the netlist. This option specifies the nets which indicate bypass capacitors (commonly VCC and GND). Bypass capacitors are automatically grouped with, and placed across the "top" of their associated ICs.

See also Netlist-Load

Auto placement tools

Placement Tools-Spread Horizontal

Shortcut a, l, h or u, h

Summary Equalizes the space between selected components. Components between the left and right most selected components are moved to the nearest x axis snap grid that will permit evenly spaced placement. Placement status for each selected component must be Free to Move before choosing this command.

Procedure First, choose a document window then choose a horizontal group of components. If necessary, use the Change Component command to set the Place Status for each component to Free To Move. Now choose Auto-Placement Tools-Spread Horizontal.

See also Edit-Select, Edit-Change-Component

Placement Tools-Spread Vertical

Shortcut a, l, v or u, v

Summary Equalizes the space between selected components. Components between the upper and lower most selected components are moved to the nearest y axis snap grid that will permit evenly spaced placement. Placement status for each selected component must be Free to Move before choosing this command.

Procedure First, choose a document window then choose a vertical group of components. If necessary, use the Change Component command to set the Place Status for each component to Free To Move. Now choose Auto-Placement Tools-Spread Horizontal.

See also Edit-Select, Edit-Change-Component

Placement Tools-Align Left

Shortcut a, l, l or u, l

Summary Moves a group of selected components to the nearest x axis snap grid which will line up the left edges of the components. Placement status for each selected component must be Free to Move before choosing this command.

Procedure Choose a vertical group of components. If necessary, use the Change Component command to set the Place Status for each component to Free To Move. Choose Auto-Placement Tools-Align Left and choose a component to use as reference.

See also Edit-Select, Edit-Change-Component

Placement Tools-Align Right

Shortcut a, l, r or u, r

Summary Moves a group of selected components to the nearest x axis snap grid which will line up the right edges of the components. Placement status for each selected component must be Free to Move before choosing this command.

Procedure Choose a vertical group of components. If necessary, use the Change Component command to set the Place Status for each component to Free To Move. Now choose Auto-Placement Tools-Align Right and choose a component to use as reference.

See also Edit-Select, Edit-Change-Component

Placement Tools-Align Top

Shortcut a, l, t or u, t

Summary Moves a group of selected components to the nearest y axis snap grid which will line up the top edges of the components. Placement status for each selected component must be Free to Move before choosing this command.

Procedure Choose a vertical group of components. If necessary, use the Change Component command to set the Place Status for each component to Free To Move. Now choose Auto-Placement Tools-Align Top and choose a component to use as reference.

See also Edit-Select, Edit-Change-Component

Placement Tools-Align Bottom

Shortcut a, l, b or u, B

Summary Moves a group of selected components to the nearest y axis snap grid which will line up the bottom edges of the components. Placement status for each selected component must be Free To Move before choosing this command.

Procedure Choose a vertical group of components. If necessary, use the Change Component command to set the Place Status for each component to Free To Move. Now choose Auto-Placement Tools-Align Bottom and choose a component to use as reference.

See also Edit-Select, Edit-Change-Component

Placement Tools-Center On Horizontal

Shortcut a, l, c or u, c

Summary Moves a group of selected components on the x axis to the center position of a another component. This can be used to center discrete components such as bypass capacitors. Placement status for each selected component must be Free to Move before choosing this command.

Procedure Select two components. If necessary, use the Change Component command to set the Place Status for each component to Free To Move. Choose Auto-Placement Tools-Center On Horizontal and choose a reference component.

See also Edit-Select, Edit-Change-Component

Placement Tools-Center On Vertical

Shortcut a, l, i or u, i

Summary Moves a group of selected components on the y axis to the center position of a another component. This can be used to center discrete components such as bypass capacitors. Placement status for each selected component must be Free to Move before choosing this command.

Procedure Select two components. If necessary, use the Change Component command to set the Place Status for each component to Free To Move. Choose Auto-Placement Tools-Center On Vertical and choose a reference component.

See also Edit-Select, Edit-Change-Component

Placement Tools-Shove

Shortcut a, l, s or u, s

Summary Spreads selected components to make room for another component using the snap grid defined using the Auto-Auto Place command. Placement status for each selected component must be Free To Move before choosing this command.

Procedure Place or move a component into position. If necessary, use the Change Component command to set the Place Status for each component to Free To Move. Choose Auto-Placement Tools-Shove and choose a reference component.

See also Auto-Placement Tools-Shove Depth

Placement Tools-Set Shove Depth

Shortcut a, l, d or u, d

Summary Sets the number of components to be cleared by the Shove command. For example, if the number is “1” only the initial component is cleared from the overlapping components. If “2” is set, shove will clear the first component, then the components that were moved to uncover the first component.

Procedure Place or move a component into position. If necessary, use the Change Component command to set the Place Status for each component to Free To Move. Choose Auto-Placement Tools-Shove and choose a reference component.

See also Auto-Placement Tools-Shove

Move To Grid

Shortcut a, g

Summary Moves each free to move component to the nearest location that will align its reference point with the grid. Placement status for each selected component must be Free To Move before choosing this command.

Procedure First, choose a document window. If necessary, use the Change Component command to set the Place Status for each component to Free To Move. Choose Auto-Move To Grid, choose a grid, or choose other to enter another value.

Density

Shortcut a, y

Summary Temporarily displays a map of the ratsnest connection density. This display can be used to predict routing difficulty for a given component placement. Depending upon the graphics card/driver combination, a number of density steps are displayed, indicating the relative density across the entire layout. Red is used to identify the most dense areas, green the least.

Procedure First, choose a document window. If a netlist is not loaded, use Netlist-Load. Choose Auto-Density. To clear the density display, press END or choose any other command which redraws the screen (e.g. Zoom commands, etc.).

See also Netlist-Load

Manual Route

Shortcut a, m

Summary Places tracks interactively for each board connection, using a ratsnest connection as a guide. Ratsnests may be partially routed leaving the ratsnest attached to the end of the last placed track.

Procedure The ratsnest is usually displayed immediately after loading a netlist. If not, choose a document window, then use Options-

Layers command to toggle on the Rats Nest layer. Use the Netlist-Show Connections-command to display the ratsnest.

To route a single connection:

1. Choose Auto-Manual Route (shortcut: a, m).

The prompt Select Connection will be displayed on the Status line.

2. Position the cursor close to a (ratsnest) connection and press ENTER of click LEFT MOUSE.

The cursor will jump to the nearest end of the connection. As you move the cursor, a highlighted track will rubberband from the starting point.

As during manual track placement, you can use the grid, zoom and toggle layer shortcuts (press * to toggle through the active signal layers) while manually routing. Press spacebar to change the track mode between “Any Angle,” “90/90,” “90/45” or “Curved.”

3. Click to place the end of each track segment.

As you route the connection, the ratsnest will be maintained from the current cursor position to the target pad.

Press * to change signal layers and automatically place a via (when the Auto Via option is selected from the Options-Preferences dialog box). You can press ESC to terminate the current track segment. Note that the ratsnest is maintained from the end of the last segment placed to the target pad and that the prompt “Select Connection” is again displayed on the Status line. You can go back at any time and resume routing the partially completed connection.

4. Extend the route to the target pad and press ENTER or LEFT MOUSE.

The last segment will be completed and redrawn without the highlight. If the connection is part of a net that extends beyond the pad, the next connection will be automatically "picked-up" by the cursor.

5. Continue placing tracks, or press ESC to release the ratsnest.

Note that the prompt "Select Connection" is still displayed, signifying that you are still in Manual Route mode.

6. Select another connection or press ESC again to leave manual route.

Press the ESC key to halt the process at any time. You can backtrack from the current routed position by pressing BACKSPACE as you route. Each press of BACKSPACE will remove one segment, restoring the ratsnest for that connection. If you are unhappy with the result of your route you can use Undo to remove the route and restore the ratsnest. You can go back at any time and use the Auto Un-Route command to remove a route and restore the ratsnest connection.

Note If the Auto Via option is enabled (Options-Preferences command) then vias can be placed automatically as you toggle the available routing layers. Layers can be toggled, on-the-fly, by pressing the *, + or - keys without terminating the Route-Manual command. Routing layers can be activated from the Options-Layers dialog box or from the layer control button in the status line.

When using Route-Manual, the track width and via size are set in the Auto-Setup Auto Route dialog box, not the settings in the Current menu.

See also Options-Layers, Netlist-Show Connections, Place-Track

Auto Route-All

Shortcut a, r, a

Summary Routes all connections on all nets within the current document window. Routing methods are determined using the Auto-Setup Auto Route command.

Procedure First, choose a document window, then use the Auto-Setup Auto Route command to define the routing parameters. Choose Auto-Auto Route-All and follow the status line prompts to complete the action. All currently set routing passes (for the standard line probe router and for Advanced Route, if available) will be run.

Note The route priority field (part of the attached netlist portion of a .PCB file) determines the order in which connections are routed by both the Advanced PCB line probe router and by Advanced Route. You can assign route priority from the Netlist Edit Net command. If no priority is established, nets are routed in the order in which they appear in the netlist. Netlists can be manually edited to change the autorouting order, using any text editor.

See also Auto-Setup Auto Route, Netlist-Edit Net

Auto Route-Net

Shortcut a, r, n

Summary Automatically routes a net.

Procedure First, choose a document window, then use the Auto-Setup Auto Route command to define the routing parameters. Choose Auto-Auto Route-Net and enter the name of a net, or enter *?* to choose from currently loaded nets. All currently set routing passes (for the standard line probe router and for Advanced Route, if available) will be tried on the net.

See also Auto-Setup Auto Route

Auto Route-Connection

Shortcut a, r, c

Summary Automatically routes a ratsnest connection.

Procedure First, choose a document window, then use the Auto-Setup Auto Route command to define the routing parameters. Use the Options-Layers command to show the Rats Nest layer then Netlist-Show Connections-to display the ratsnest lines. Choose Auto-Auto Route-Connection, then choose a ratsnest line. The standard line probe router will be tried on the connection. The balance of any net will be left unrouted.

See also Auto-Setup Auto Route

Auto Route-On Component

Shortcut a, r, o

Summary Automatically routes all ratsnest connections on a component.

Procedure First, choose a document window, then use the Auto-Setup Auto Route command to define the routing parameters. Choose Auto-Auto Route-On Component, then choose a component. All currently set routing passes (for the standard line probe router and for Advanced Route, if available) will be tried on the connections (not nets) associated with the component. The balance of any nets will be left unrouted.

See also Auto-Setup Auto Route

Auto Route-Selected Components

Shortcut a, r, o

Summary Automatically routes all ratsnest connections on a group of selected components.

Procedure First, choose a document window, then use the Auto-Setup Auto Route command to define the routing parameters. Choose Auto-Auto Route-On Component, then choose a component. The standard line probe router will be tried on the connections (not nets) associated with the components. The balance of any nets will be left unrouted.

See also Auto-Setup Auto Route

Auto Route-Pad To Pad

Shortcut a, r, p

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

Procedure First, choose a document window, then use the Auto-Setup Auto Route command to define the routing parameters. Choose Auto-Auto Route-Pad To Pad, then choose two component pads as prompted by the status line. The standard line probe router will be tried on the connection.

Auto Route-Advanced Route Connections

Shortcut a, r, p

Summary Automatically routes tracks between two pads in a netlisted connection using Advanced Route passes. This autorouting function requires a netlist.

Procedure First, choose a document window, then use the Auto-Setup Auto Route command to define the routing parameters.

Choose Auto-Auto Route-Advanced Route Connections. The Advanced Route (Maze) passes will be tried on the connection. Any balance of the net will be left unrouted.

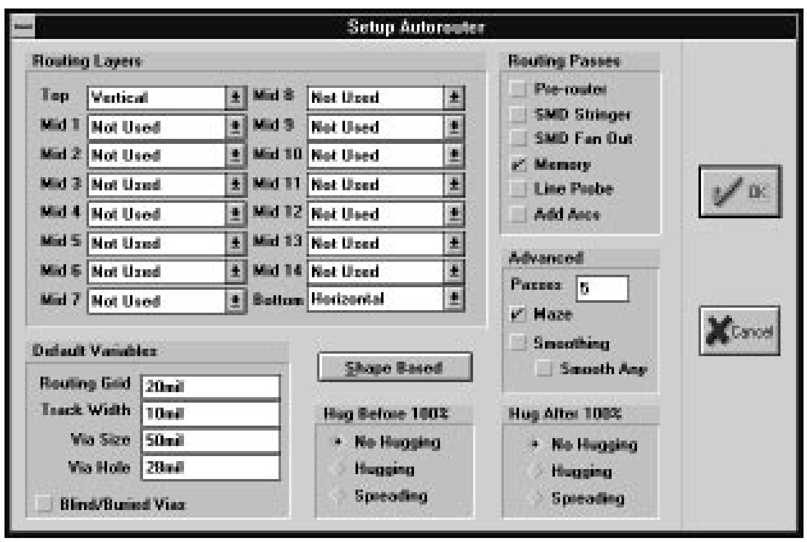
See also Auto-Setup Auto Route

Setup Auto-Route

Shortcut a, t

Summary Defines use of Advanced PCB and Advanced Route passes and options. Options are also applied when using the Auto-Manual Route command or when using Advanced SB Route.

Procedure First, choose a document window. Choose Auto-Setup Auto Route to define the parameters.



These options include:

Routing layers

Sixteen layers are available for routing. This option assigns the layer bias for each routing layer. Four options are available for each layer:

Not Used	The layer is disabled and will not be used for routing.
Horizontal	Tracks will be placed in a horizontal orientation. Short vertical tracks will be placed on this layer to avoid planting a via.
Vertical	Tracks will be placed in a vertical orientation. Short horizontal tracks will be placed on this layer to avoid planting a via.
No Preference	Router will place tracks either horizontally or vertically. If only one active routing layer is available, router assumes a single sided board with no vias.

If you want to route multilayer without vias, route each of the layers separately using No Preference. Then, when running Smoothing, route multiple layers. The smoother won't reroute with more vias than there were in the original route.

Routing Passes

These passes can be run independently, in any order. When more than one pass is enabled, the passes run in the order that they appear in the dialog box: Pre-router, SMD, Memory and so on.

Pre-router	This pass checks all the connections and makes sure that the router will only attempt to route any that have not been previously completed.

When using the Manual Route command, completed connections are removed from the (internal) netlist and this option is not needed to protect these routes. Default is "Off."

SMD Stringers	When using power and ground planes with SMD components, this option will route those pins that need to be connected to the planes (with a short track segment, through hole pad and thermal relief connection to the plane). Default is “Off.”
SMD Fan Out	Routes “fan out” track structures for SMD components. This option helps avoid clogged routing channels around SMD parts.
Memory Router	A fast heuristic pattern routes a wave structure for memory busses. Default is “On.”
Line Probe	This option enables a number of line probe and pattern routing passes. They use a basic L, C or Z shape, with the separate sections completed by a line probe router. Default is “On.”
Add Arcs	Replaces any corners (on one layer) with a 90 degree arc, wherever the current clearance settings permit. Default is “Off.”

Advanced Route

These passes are part of the Advanced Route package and run after all enabled line probe router passes are completed. Options include:

Maze Router	A gridded wave expansion router with rip-up and re-try capabilities. It will rip-up and reroute other connections which block its path.
Smoothing	Checks the board for vias that can be removed by swapping tracks from one

	side of the board to the other; minimizes the number of track segments and cleans-up parallel tracks on the same net.
Smooth Any	When enabled, this option applies smoothing to Pre-routes (connections completed other than by an Advanced PCB router, e.g. manually placed tracks).

Smoothing works by ripping up the entire net and rerouting it with different internal costs. Smoothing places high priority on reducing the via count and copper sharing. The smoother will not smooth pre-routes unless the Smooth Any option is enabled. It will smooth manually routed tracks. Running multiple smoothing passes will continue to improve the result over several passes. Use the Router Log file to monitor progress made during each pass. Smoothing may take a long time on a complex board.

Passes	Sets the number of passes for the Maze and/or Smoothing passes The board will be routed as high as possible using the maze route then all the connections that will have been routed will be rerouted using the smoothing pass. With each pass, the process is repeated again to improve the completion of those last difficult connections. Maze and Smoothing passes can require considerable time to complete. The Maze and Default setting is “5.”

Default Variables

These values are used by the router. They are independent of the Current (default) settings.

Grid Size	Grid used to route the board and will set the location of tracks, vias and arcs placed during autorouting.

For all the routing passes, decreasing the grid size will improve the completion rate, but slow the router and increase the memory usage during the Maze router passes. The default value is 25 mils. The range is .005–100.000 mils.

Warning The placement grid should be an even multiple of the routing grid to prevent off-grid pins that make autorouting more difficult.

Track Width	Controls the width of any tracks, straight or curved, put in by the router. Range is 1.000–9999.999 mils.
Via Siz/Hole	The diameter of any vias place by the router. Range is 1.000–500.000 mils. Default is 50 mils (with a 28 mil hole).

Note Track width and via size settings will be overridden by track or via settings predefined for individual nets, using the Netlist-Edit Net command (shortcut: e, n or n, e).

Blind/Buried Vias	When enabled, the router will try to route between layer pairs where valid blind or buried via placements can be made. This will improve the density of the board when four or more signal layers are used.
Shape Based	Provides access to Advanced SB Route setup dialog boxes.

Hugging/Spreading options

Hugging routes tracks adjacent to existing tracks to conserve as much of the vacant board space as possible for future routes to improve completion for tight layouts.

Spreading does the opposite and will try to equalize the distance between tracks, thereby making maximum use of the available space for a more “manufacturable” result.

Hug Before 100%	Applies hugging or spreading prior to the initial completion of 100% of all routes. The No Hugging setting disables this feature.
Hug After 100%	Applies hugging or spreading after the board has been completely routed.

See also Netlist-Design Rules

Un-Route-All

Shortcut a, u, a

Summary Removes all tracks and vias on all valid netlist connections within the current document window. Restores (ratsnest) connections.

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

Note Auto-Un-route commands will not remove placed tracks unless they were routed from the current netlist (routed using either the autorouter or the Auto-Manual Route command).

When a new netlist is loaded, Advanced PCB compares the existing internal netlist connections (if any) with the nodes described in the new netlist. Existing and any new nodes are matched. Existing unrouted connections will be updated, if changed and re-optimized. If a connection has been changed in the new netlist, the physical connection (tracks, etc.) in the PCB will be removed and a logical (ratsnest) connection will be created from the updated netlist. Auto-Unroute commands will not work for any unreconciled connections.

Files created in Protel Autotrax cannot be unrouted, as there is insufficient information in the Autotrax file format. To unroute all of the tracks and vias on an Autotrax board, use the Auto-Unroute All command to clear the connections in the internal connection list. Track and vias will not be removed. The Select Free Primitives command and be used to first select and then delete all tracks and vias.

See also Netlist-Load

Un-Route-Net

Shortcut a, u, n

Summary Removes all free primitives on all valid netlist connections on a net. Restores (ratsnest) connections.

Procedure First, choose a document window. Choose Auto-Un-Route-Net and follow the status line prompts to complete the action.

See also Auto-Unroute All

Un-Route-Connection

Shortcut a, u, c

Summary Removes all free primitives on a valid netlist connection. Restores (ratsnest) connections.

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

See also Auto-Unroute All

Un-Route-On Component

Shortcut a, u, o

Summary Removes free primitives on valid netlist connections on a component. Restores (ratsnest) connections.

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

See also Auto-Unroute All

Un-Route-Track

Shortcut a, u, t

Summary Converts a routed track segment back into an unrouted ratsnest. The track must have connection information (e.g. a netlist must be loaded and the track must have been placed by the autorouter or Auto-Manual Route command).

Procedure Choose a document window. Choose Auto Un-Route Track, then click on a routed track segment. Continue to “rip-up” segments one-at-a-time, providing that they are contiguous. A connection cannot be broken at two locations (e.g., at either end) at the same time. Obsolete vias are removed.

See also Auto-Unroute All

Un-Route-Selected Components

Shortcut a, u, o

Summary Removes free primitives on valid netlist connections on a component. Restores (ratsnest) connections.

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

See also Auto-Unroute All

Un-Route-Selection

Shortcut a, u, s

Summary Removes all selected free primitives on valid netlist connections. Restores (ratsnest) connections.

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

See also Auto-Unroute All, Edit-Select, Edit-De-Select

Current menu

Pad Type

Shortcut c, p or CTRL+P

Summary Defines the default pad attributes that will be assigned during subsequent pad placements. This pad type is also displayed on the status line.

Procedure Choose Current-Pad Type.

See also Library-Pads

Track

Shortcut c, t or CTRL+T

Summary Defines the default track width that will be assigned during subsequent manual track placements. This track width is displayed on the status line. Range is 1–255 mils.

Procedure Choose Current-Track, choose a track width, or choose Other to enter another value.

Via-Diameter

Shortcut c, v, d

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. Range is 2.000–500.000 mils.

See also Current-Via-Hole Diameter

Via-Hole Diameter

Shortcut c, v, h

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. Range is 2.000–500.000 mils.

See also Current-Via-Diameter

Via-Type-Through Hole

Shortcut c, v, t, t

Summary Defines Through Hole as the type of via that will be assigned 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 c, v, t, b

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

Procedure Choose Current-Via Type-Blind/Buried.

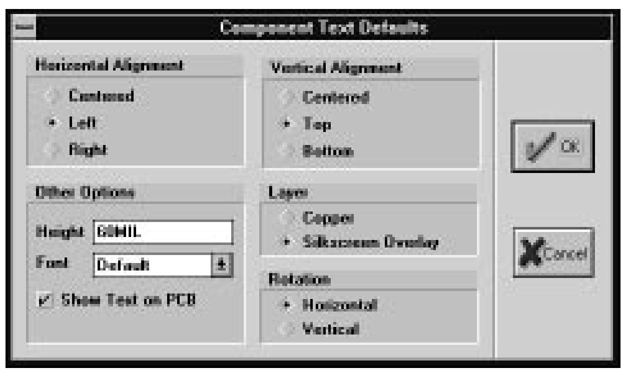
See also Current-Via Type-Through Hole

Component Text-Comments

Shortcut c, c, c

Summary Defines the component comment attributes that will be assigned during subsequent component placements.

Procedure Choose Current-Component Text Comments.



The options include:

Horizontal Alignment	Sets horizontal position of comment text relative the component outline.
Vertical Alignment	Sets vertical position of comment text relative the component outline.

Layer	Places comment text on the top (or bottom) silkscreen overlay or on the top or bottom (copper) layer.
Height	Default size of text in mils or mm, depending upon current unit of measure.
Font	Default font when placing component comment text.
Show Text on PCB	If disabled, then the text will not be displayed nor printed/plotted. No information is lost by hiding the comment. It can be un-hidden at any time.
Rotation	Sets the comment text orientation to Horizontal or Vertical.

Note Component text attributes can be changed after placement from the Change Component dialog box (Edit-Change-Component command).

See also Current-Component Text Designators

Component Text-Designators

Shortcut c, c, d

Summary Defines the default component designator attributes that will be assigned during subsequent component placements.

Procedure Choose Current-Component Text Designators.

The options are:

Horizontal Alignment	Sets horizontal position of designator text relative the component outline.

Vertical Alignment	Sets vertical position of designator text relative the component outline.
Layer	Places designator text on the top (or bottom) silkscreen overlay or on the top or bottom (copper) layer.
Height	Default size of text in mils or mm, depending upon current unit of measure.
Font	Default font when placing component designator text.
Show Text on PCB	If disabled, then the text will not be displayed nor printed/plotted. No information is lost by hiding the designator It can be un-hidden at any time.
Rotation	Sets the comment text orientation to Horizontal or Vertical.

Note Component text attributes can be changed after placement from the Change Component dialog box (Edit-Change-Component command).

See also Current-Component Text Comments

Free Text-Height

Shortcut c, f, h or CTRL+S

Summary Defines the text height that will be assigned during subsequent placements of free text.

Procedure Choose Current-Free Text Height, choose a size or choose Other to enter another value between 0.100–4000.000 mils.

Free Text-Font Default

Shortcut c, f, f, d

Summary Defines that the font assigned during subsequent placements of free text will be the internal Advanced PCB font.

Procedure Choose Current-Free Text Font Default.

See also Current-Free Text Font Serif, Current-Free Text Font Sans Serif

Free Text-Font Sans Serif

Shortcut c, f, f, a

Summary Defines that the font assigned during subsequent placements of free text will be Sans Serif.

Procedure Choose Current-Free Text Font Sans Serif.

See also Current-Free Text Font Sans Serif, Current-Free Text Font Default

Free Text-Font Serif

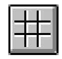
Shortcut c, f, f, s

Summary Defines subsequent placements of free text as Serif.

Procedure Choose Current-Free Text Font Serif.

See also Current-Free Text Font Default, Current-Free Text Font Sans Serif

Snap Grid

Shortcut c, g or CTRL+G or 

Summary Defines the alignment grid for manual movement and placement.

Procedure First, choose a document window. Choose Current-Snap Grid, then choose a size or choose Other to enter another value between 0.001–1000.000 mils.

Visible Grid 1, 2

Shortcut c, 1 (or c, 2)

Summary Defines the first display grid.

Procedure First choose Current-Visible Grid 1 (or 2), then choose a size or choose Other to enter another value between 0.001–1000.000 mils.

Layer

Shortcut c, l or CTRL+L

Summary Chooses a current layer, from all layers, including inactive layers. Once chosen, a layer becomes active. The current layer is displayed on the status line. Change layer and layer color assignment by choosing the Options-Layers command.

Procedure First, choose a document window. Choose Current-Layer, then choose a layer from the list box.

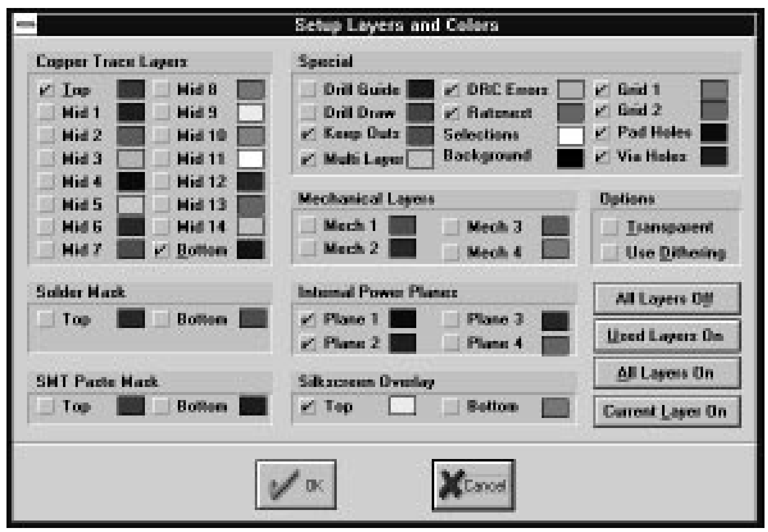
See also Options-Layers

Options menu

Layers

Shortcut o, l

Summary Toggles layer display and defines colors.



Procedure To activate layers and assign layer colors:

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.

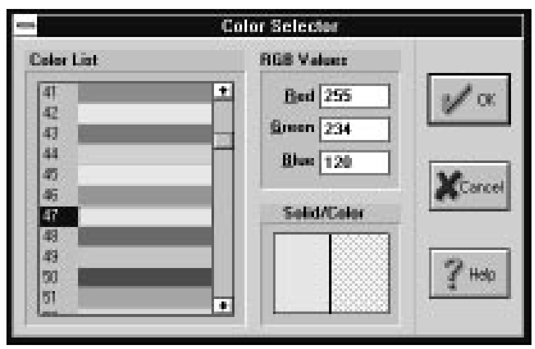
For each of the layers on the printed circuit board there is a check box which you can click (LEFT MOUSE) to turn the layer on or off, just to the left of the layer name. An “x” in the box indicates that a layer is active.

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. Click in the color box, to the right of the layer name to open the Color Selector box.



The Advanced PCB Color Selector includes 233 predefined RGB colors. If your system has a 24-bit graphics card (and the appropriate driver) installed. All of these colors are available as solid (rather than dithered, or mixed) shades.

2. Scroll through the Color List to choose from one of the available shades, or enter a Red/Green/Blue combination to define a custom color.

The Color/Solid boxes display the current layer color assignment and the nearest available solid color.

3. Click in the Solid (right) pane to assign the solid color equivalent for a dithered assignment, if desired.

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

Other options in the Setup Layers and Colors dialog box include:

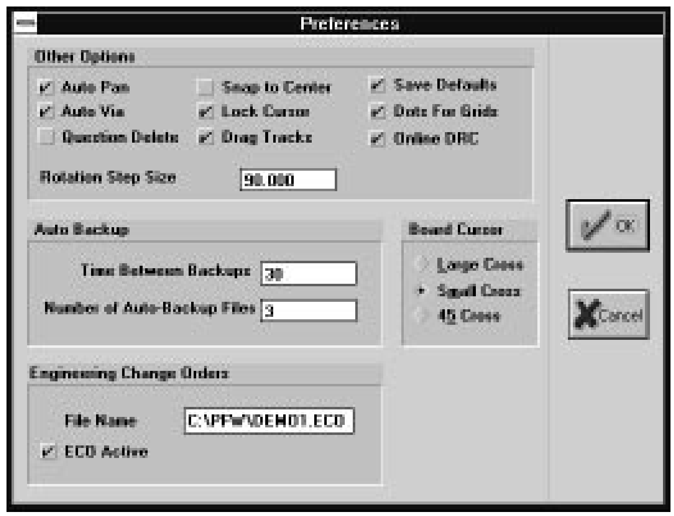
Transparent	You can make selected display layers Transparent (color and gray scale displays only) rather than solid color. This makes identification of overlapping objects easier when several layers are active at the same time. However, transparent colors will slow screen redraw, so you may wish to leave this option off, until needed (e.g. during final clean-up of the board).
Dithered	Dithered colors are created by mixing two solid colors. This supports graphics cards/drivers that support less than 256 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 all layers.
Current Layer On	Activates the Current layer. Leaves other layer selections undisturbed.

Preferences

Shortcut o, p

Summary Defines various system settings. These options affect all document windows and are stored between sessions.

Procedure First, choose a document window, then choose Options-Preferences.



Options include:

Auto Pan — When this option is enabled, the workspace window will pan automatically when using a Place or Move command or tool, when the cursor (or moving primitive) reaches the edge of the current workspace window.

Auto Via — This option places vias automatically when using the Edit-Place-Track command (or Track tool button). A via is placed each time the current layer is changed.

Question Delete	When enabled, prompts the user to confirm any deletion of primitives from the current window.
Snap to Center	If Snap to Center is enabled the cursor will jump to the center of a pad, track, etc., as it is moved. If you have the Snap to Center off, the cursor position will remain relative to the point where the selection was made. When you select a component with Snap to Center Option on, then the cursor will jump to the reference point of the component (generally the center pin 1 or the component center). If this option is off then when you select a component the cursor will remain in the same position relative to the component. If you select a text string with this option turned on then the cursor will jump to the bottom left hand corner of the text string. If you have this option off when you select the text string then the cursor will remain in the same position relative to the string.
Lock Cursor	Locked cursor mode links the “logical” cursor that can be positioned anywhere in the Windows display with the “physical” cursor that is displayed in the workspace as you make selections, etc. An unlocked cursor allows you to position the cursor in the workspace, press ENTER or LEFT MOUSE to update the coordinate, then use the mouse to access a menu command without “losing” the original cursor position (use Edit-Jump-

	New Location to return to the original position). When the cursor is locked, the coordinates are continuously updated as it is moved.
Drag Tracks	When enabled, this option maintains track to component pad connections when components are moved. As you move components, the tracks will drag (stretch) to maintain the connection. Within limits, orthogonality will be maintained – that is, tracks will keep their horizontal/ vertical orientation as far as possible. If you rotate the components or move the component beyond the reach of the orthogonality feature, some segments will rubberband to an "any angle" placement. You can use the Edit-Move-Break Track command (or press CTRL+SHIFT while clicking on track segments) to restore orthogonal placement.
Save Defaults	When enabled, user settings, preferences and defaults are written to a defaults file (PFW.PIF) when exiting the program.
Dots for Grids	When enabled, displays the visible grid as a series of dots. Visible grid colors are assigned using the Options-Layers command. The Current-Visible Grid (1 or 2) command is used to define the grid spacing.
Online DRC	Activates on-line Design Rules Check system, which signals any clearance violations while placing or moving objects in the document window. Primitives which violate minimum clearances will be flagged by a

	highlighted outline which is displayed on a special DRC Errors layer (Options-Layers command).
Rotation Step Size	Sets the rotation angle for the SPACEBAR shortcut, used to rotate items when they are placed or moved. Each time SPACEBAR is pressed, the selection or moved primitive rotates the designated number of degrees. Range is 00.001–360.000 degrees. Default is 90.000 degrees.
Auto Backup	Sets interval (in minutes) for generating an automatic backup of the active workfile. To disable this feature, type 0. The default interval is 30 minutes. Auto Backup works on the basis of a rotating file system. For example, if you have a file called TEST.PCB then the first backup will be saved as TEST.AB0. After the backup interval, a second backup, TEST.AB1, etc., up to the total number of files selected. The next backup would again be TEST.AB0, overwriting any previous version with that extension.
Board Cursor	Options include large cross, small cross (default) or 45 degree cross.
Engineering Change Orders	Specifies the filename for Engineering Change Order (.ECO) files. This file stores information about the current project and is used to update Advanced Schematic (version 2.0 or later) and Advanced PCB (version 2.0 or later) files. The ECO Active option enables/disables the ECO feature. When this option is enabled, several system events, such as opening a .PCB file, adding a component,

deleting a net, etc. are logged in the .ECO file. See also Netlist-Run ECO File.

Display

Shortcut o, d

Summary Defines display attributes for each type of primitive. These attributes affect all document windows.

Procedure First, choose a document window, then choose Options-Display. Choose between Final (solid rendering), Draft (outline rendering) and Hidden for each primitive type. These options effect display only, not printing or plotting.

Draft Track Threshold allows the user to set a minimum size for fully rendered track outlines in Draft mode. Tracks below this size will be displayed as single-pixel wide lines, which speeds re-draw of these draft mode tracks.

Pad display can include the net names and pad designators, if these options are enabled.

Track Mode-Any Angle

Shortcut o, t, a

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

Procedure First, choose a document window, then choose Options-Track Mode-Any Angle.

Track Mode-90/90 Line

Shortcut o, t, 9

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

Procedure First, choose a document window, then choose Options-Track Mode-90/90 Line.

Track Mode-45/90 Line

Shortcut o, t, 4

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

Procedure First, choose a document window, then choose Options-Track Mode-45/90 Line.

Track Mode-90 Arc/Line

Shortcut o, t, r

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

Procedure First, choose a document window, then choose Options-Track Mode-90 Arc/Line.

Tool Bar

Shortcut o, o

Summary Toggles the Tool Bar display on or off.

Procedure Choose Options-Tool Bar.

Status

Shortcut o, s

Summary Toggles the Status Line display on or off.

Procedure Choose Options-Status.

Scroll Bars

Shortcut o, c

Summary Toggles the Scroll Bars on or off.

Procedure First, choose a document window, then choose Options-Scroll Bars.

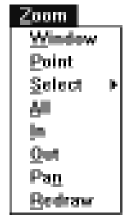
Toggle Units

Shortcut o, u or press q

Summary Toggles the current unit between imperial (mils) and metric (mm).

Procedure First, choose a document window, then choose Options-Toggle Units.

Zoom menu



Window

Shortcut z, w or 

Summary Resets the display window using a user-defined window.

Procedure First, choose a document window, then choose Zoom-Window.

See also Zoom-Point

Point

Shortcut z, p

Summary Resets the display window from a user-defined point.

Procedure First, choose a document window. Choose Zoom-Point.

See also Zoom-Window

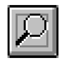
Select

Shortcut z, s

Summary Sets the display magnification of the current document window.

Procedure First, choose a document window. Choose Zoom-Select and Choose a magnification level or choose Other to enter another value between 1–400. 1 is the highest magnification (approximately 0.6 mils per screen pixel at 800 x 600 screen resolution).

All

Shortcut z, a 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 use this feature with a new document, it first must be saved and loaded from a file.

Procedure First, choose a document window, then choose Zoom-All.

See also File-Save, File-Save As, File-Open

In

Shortcut z, i or PGUP

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

Procedure First, choose a document window, then choose Zoom-In.

Out

Shortcut z, o or pgdn

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

Procedure First, choose a document window, then choose Zoom-Out.

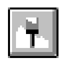
Pan

Shortcut z, n or HOME

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

Procedure First, choose a document window, then choose Zoom-Pan.

Redraw

Shortcut z, r or END or 

Summary Redraws the current document window without changing the zoom level.

Procedure First, choose a document window, then choose Zoom-Redraw.

Info menu

System Status

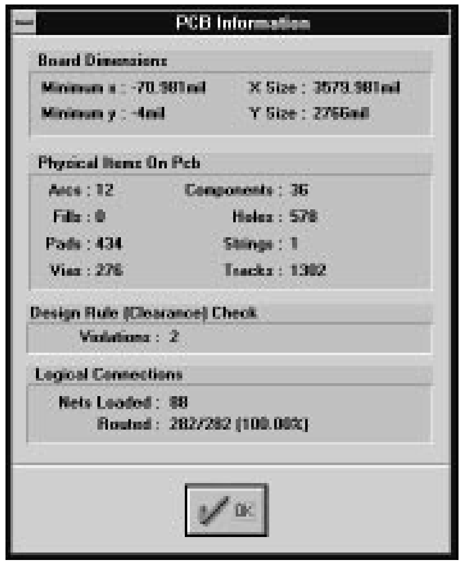
Shortcut i, s

Summary Displays disk and memory status including disk space, memory, date and time settings. Free memory refers to the amount of virtual memory available – the sum of your physical memory left and your disk space available on the drive where your swap file is setup.

Procedure First, choose a document window, then choose Info-System Status.

Board Status

Shortcut i, b



Summary Displays board statistics: dimensions, primitive counts, current design rule violations, number of nets and routing completion.

Procedure First, choose a window, then choose Info-Board Status.

Components On PCB

Shortcut i, c

Summary Lists all placed components in the current document window, listed by designator and comment, if any.

Procedure First, 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.

Selected Pins

Shortcut i, e

Summary Lists the component label and pin designators for all selected pins in the current document window. Entries are sorted by component designator then pin designator (e.g. U1–16).

Procedure First, 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.

Nets

Shortcut i, n

Summary Lists all currently loaded nets in the current document window.

Procedure First, 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.

Measure Distance

Shortcut i, m

Summary Displays the distance between any two points designated by the user.

Procedure First, choose a document window. Choose Info-Measure Distance. Status line will prompt to click at the first location, then another location anywhere in the same board window. The measured distance will be displayed using the current units (mils or mm).

Length of Selection

Shortcut i, l

Summary Displays the total physical connection length of selected tracks within the current document window. Arcs will be included in the calculation (if selected), however the end point diagonal distance will be calculated, not the chord.

Procedure First, 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

Power Planes-Plane 1, 2, 3, 4

Shortcut i, p, 1 (or 2 or 3 or 4)

Summary Displays the designators of all pins with direct or thermal relief connections on Plane 1 (or 2, 3, 4) within the current document window.

Procedure First, choose a document window, then choose Info-Power Planes Plane 1 (or 2, 3, 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 menu

Tile

Shortcut w, t

Summary Tiles all non-minimized document windows.

Procedure Choose Window-Tile.

Cascade

Shortcut w, c

Summary Cascasdes all document windows .

Procedure Choose Window-Cascade.

Arrange Icons

Shortcut w, i

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

Procedure Choose Window-Arrange Icons.

Close All

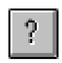
Shortcut w, a

Summary Closes all Document windows and icons.

Procedure Choose Window-Close All.

Help menu

Index

Shortcut h, i or F1 or 

Summary Displays a list of help topics, that include all areas of the help system.

Procedure Choose Help-Index.

Using Help

Shortcut h, h

Summary Displays information about using the help system.

Procedure Choose Help-Using Help.

Basic Concepts

Shortcut h, b

Summary Displays an index of basic operation topics, that include detailed information about Advanced PCB techniques.

Procedure Choose Help-Basic Concepts.

Commands

Shortcut h, c

Summary Displays the Advanced PCB commands index, that includes detailed information about each command.

Procedure Choose Help-Commands.

Printing

Shortcut h, p

Summary Displays an index of printing topics, that include detailed information about printing and plotting.

Procedure Choose Help-Printing.

Managing Files

Shortcut h, f

Summary Displays an index of file management topics, that include detailed information about printing and plotting.

Procedure Choose Help-Managing Files

Screen Regions

Shortcut h, s

Summary Displays an index of screen area topics, that include detailed information about using the Advanced PCB interface.

Procedure Choose Help-Screen Regions

Reference

Shortcut h, r

Summary Displays an index of reference topics.

Procedure Choose Help-Reference.

About

Shortcut h, a

Summary Displays the software version number. Click the Set Access Codes button to activate additional Advanced PCB (or other Protel Design System) options.

Procedure Choose Help-About.

Note Protel Design System access codes for each software module are stored in files identified by the extension .CDE, located in the Windows director, either on a server (if running the software over a network) or on an individual machine. The .CDE file is named with the Hardlock (protection device) serial number (e.g. S1000899.CDE).

Support for multiple .CDE files allows the use of different hardlock licenses on the same machine.

Multiple Hardlocks and multiple CDE files are allowed. All .CDE files and Hardlocks are scanned when the software is run. The Set Access Codes dialog box lists all Hardlocks and access codes found on start-up. All enabled modules are listed in the right-hand list box. An enabled module is only listed once, irrespective of the number of valid license codes recognized by the system. Options and licensing arrangements for each Protel Design System Module may vary from country-to-country.

See the *Protel Design System Environment Guide* or contact your Protel representative for further information.