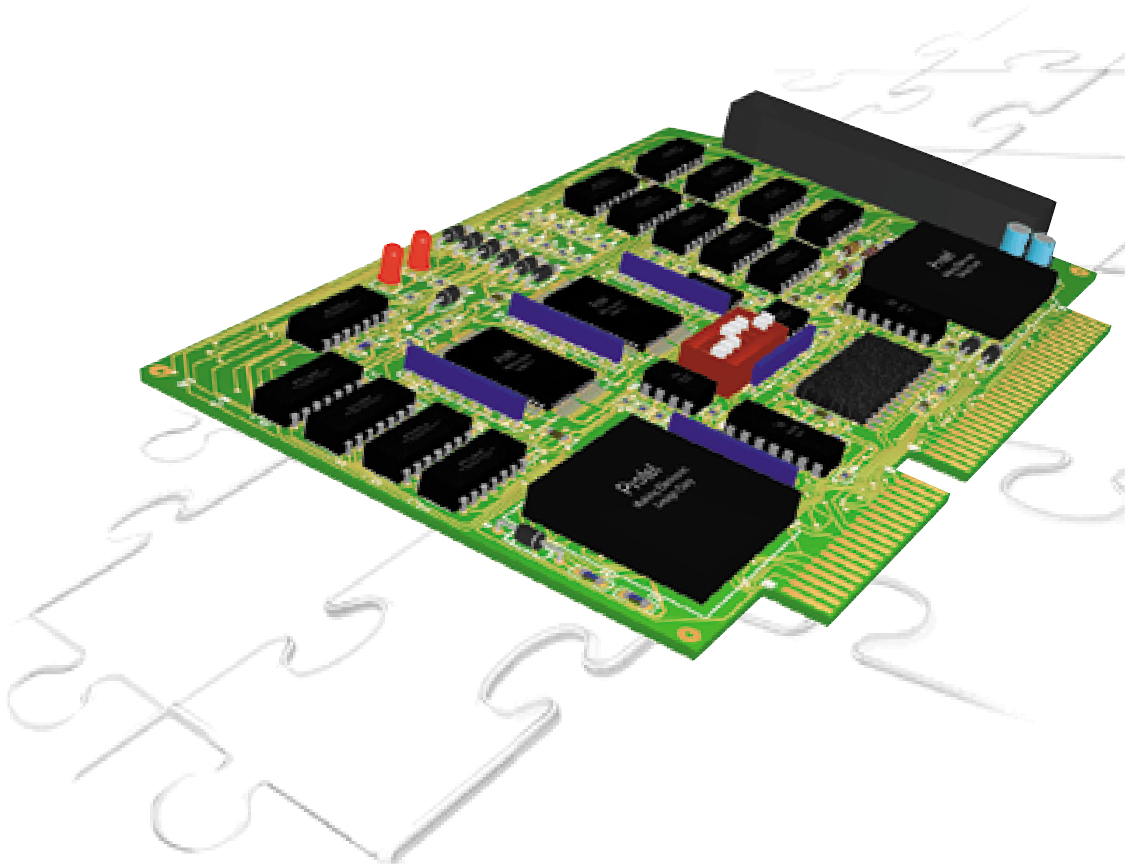
Protel
Making Electronic Design Easy™

Protel 99 SE

Designer's Handbook Supplement



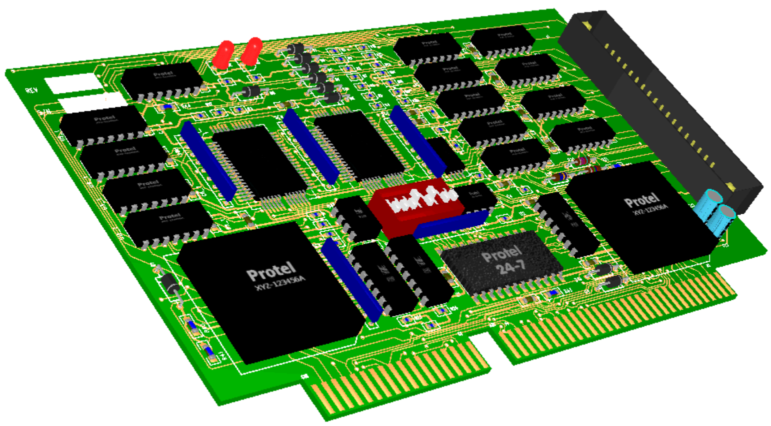
Runs on Windows NT/95/98

Increase your PCB Design Productivity with Protel 99 SE

Welcome to the Protel 99 SE Designer's Handbook Supplement, your comprehensive guide to exploring and using the new and enhanced features included in Protel 99 SE. The information in this supplement will help you get the most from Protel 99 SE's advanced board design features, and complements the comprehensive information provided in the Protel 99 Designer's Handbook.

Protel 99 SE is the latest version of Protel's integrated board-level design system for the Windows NT/98/95 operating system. It builds on the foundation of Protel's unique Design Explorer platform, introduced with the release of Protel 99, by adding a host of new and enhanced features aimed at streamlining the board design process.

Protel 99 SE's Design Explorer integration platform has been optimized to give fast application and design document opening, more responsive performance and more efficient memory usage. You now have a choice of design data storage methods – save your integrated design in a single Access database, or as stand-alone files and folders using the simplicity of the Windows File System. With either storage method you have available the full power and convenience of Protel 99 SE's design management and integration features.



From design entry through to manufacturing output creation, Protel 99 SE gives you greater design flexibility and power. Capture your design faster and more accurately with Protel 99 SE's enhanced schematic editor, that now features direct on-sheet text editing, sheet-by-sheet and positional annotation, automatic component class creation based on source schematic sheets, plus a host of time-saving editing and interface enhancements.

Create your board from 32 signal layers, 16 internal plane layers, and 16 mechanical layers, with fully-definable layer stackup and drill layer pairing. With enhanced power plane connectivity, new design rules and rule scopes, and import/export of design rule sets, Protel 99 SE's PCB editor gives you unparalleled versatility in board design definition.

New and enhanced interactive component placement tools will slash design layout time. Protel 99 SE supports on-board graphical creation and editing of placement rooms, dynamic real-time optimization of connection lines during component moves, and the ability to group components for fast placement of component blocks.

New PCB design features in Protel 99 SE include a powerful PCB print management system, an advanced 3D PCB renderer and viewer, and an invaluable CAM Manager that gives you "one click" output generation.

The above enhancements and features are just a taste of the many ways in which Protel 99 SE makes your desktop a more productive board design environment. Please explore this supplement and see just how much easier design can be with Protel 99 SE.

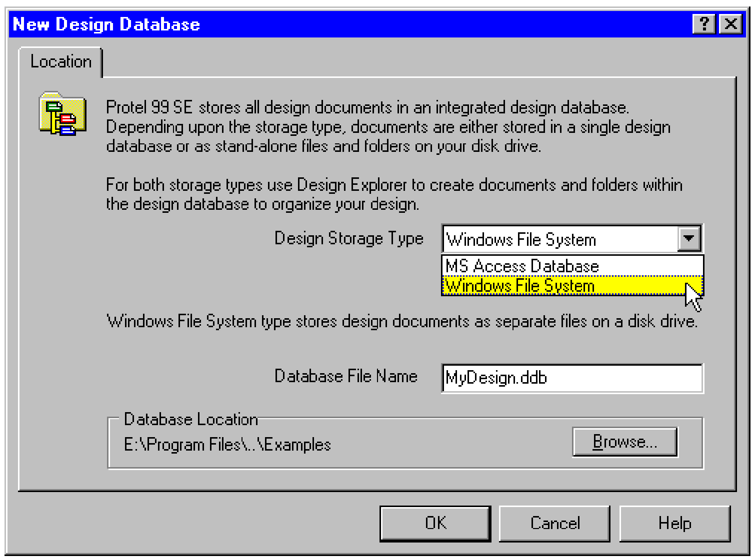
Design Explorer

The Design Explorer is the name given to the Protel 99 SE design environment. When you select Protel 99 from the Windows Start menu, the Design Explorer opens. The Design Explorer is the interface to your designs, and the various design tools that you use to create your designs.

Windows File System Storage Option

Protel 99 SE includes a new document storage option that stores design documents directly on a disk drive. The **New Design Database** dialog includes a new Design Storage Type option, where you specify if the design documents will be stored in a single integrated Microsoft Access database, or if they will be stored directly on a disk drive.

If the Storage Type is set to MS Access Database then all design documents are stored in a single database. If the Storage Type is set to Windows File System then all design documents are stored directly on the disk drive in the location specified at the bottom of the dialog.



Select the document storage type when you create a new design

Regardless of the storage system that is chosen, the way you work in the Design Explorer is exactly the same. If your design uses the Windows File System Storage Type you still open the design first – then open the schematic, PCB, or other design documents.

New documents are created in the same way for both storage types, select **File » New** to create a new document. Note that you can not move documents into a Windows File System database with the Windows File Explorer, they must be imported into a database before they can be opened. You can import a number of files by importing a folder, or by dragging from the Windows File Explorer directly into an open design.

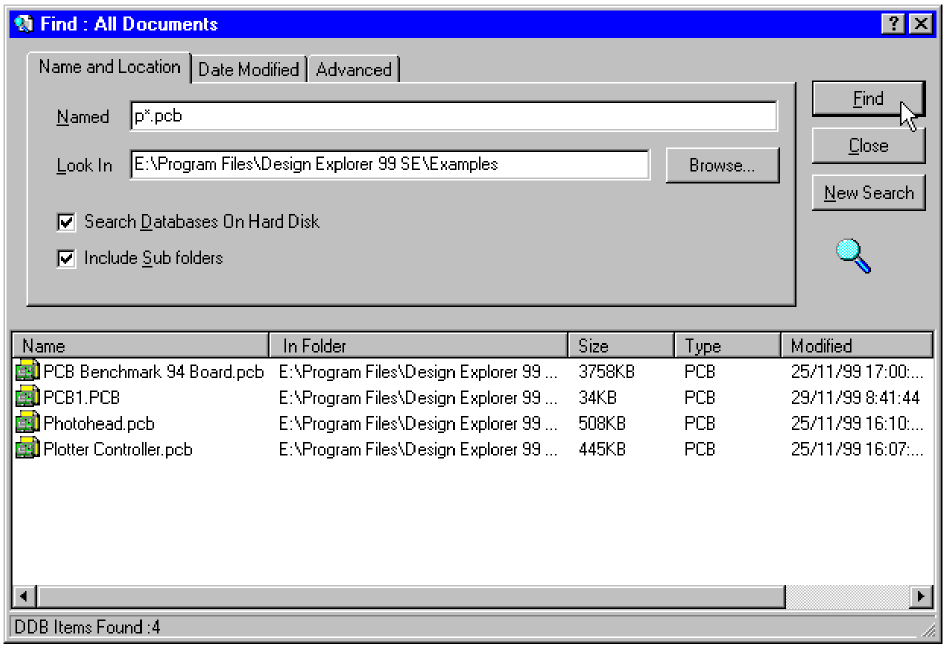
Designs that use the Windows File System storage do not support any of the DesignTeam features, such as document access control. Other design integration features, like synchronization and background document opening when printing and netlisting, are supported.

Emailing from the Design Explorer

Design documents can be emailed directly from the Design Explorer. Select the documents in the Design Explorer that you want to email, then select **File » Send by Mail** from the menus.

New Document Find Feature

Protel 99 SE includes a new Document Finder, which operates like the File Find Feature in the Microsoft Windows Explorer. The document finder can be used to search for documents in designs that are currently open in the Design Explorer, or it can be used to search for documents in designs that are on the hard disk. Select **File » Find Files** from the Design Explorer menus (when a folder is the active window) to pop up the **Find All Documents** dialog.



Use the Document Finder to quickly locate a design document

Improved Performance

Protel 99 SE has been optimized for faster performance, with reduced Design Explorer startup time, as well as faster opening of designs and design documents. Network performance has been enhanced with reduced inter-design station communications and CPU usage.

Floating Licenses

Protel 99 SE supports floating licenses. This system allows you to install Protel 99 SE on as many PCs as you like – Protel 99 SE automatically monitors how many copies are running and displays a warning message when there are too many copies running at the same time.

If your network includes PCs that must have a single-user license permanently allocated disable the Broadcast and Receive Floating Access Codes options at the bottom **Security Locks** dialog.

Enhanced Status Bar

The message zones on the Design Explorer Status Bar can be resized – position the cursor over the arrow symbols on the status bar, when the cursor changes to a double-headed arrow click and drag to resize a message zone.

PCB Design

Protel 99 SE includes a large number of productivity enhancements for the PCB designer – increased design layers and a new layer stack manager, extended and enhanced design rules, a new printing engine with sophisticated print previewing and printout manipulation, a new manufacturing output file management system, a powerful 3D visualization tool, enhanced placement features, new autorouting cleaning passes, new import and export capabilities, enhanced library editing and management features, and numerous workspace selection and editing improvements.

The PCB Editor in Protel 99 SE uses a new file format (PCB4.0). Refer to the *File Importers and Exporters* topic later in this section of the supplement for details on exporting a PCB file to an older version file format.

PCB Layer and Power Plane Enhancements

A PCB is fabricated as a series of layers, including copper electrical layers, insulation layers, protective masking layers, and text and graphic overlay layers.

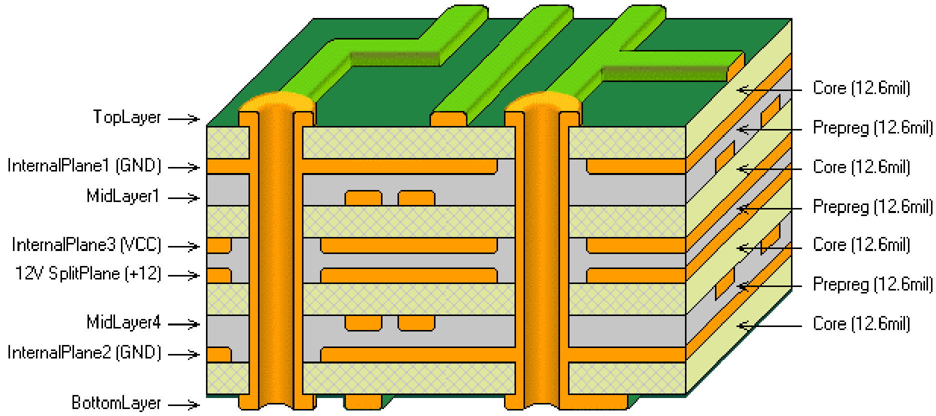
There are 2 types of electrical layers – signal layers, which contain the signal interconnect paths, and power planes, which are layers of unbroken copper used to distribute current to power the components. In Protel 99 SE these signal and plane layers are made available in the workspace by defining the layer stack-up.

Defining the Layer Stack

To define the layer stack select **Design » Layer Stack Manager** from the menus. The **Layer Stack Manager** dialog will appear. In the center of the dialog there is a image that shows the current layer stack, the default is for a double-sided board. More layers can be added to the design by clicking on the Add Layer and Add Plane buttons. Each new layer is added below the currently selected layer in the layer stack.

The Menu button at the bottom of the dialog also includes a number of pre-packed example layer stacks. Note that these example layer stacks are not fixed, you can start with one of these and easily modify it. Once the required layers have been added, use the Move Up and Move Down buttons to configure the layer stack. New layers can be added at any point in the design process.

There are a total of 32 signal layers available (top layer, bottom layer, and 30 mid-layers) and 16 plane layers.



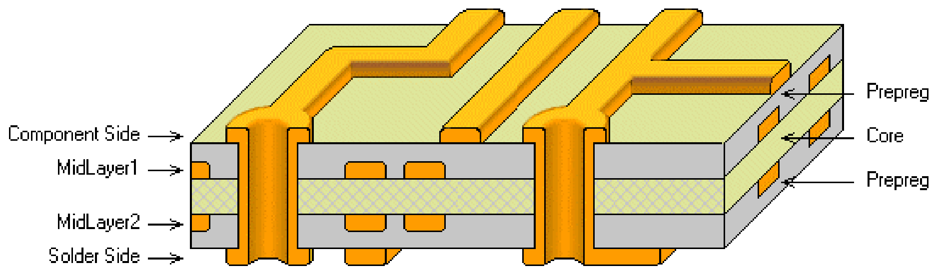
Layer stack for an eight layer board with 4 signal layers and 4 plane layers

Selecting the Layer Stack-up Style

As well as the electrical layers, the stack-up includes the non-electrical insulation layers. There are typically 2 kinds of insulation used in the fabrication of a PCB, usually referred to as *core* and *prepreg*. What are these insulators and how are they used?

Consider the typical fabrication process for a 4 layer PCB. A 4 layer PCB normally starts out as a piece of insulating core (usually a fiberglass material) which has thin layers of copper laminated on either side – in fact it is simply a thin double-sided PCB.

These copper layers are etched to create the signal tracks. Once this inner 'slice' is ready, a layer of prepreg insulation is applied to either side, then a layer of copper foil is applied to the outside of these prepreg layers. This layered structure is then laminated (bonded together) under heat and pressure, causing the prepreg layers to soften slightly and bond the various layers together. If the finished board were sliced so you could see inside, it would look like the figure below.



A cut-away of a 4 layer PCB showing the arrangement of the insulation layers

The stack-up style refers to the order of the insulation layers through the layer stack. Three default stack-up styles are supported – layer-pairs, internal layer-pairs, and build up. Changing the layer stack-up style changes the way that the core and prepreg layers are distributed through the layer stack.

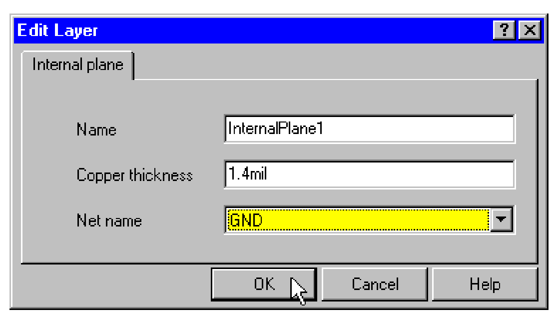
Select the preferred stack-up style at the top left of the **Layer Stack Manager** dialog. The stack-up style is only important if you plan to use blind and buried vias, and for signal integrity analysis. If you are not using blind and buried vias, or planning on performing a signal integrity analysis you can ignore this option. If you are planning to use blind and buried vias you must consult with your PCB manufacturer to ensure that that they can fabricate the design, and that the correct stack-up style is selected.

Setting up the Layer Properties

The properties of each layer are defined by double-clicking on the layer name in the **Layer Stack Manager** dialog (or on the layer in the stack-up image).

Signal layers properties that can be defined include the layer name and the copper thickness. The copper thickness setting is used for signal integrity analysis.

Plane layer properties that can be defined include the layer name, the copper thickness, and the net name. The net name is selected from the list of currently available nets. All thru-hole pads and vias assigned to this net are automatically connected to this plane, in accordance with the appropriate Power Plane Connection Style design rule (select **Design » Rules** from the menus to edit the rules).



Layer properties dialog for an internal plane layer

If you have not transferred the schematic design into the PCB workspace no nets will be available – if this is the case, leave the net name unassigned until the design has been transferred.

Core and prepreg properties that can be defined include the material, the thickness, and the dielectric constant. These settings are used during signal integrity analysis, leave them at their defaults unless you have a specific need to change them.

Setting up the Drill-Pairs

The last step in defining the layer stack-up is to specify the drill-pairs. The term drill-pairs refers to the 2 layers that a drilling operation starts from, and stops at. Unless the board includes blind and buried vias only one drill-pair is required, comprising the Top and Bottom layers. This drill-pair is on by default and can not be deleted or modified.

Drill-pairs are defined in the **Drill-Pair Manager** dialog, click on the Drill-Pairs button in the **Layer Stack Manager** dialog to display this dialog.

If the design includes blind and buried layers then the drill pairs must be defined to suit the layer stack-up style. This should be done in consultation with your board manufacturer to ensure that your design matches their fabrication technology. For more information on blind and buried vias refer to the *Vias* topic in the *PCB Design Objects* chapter of the Designer's Handbook.

Enabling and Naming the Mechanical Layers

Mechanical layers are general purpose non-electrical layers that can be used for any task your design requires. Some examples of what they can be used for include: defining the board outline, dimensioning, assembly information, NC routing details, special mask requirements (such as glue masks or peelable solder masks), title block and border details, and so on.

To enable the mechanical layers select **Design » Mechanical Layers** from the PCB Editor menus. Each mechanical layer can be named. Like other layers, a mechanical layer can only be disabled if there are no design objects on it, if a layer has objects on it the Enable check box will be grayed out. Note that you can also selectively display mechanical layers when single layer mode is enabled (single layer mode hides all layers except the current layer – press the SHIFT+S shortcut keys to toggle single layer mode on and off).

Controlling the Display of Layers

The display of enabled layers is controlled in the **Document Options** dialog, select **Design » Options** to display this dialog. As well as the user-defined signal, plane and mechanical layers, this dialog gives you control over the display of all the other layers available in the PCB Editor workspace.

Layer colors are defined in the Colors Tab of the **Preferences** dialog, select **Tools » Preferences** to display this dialog.

Layer Specific Keepout Objects

Layer-specific keepout objects can be placed on electrical layers to act as a routing barrier. Select **Place » Keepout** from the PCB Editor menus to place a keepout track, fill or arc (or use the K shortcut key to pop up the **Keepout** sub-menu). Keepout objects are rendered in the same color as the layer they are placed on, with a border drawn in the current keepout color. Keepout objects are not included in Gerber generation, they can be included in printouts by enabling the Print Keepout Objects option in the Power Print **Preferences** dialog.

PCB Design Rules

Design Rules are the basis of design specification and control in the PCB Editor. The enhancements to the rule system include: new rules, new rule scopes, the ability to disable individual rules, importing and exporting of rule sets, new rule reports, and the ability to interrogate any design object on the board to establish what rules apply to that object. The PCB Editor panel has also been enhanced to include rules, making it easy to examine the rules and what each rule applies to.

This topic should be read in conjunction with the chapter **Specifying the PCB Design Requirements** in the Protel 99 Designer's Handbook.

New and Enhanced Routing Rules

Width Constraint

This rule now includes Minimum, Maximum, and Preferred settings. The Minimum and Maximum settings are obeyed by on-line and batch DRC. The Preferred setting is obeyed during manual and auto routing, and can be changed on-the-fly during manual routing by pressing the TAB shortcut key.

Routing Via Style

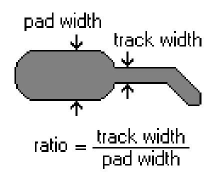
This rule now includes Minimum, Maximum, and Preferred settings. The Minimum and Maximum settings are obeyed by on-line and batch DRC. The Preferred settings are obeyed during manual routing, and (for the Board scope rule) during autorouting. The Preferred settings can be changed on-the-fly during manual routing by pressing the TAB key.

SMD To Plane Constraint

This rule specifies the maximum total route length allowed from the center of an SMD pad to the center of the power plane connection pad/via. This rule is obeyed by on-line and batch DRC.

SMD Neck-Down Constraint

This rule specifies the maximum allowable ratio of the track width to the SMD pad width, expressed as a percentage. Tracks that result in a ratio larger than this amount are flagged as a violation. This rule is obeyed by on-line and batch DRC.



New Manufacturing Rules

Hole Size Constraint

This rule specifies the maximum and minimum allowable hole size, expressed either as exact numeric values, or as a percentage of the pad size. This rule is obeyed by on-line and batch DRC.

Layer-Pairs

This rule checks that vias and pads only connect between the defined drill pairs. To define the allowable layer-pairs select **Design » Layer Stack Manager**, then click on the Drill Pairs button to display the **Drill-Pair Manager** dialog. This rule is obeyed by the auto-via feature during manual routing, and by on-line and batch DRC.

Testpoint Style

This rule specifies the allowable physical parameters of pads and vias that are flagged as testpoints. A pad or via is defined as a testpoint when it has one or both of its Testpoint attributes enabled. All rule attributes apart from the preferred size settings are checked by the online and batch DRC.

The rule is also used by the **Find and set Testpoints** feature (**Tools** menu), which searches the PCB for a suitable pad/via in each net that complies with this rule, and then enables its testpoint attribute.

The autorouter also uses this rule if the Add Testpoint option is enabled in the **Autorouter Setup** dialog. The autorouter places round pads of the size defined in the Preferred fields of the **Testpoint Style Rule** dialog, on the layer specified in the Allowed Side settings.

The Allowed Side settings are used by the testpoint finder and the autorouter in the following pre-defined order: Bottom (SMD Pads), Top (SMD pads), Bottom Thru-hole, and Top Thru-hole, depending on which are enabled of course.

Testpoint Usage

This rule specifies which nets are required to have a testpoint. Note that it can also be used to specify nets that must not have a testpoint. This rule is obeyed by the Find Testpoint feature and by the Autorouter during testpoint placement, and by the on-line and batch DRC.

Use the DRC report to identify nets that should have a testpoint, but do not. Use the Testpoint report feature in the CAM Manager to identify the location of all testpoints (select **File » CAM Manager** from the menus to configure this).

New and Enhanced Placement Rules

Component Clearance Constraint

This rule specifies the minimum clearance allowed between components. The Gap setting in the rule is the distance allowed between components targeted by the rule scope. This rule is obeyed by on-line and batch DRC, and by the cluster-based autoplacer.

Room Definition

This rule is used to define the location of a placement room, and the set of components that must be in that room. Read the *PCB Placement Enhancements* section later in this document for more details on how to use Rooms.

Other Rules Enhancements

Unconnected Pin Constraint

This rule detects pins that have no net assigned and no connecting tracks. This rule is obeyed by on-line and batch DRC.

New and enhanced Rule Scopes

The scope of each rule defines exactly what that rule must target. To enhance the power and flexibility of the rule system a number of new rule scopes have been added.

Pad Specification

This scope uses a pad specification to define what the design rule is to apply to. An example of when this scope could be used is to target particular pads to define their power plane connection style.

Via Specification

This scope uses a via specification to define what the design rule is to apply to. An example of when this scope could be used is with the solder mask rule – use it when you need to tent a specific-sized via on one side of the board only.

Footprint

This scope allows a design rule to target all components that use the specified footprint.

Footprint Pad

This scope allows a design rule to target a specific pad (or pads if wildcards are used) in the specified footprint.

Pad Class

This scope allows a design rule to target the set of pads included in the class. Pad classes are created by selecting **Design » Classes** from the menus. Refer to the *PCB Selection* topic later in this supplement for information on new ways of quickly creating classes.

Object Kind = Polygon

This scope now includes polygon. Use this to pour polygons with a different clearance. Use it in combination with the compound rule scope feature to target polygons on a specific net or a specific layer.

Other Enhancements to Rule Scopes

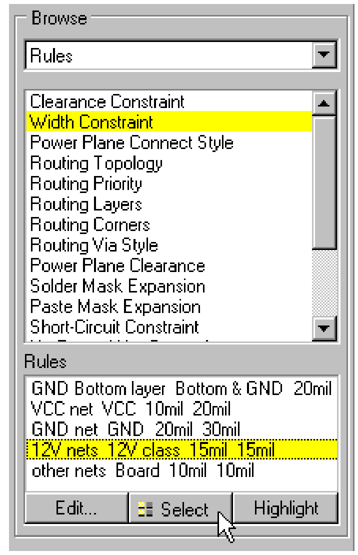
A new **Mask** field has been added to speed the process of selecting from a list of names in the individual design rule dialogs. The Mask field appears when the Filter Kind is set to Component, Net, Pad or Footprint. The mask is dynamic, the list is filtered as you type.

A new Edit Class button appears in the **Design Rule** dialog when a rule scope is set to Pad Class, Net Class, Component Class or From-To Class, making it easy to modify or create a class while setting up a design rule.

Other Enhancements to the Rule System

Working with Rules from the PCB Panel

Rules have been added to the Browse modes available in the PCB Editor Panel. When Rules are selected the top region of the panel displays all the types of rules, and the lower region displays all the rules of the selected type that are currently defined. Use the buttons on the panel to Edit a rule, Select all the objects that this rule applies to, or Highlight all the objects that this rule applies to.



Set the Browse mode to Rules to easily examine and edit the rules

Naming Rules

Rules can now be named in the individual design rule dialogs. When a new rule is created it is assigned a default name. The rule name is displayed in the panel when the Browse mode is set to Rules.

Disabling Rules

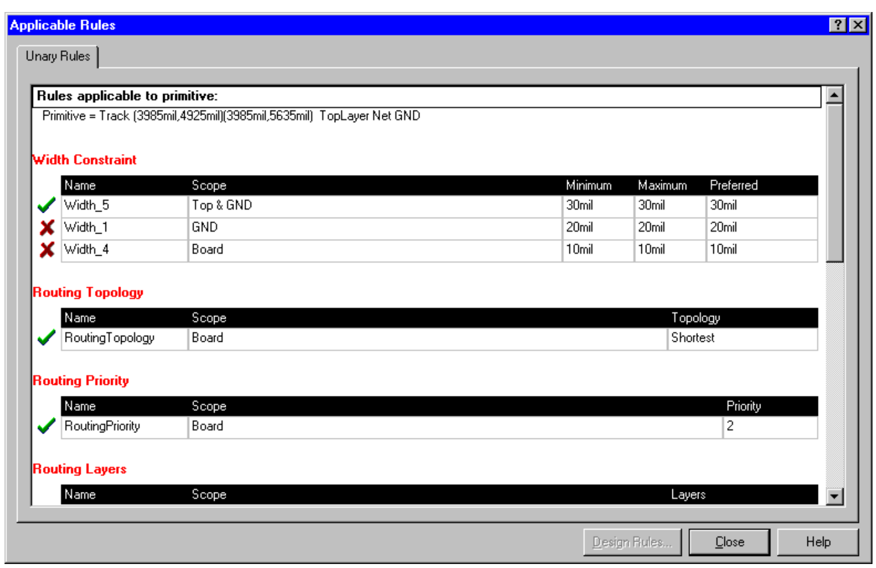
Rules can be individually disabled. Disabling a rule has the same effect as deleting the rule in terms of how it is handled by the on-line and batch DRC routines. Disabled rules are displayed in the panel with a line drawn through their name.

Importing and Exporting Rule Sets

Rule sets can be exported and imported, allowing you to store and retrieve favorite rule sets. To import or export a rule set click on the Menu button at the bottom of the **Design Rules** dialog. When importing rules you can either Overwrite or Add them by clicking the appropriate button in the rule **Import Options** dialog. Overwrite occurs when a rule with the same rule name is encountered.

Checking which Rules Apply

There are 2 new applicable rules options in the right-click floating menu. After selecting either **Applicable Unary Rule** or **Applicable Binary Rule** from the floating menu you will be prompted on the Status bar to select either one primitive (**Applicable Unary Rule**) or 2 primitives (**Applicable Binary Rule**).



The Applicable Rules dialog details all rules that apply to the primitive(s)

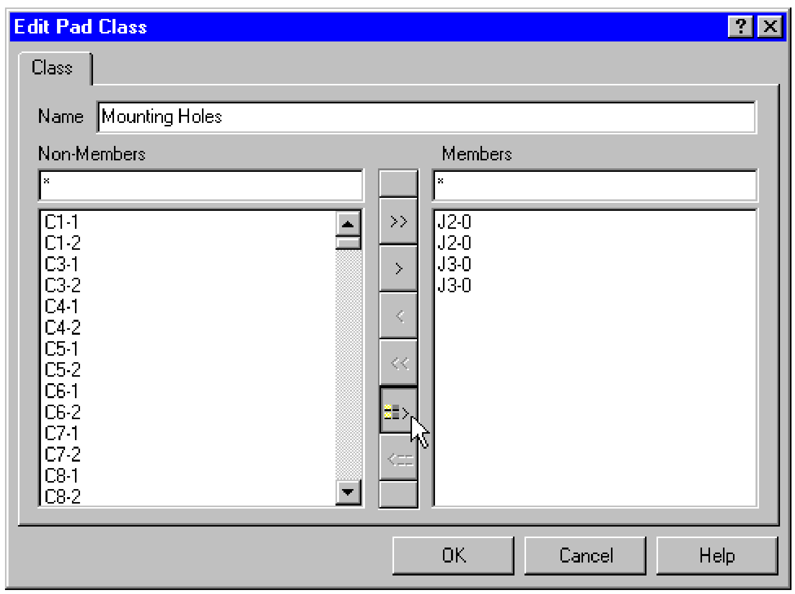
The **Applicable Rules** dialog then appears, displaying all design rules that apply to this primitive(s). Note that all current design rules that could be applied to the selected primitive(s) are analyzed and listed in the **Applicable Rules** dialog. Each rule that is listed in the dialog will have either a tick or a cross next to it. A tick indicates that this is the highest priority rule out of all applicable rules, and is the rule being applied. Lower priority rules of the same kind are listed with a cross next to them, indicating that they are applicable but as they are not the highest priority rule they are not currently applied.

On a more global level you can examine what objects each rule applies to by setting the browse mode in the Panel to Rules, and then using the Highlight and Select buttons to show which objects any rule applies to.

When you want to know why an object is flagged as being in violation right click on it and select **Violations** from the floating menu. The **Violation Inspector** dialog will appear, detailing which design rule(s) this object is not complying with.

New Object Class Added

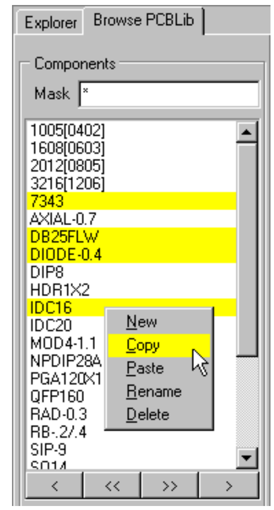
Classes of pads can now be created in the **Object Classes** dialog. Note the new button in the **Edit Pad Class** dialog that allows a class to be created from the currently selected pads. There is also a new Select button in the **Object Classes** dialog that works the other way, click this to select the objects on the board that belong to a class.



Use the new 'take over selected objects' button to quickly create a pad class of the selected pads

PCB Library Editor and Component Enhancements

Working with components, both in the Library Editor and in the PCB Editor has been enhanced. The Library Editor has multiple Undo/Redo, complete support for copying and pasting components, and a new component rule checker. Components can also be modified directly on the board, including adding and removing primitives.



Right-click in the Panel to copy the selected components

PCB Library Enhancements

Undo/Redo

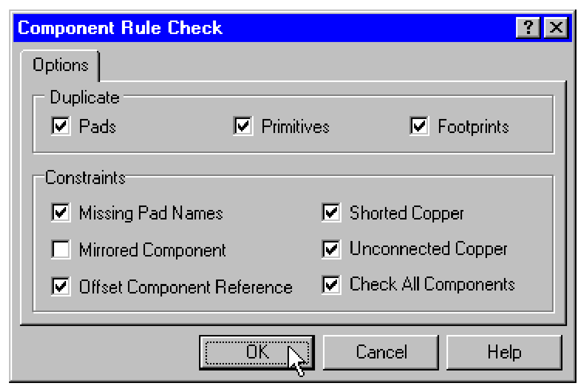
The Library Editor supports multiple Undo/Redo, for each modified component.

Copying Components

Components can be copied and pasted between libraries, from a library to a PCB, and from a PCB to a library. To copy between libraries, or from the library to a PCB, select the component(s) in the Library Editor panel using the standard Windows selection keys (left-click, SHIFT+click and CTRL+click). Once the components have been selected click the right mouse button to pop up the floating menu and select **Copy**. Change to the target library, right-click in the Library Editor panel, and select **Paste** to add them to the target library. If you are pasting directly onto a board select **Edit » Paste** from the PCB Editor menus.

Component Rule Checker

Select **Reports » Component Rule Check** to pop up the **Component Rule Check** dialog. The Component Rule Checker tests for duplicate primitives, missing pad designators, floating copper and inappropriate component reference.



Check PCB footprints with the Component Rule Check

Adding, Removing and Browsing Libraries

Component libraries can be added and removed individually from the PCB Editor library list. When you add a design that includes PCB libraries in the **PCB Libraries** dialog, all libraries in the dialog are listed at the bottom of the dialog. Use the Remove button to remove a library from the list.

The **Browse Libraries** dialog includes a Status bar at the bottom of the dialog. The Status bar reports the component X and Y size, and the properties of the most common pad.

Component Enhancements

Modifying a Component in the PCB Workspace

Component primitives can be added, modified and deleted from a component in the PCB workspace. To modify component primitives on the board you must first unlock them – to do this double-click on the component and disable the Lock Prims option. Once you close the dialog you can edit, and delete the existing primitives.

To add new primitives to a component first disable the Lock Prims option as before. Now place the new primitives as required, select the new primitives (but not the existing component primitives), then select **Tools » Convert » Add Selected Primitives to Component** from the menus. You will be prompted to click on the component that you want to add the selected primitives to – when you click on the component the new primitives are added and then deselected. Don't forget to re-enable the Lock Prims option when you have finished modifying the component. Note that these changes only affect this component, they do not affect the component footprint in the library.

Polygon primitives can also be included in a component footprint. To include polygon primitives in a component footprint the polygon must be added in the PCB workspace. The method of adding primitives to a component is described above. There is an extra step in adding polygon primitives to a component – after placing the polygon and selecting it (shortcut: SHIFT+click), select **Tools » Convert » Explode Polygon to Free Primitives** from the menus and click on the polygon to explode it. Now add these polygon primitives as you would add any new primitive to a component.

Including Routing in Component Footprints

Library components can include routing primitives. If your components include routing primitives there is a new option that automatically updates the net name of these primitives as the netlist is transferred from the schematic to the PCB. Enable the Assign Net to Connected Copper option in the **Update Design** dialog when you select **Design » Update PCB** from the Schematic Editor menus.

Net names can also be applied to routing primitives that are built into components at any time by selecting **Design » Netlist Manager** from the PCB Editor menus. Select the **Update Free Primitives from Component Pads** option from the Menu button at the bottom of the dialog to reapply the pad net names to all connected copper.

Component Designators and Pad Designators

Duplicate component pad designators are supported. Components can be designed with multiple pads with the same pad designator. The PCB netlist analyzer detects the presence of multiple pads and calculates the connection lines accordingly.

Component designator and comments strings can be changed during global editing, including partial string substitutions. For example, to globally change all IC designators from U1, U2, etc to IC1, IC2, etc, double-click to edit one of the ICs. In the **Component** dialog change its designator from U to IC, then click the global button to extend the dialog. In the Attributes to Match By column enter U* in the Designator field (meaning – find all components whose designator starts with the U character). In the Copy Attributes column set the Designator field to {U=IC} (meaning – for all matched components change any designators that start with the letter U to start with the letters IC). Set the Change Scope at the bottom of the dialog to All Primitives (meaning – include component primitives in this change), then click the OK button to carry out the change.

Changing components from one side of the board to the other flips pad stacks as well. If a component pad has the Pad Stack option enabled and the Top Pad settings are different from the Bottom Pad settings, these 2 settings are automatically swapped as the component is flipped from the top to the bottom of the board.

Component designators can be up to 255 characters in length and component pad designators can be up to 20 characters in length.

PCB Placement

The design has been transferred from the schematic, the board boundary and layer stack defined – you are now ready to start placing the components. Good component placement is fundamental to a well designed board. During placement you must consider many issues – including mechanical requirements, thermal considerations, as well as the signal integrity and routability.

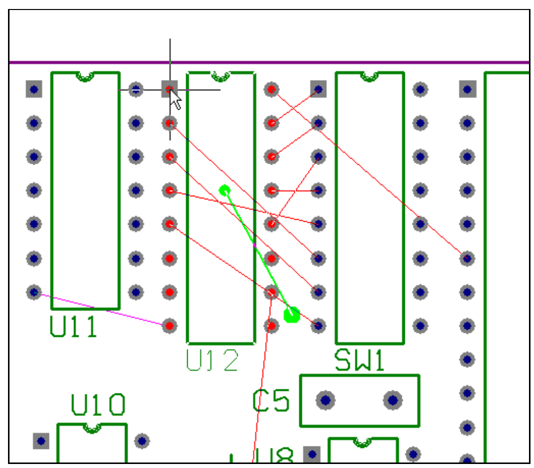
Protel 99 SE includes a number of features to help in the placement process, including schematic to PCB selection, automatic PCB component class creation, dynamic reconnect with smart connection line display, on-line component clearance checking, component unions, placement rooms and a number of placing, aligning and spacing tools.

Dynamic Reconnect with Smart Connection Line Display

Your basic guide to selecting suitable positions for each component are the connection lines from that component to the other components on the board. Ideally each component is placed to minimize the overall length of the connection lines, helping minimize the finished routing paths.

However, on a dense board it can be difficult to make sense of the maze of connection lines, which are often referred to as 'the ratsnest'. The other important fact to be aware of is that as you move a component around the board, the connection lines may no longer accurately reflect the connectivity, it could be that the connection lines could connect to other pins on the same net that are closer to the new component location.

Protel 99 SE includes a dynamic reconnect feature with smart connection-line display. When you move a component, a group of selected components or a union of components, all connection lines are temporarily hidden except for those that connect from a moving component to a component on the board. These nets are analyzed and reconnected as you move the components across the board, making it easy to select the appropriate placement location.



Connection lines that are part of the move are automatically displayed

With this feature it is easier to hide all the connection lines, then as you move the components the appropriate connection lines are automatically displayed and updated. Connection lines can be hidden by selecting **View » Connections** from the menus.

Note that you can also temporarily hide all the connection lines during component moves (including those normally displayed by the smart connection-line display feature) by pressing the N shortcut key. Doing this temporarily disables the connectivity analyzer.

As you move components the new dynamic connection length analyzer continually assesses placement quality based on connection lengths, and displays a green (strong) or red (weak) vector indicating current placement quality. The far end of the vector indicates a location for the component(s) that would minimize overall connection lengths.

Maintaining Component Clearances

Configure the Component Clearance Constraint in the Placement Tab of the **Design Rules** dialog. This rule is obeyed by both the on-line and batch DRC, enable these in the **Design Rule Check** dialog.

Working between the schematic and PCB

Transferring the design from the schematic editor to the PCB editor is more than a process of transferring component and connectivity information. There is also structural information about the design embodied in the schematic – a multi-sheet schematic breaks the design into logical groups of components, and the components are often arranged on these sheets in a manner that reflects how closely they connect to each other. This information makes the schematic an ideal platform to guide the PCB placement process from. Protel 99 SE incorporates features that help you transfer these design requirements from the schematic to the PCB.

Creating Classes and Rooms from the Schematic

On many designs the way the components are grouped on the schematic sheets directly reflects the way they should be grouped on the PCB. Protel 99 SE includes features that help you transfer this grouping information from the schematic to the PCB.

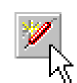
PCB component classes can automatically be created from each schematic sheet when the design is transferred from the Schematic Editor to the PCB Editor. To do this enable the Generate Component Classes and Placement Rooms for all schematic sheets in project option in the **Update Design** dialog.

A PCB component class is created for each sheet in the schematic, the class includes all the components on that schematic sheet. Each component class is given the same name as the schematic sheet it is created from, with any spaces removed. Multipart components that span more than one sheet are included in the class of the sheet that contains the first part of the component. A placement room is also created for each component class. For information on working with placement rooms refer to the topic *Working with Placement Rooms* later in the *PCB Placement* section.

Selecting PCB Components from the Schematic

Another feature that helps you transfer component relationship information from the schematic to the PCB is the ability to directly select PCB components from the schematic. To do this select the components on the schematic sheet, then select **Tools » Select PCB Components** from the Schematic Editor menus. These components will be selected on the PCB, and the PCB view zoomed to show the selection. With this set of selected components you can now create a component class, or a component union. Refer to the *PCB Selection* topic later in this supplement for details on how to create a component class from a selection. Refer to the following topic for information on creating component unions. You can also use the Arrange Within Rectangle feature to quickly pull the selected components out of a large group of components and arrange them in a group. Refer to the topic *Using the Interactive Placement Tools* later in *PCB Placement* section for information on how to do this.

Cross Probing from the Schematic to the PCB

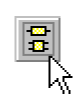


You can also move from an individual schematic component to the corresponding PCB component using the Cross Probe feature. Click on the Cross Probe button on the main toolbar, then click on the component on the schematic sheet. The PCB component will be located and centered in the window. You can also cross probe from a component pin, and from a net identifier, as well as perform the same cross probe options from the PCB back to the schematic.

Using Component Unions

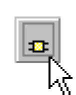
Create component Unions – unions are sets of components that you want to work with as a block. Components in a union maintain their relative positions within the union as they are moved, making it a valuable placement aid.

Creating a Component Union



To create a union select the components that you want in the union and click on the Create Union button on the Component Placement toolbar.

Removing Components from a Union

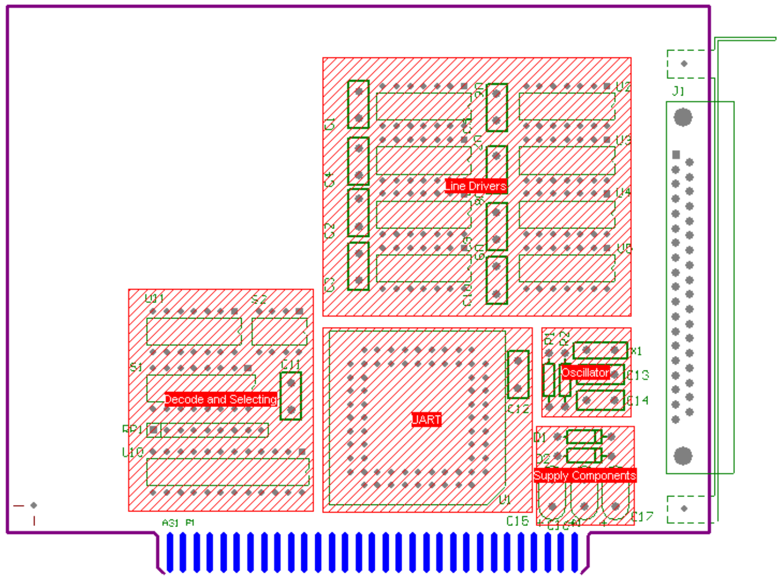


Unions can be broken by clicking on the Break Union button on the Component Placement toolbar, then clicking on one of the components in the union. Select the component(s) that you want to remove from the union in the **Confirm Break Component Union** dialog and click OK.

Breaking a Union

Select **Tools » Convert » Break All Component Unions** to break all current component unions. Note that a component can only belong to one Union at a time.

Working with Placement Rooms

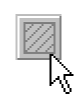


Create placement rooms to help with the placement process

Placement rooms are rectangular regions that assist in the placement of components. Components are assigned to rooms and can be automatically moved into their room, they also move with the room whenever the room is moved.

Room Definitions are part of the design rule system, and compliance with room definition rules can be checked by the on-line and batch DRC. Rooms can also used by the cluster-based autoplacer, by selectively locking components you can also autoplace on a room-by-room basis.

Creating Placement Rooms



Rooms can be placed from the **Place** menu, through the Placement Tab of the **Design Rules** dialog, or the Place Room button on the PlacementTools toolbar. Rooms can exist on the top or bottom layers, they are placed on the top layer by default. Once a room has been placed its properties can be defined by double-clicking on the room.

Assigning Components to a Room

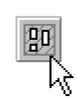
The scope of the Room Definition design rule defines the set of components that are assigned to that room. Edit the rule to set the scope and assign the components to the room.

Positioning and Sizing Rooms

A room can be modified graphically at any time, click once to focus it, then click on a handle to resize it. The entire room can be moved by clicking and dragging.

Note that when a room is moved the components assigned to the room are automatically moved with it. To prevent this and move a room without moving the components disable the Room Definition rule in the **Design Rules** dialog (disable a rule by unchecking the Enabled option). Rooms can also be locked in the **Room Definition** dialog, lock a room to prevent accidentally moving it.

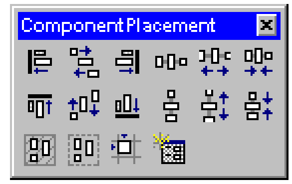
Placing within Rooms



You can quickly pull all the components that are assigned to a room into that room by clicking the Arrange Within Room button on the Component Placement toolbar.

Using the Interactive Placement Tools

The Component Placement toolbar include a number of alignment and spacing tools to help you quickly arrange the components.



Alignment Tools

Align left edge of selected components

Using the left edge of the left-most component as a reference, slides the selected components to the left, packing them as tightly as the component clearance rule allows.

Align right edge of selected components

Using the right edge of the right-most component as a reference, slides the selected components to the right, packing them as tightly as the component clearance rule allows.

Align vertical centers of selected components

Places selected components in a single column, aligned by their vertical centers. After clicking the button you are prompted to select a reference component, the other selected components are placed above this component.

Align top edge of selected components

Using the top edge of the top-most component as a reference, slides the selected components up, packing them as tightly as the component clearance rule allows.

Align bottom edge of selected components

Using the bottom edge of the bottom-most component as a reference, slides the selected components down, packing them as tightly as the component clearance rule allows.

Align horizontal centers of selected components

Places selected components in a single row, aligned by their horizontal centers. After clicking the button you are prompted to select a reference component, the other selected components are placed to the right of this component.

Spacing Tools

Make horizontal spacing equal for selected components

Distributes selected components equally between the left-most and right-most components in the selection. Their vertical position is not changed.

Increase horizontal spacing for selected components

The horizontal distance between the component reference points is increased by the amount specified in the X component placement grid.

Decrease horizontal spacing of selected components

The horizontal distance between the component reference points is decreased by the amount specified in the X component placement grid.

Make vertical spacing equal for selected components

Distributes selected components equally between the top-most and bottom-most components in the selection. Their horizontal position is not changed.

Increase vertical spacing of selected components

The vertical distance between the component reference points is increased by the amount specified in the Y component placement grid.

Decrease vertical spacing of selected components

The vertical distance between the component reference points is decreased by the amount specified in the Y component placement grid.

Moving Tools

Move to Rectangle

This tool arranges the selected components within the defined rectangle. Click on the button, then click once to define the first corner of the rectangle, move the mouse and click a second time to define the opposite corner of the rectangle. The selected components are arranged within the rectangle, working across and down.

Move to Room

This tool arranges the components that are assigned to a room within that room. Click on the button, then click on the room – the components associated with that room by the rule scope are arranged within the room, working across and down.

Move to Grid

Move the components to the nearest point on the component placement grid. The component X and Y placement grids are defined in the Options Tab of the **Document Options** dialog. Locked components are not moved.

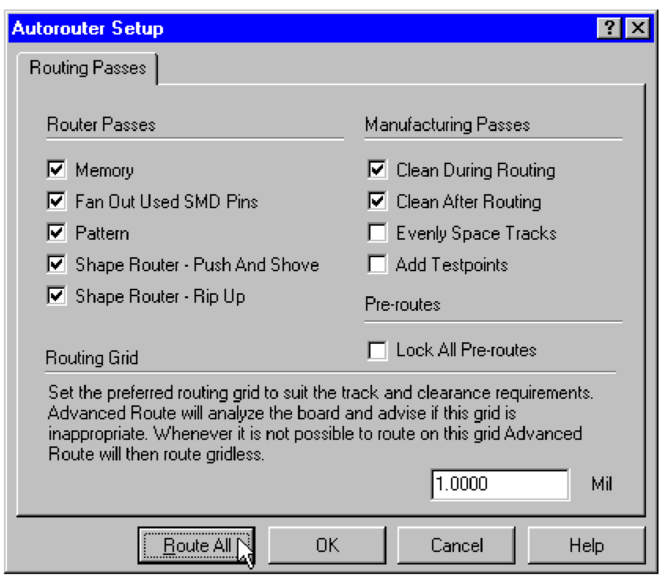
Autorouting

Design Compliance

Autorouter rule compliance is reported in the **Design Rules** dialog. The status line at the bottom of the dialog reports if the currently selected design rule definition is followed by the autorouter.

Cleanup Passes

Two new cleaning passes have been added to the autorouter. The first runs during the routing sequence, at the end of each routing pass, the second runs after all the routing passes are complete. The clean passes are enabled in the **Autorouter Setup** dialog. The clean passes are designed to be used in conjunction with the main routing passes, they focus on straightening the routing connections and cleaning the pad entries.



Enable the cleaning passes in the Autorouter Setup dialog

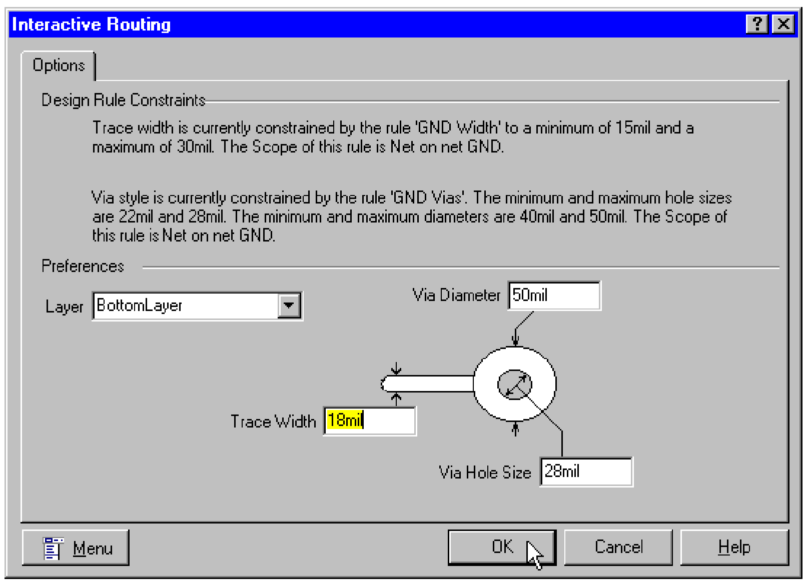
Manual Routing

Interactive Connection-line Selection

A new interactive connection-line selection feature has been added, when you click to start routing from a pad with multiple connection lines one is automatically selected, ready for routing. Press the CTRL+SPACEBAR shortcut to cycle through each of the connection lines coming from this pad.

Changing Routing Parameters on-the-fly

During manual routing the track width and routing via parameters can be changed on-the-fly by pressing the TAB key. This pops up the **Interactive Routing** dialog where you can change the track width, the via diameter and the via hole size. Changes made to these settings updates the Preferred attributes of the applicable **Width Constraint** and **Routing Via Style** design rules. If the values are changed to be outside the current maximum and minimum settings of the applicable rule they are automatically clipped.



Routing parameters can be changed on-the-fly by pressing TAB

Power Plane Connections

In Protel 99 SE vias can connect to power planes. This connection is created automatically when the net attribute matches the net attribute of a plane layer. If there are multiple plane layers defined for that net the via connects to each of these planes.

There is also a new shortcut key to quickly place a power plane connection via during manual routing. Press the forward slash key on the numeric keypad while routing to place a via connecting to the appropriate power plane. The via size is defined by the Routing Via Style design rule, the via-stack style (thru-hole or blind) is defined by the allowable drill-pairs specified in the Drill-Pair Manager (click on the Drill-Pairs button in the **Layer Stack Manager** dialog to setup the drill-pairs).

Polygon Plow Thru Mode

There is a new polygon plane plow through mode. If the Plow Through Polygon option is enabled in the **Preferences** dialog you can route over the top of a polygon when the Interactive Routing Mode option is set to Avoid Obstacle.

When you finish routing the polygon automatically repours, depending on the settings of the Polygon Repour options. The Repour option defines when a repour should occur, if Threshold is selected then polygons with more than the Threshold number of primitives will prompt to confirm repour before performing the repour.

Routing Shortcuts

Hold the CTRL shortcut key to temporarily suspend the electrical grid when the cursor is a crosshair. Use this during manual routing or primitive placement.

ALT shortcut key temporarily switches from the Avoid Obstacle mode to the Ignore Obstacle mode when the cursor is a crosshair. This allows you to route in Avoid Obstacle mode, and temporarily change to Ignore Obstacle if necessary.

When you press the * shortcut key to toggle to the next available routing layer the autovia feature adds a via. If the cursor is within a pad when the * shortcut is pressed a via is not added. Use this when you start routing and then realize that you are on the wrong layer, or when you are routing up to a pad and want to continue routing out from the pad on another layer.

PCB Selection

Selecting PCB Components from the Schematic

To help in the process of working between the 2 views of your design – the schematics and the PCB – you can now directly select PCB components from the schematic. To do this select the components on the schematic sheet, then choose **Tools » Select PCB Components** from the Schematic Editor menus. These components will be selected on the PCB, and the PCB view zoomed to show the selection. With this set of selected components you can now create a component class, which is described in more detail below.

Creating Selection Queries

Complex PCB selection queries can be created and stored with the new Query Manager. Select **Edit » Query Manager** to pop up the **Query Manager** dialog. To create a new query, first type in a Name then click the Add button to pop up the **Statement** dialog. Each statement specifies if an Object's Property is equal to (or not equal to) a Value. Define the statement and click OK to return to the Query Manager. Multiple statements can be added to a query, these are logically ANDed together.

When the query has been defined click the Apply button to select all objects that satisfy the query conditions.

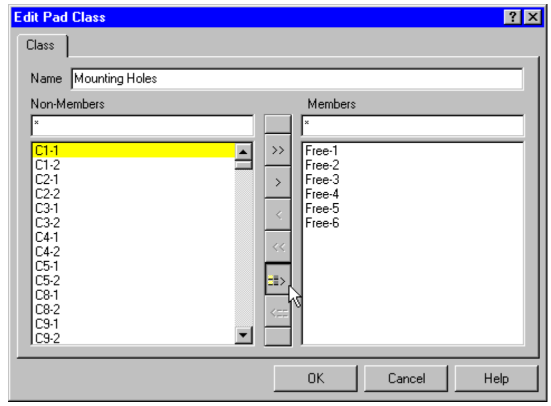
Selecting from the Panel

As you manipulate your design one of the most commonly used tools is selection. Selection can be used to move, copy or delete a group of objects, and is also a convenient way of highlighting an object or a group of objects. As well as direct selection (SHIFT+click) and selection from the menus, you can also select directly from the Browse PCB Panel. The following selection operations are available:

- Select the current net, and any node in that net
- Select the current component, and any pad on that component
- Select the current net class, and any net in that net class
- Select the current component class, and any component in that class
- Select objects targeted by a design rule

Set the Extend Selection option in the **Preferences** dialog to create multiple selections, disable it to always replace the current selection with the new selection.

Creating Classes from Selections



Creating a new pad class from the selected pads

Like selection, classes are an excellent tool for working on a set of objects. The advantage of classes over selections is that they are stored and can be used at any time. Classes can be used as the scope of a design rule, and also as a method of selection. Classes can be created for components, nets, pads, and from-tos.

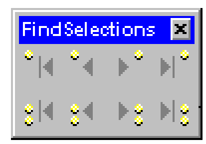
A convenient way of creating a class is to create it based on the current selection. Select pads, components, or nets using the standard selection strategies, or the selection buttons in the Browse PCB Panel.

To create a class from a group of selected pads select **Design » Classes** to pop up the **Object Classes** dialog, then click the Add button on the Pad Tab of the dialog to add a new class. In the **Edit Pad Class** dialog click the take-over-selected objects button to transfer the pads currently selected on the board from the Non-Members list to the Members list. Follow the same process for nets and components.

Stepping through Selected Objects

After selecting the objects you need to work on, use the Find Selection toolbar to quickly move from one selected object to the next. To display the toolbar press the B shortcut to pop up the **Toolbars** sub-menu and choose **Find Selections** from the menu.

The top row of buttons are used to step through the currently selected primitive objects (tracks, pads, vias, arcs, fills and strings), the bottom row of buttons are used to step through the selected group objects (components, dimensions, coordinates and polygons).



Use the Find Selection toolbar to step through the selected objects

PCB Editor Workspace

Testpoints and the Testpoint Finder

Pads and vias have 2 testpoint attributes, which can be enabled to define that pad or via as a top layer testpoint, bottom layer testpoint, or both layer testpoint. As well as enabling these attributes manually, they can also be enabled by the Testpoint Finder, and are automatically enabled for testpoint pads placed by the autorouter.

Specifying the Testpoint Requirements

The Testpoint Style rule defines the properties of testpoints, and the Testpoint Usage rule defines which nets must have testpoints. Refer to the *PCB Design Rules* topic earlier in this supplement for details on setting up these rules.

Finding Existing Pads and Vias that can be used as Testpoints

To find existing pads and vias that comply with the Testpoint Style design rule use the testpoint finder. To run this select **Tools » Find and Set Testpoints** from the menus. The board is scanned for pads and vias that comply with the testpoint design rule requirements. A message box reports how many testpoint vias and testpoint pads have been located, and how many nets failed to be allocated a testpoint. Existing testpoints can be cleared at any time by selecting **Tools » Clear all Testpoints** from the menus.

Automatically Placing Testpoints with the Autorouter

The **Autorouter Setup** dialog includes a testpoint pass. If this is enabled the autorouter checks each net to see if it has a testpoint, if not it attempts to add one.

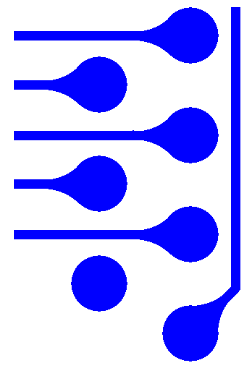
Reporting the Location of Testpoints

The CAM Manager is used to report the location of testpoints, refer to the topic *Generating the Manufacturing Files* later in this supplement for information on using the CAM Manager.

Reporting Which Nets Do Not Have a Testpoint

The design rule checker will report any nets which should have a testpoint, but do not. Enable the Testpoint rule checks in the **Design Rule Check** dialog (**Tools » Design Rule Check**) to identify these nets.

Adding Teardrops to Pads



Arc teardrops added to pads

Teardrops can be added to pads and vias by selecting **Tools » Teardrops** from the menus. The **Teardrop Options** dialog is used to control which objects must have teardrops. The Force Teardrops option will include teardrops on pads and vias which results in a DRC violation. There is also an option in the dialog to remove teardrops.

Selectively setting Mask Expansions for Pads and Vias

Normally solder and paste mask expansions are controlled by setting up solder and paste mask expansion design rules. If you need control over these settings for a specific pad or via use the Mask attributes in the **Pad** and **Via** dialogs. There is also a tent attribute, enabling this automatically closes the opening in the mask for this pad or via. Note that once the local pad/via mask settings are enabled these pads and vias are no longer covered or checked by the design rule system.

Netlist Manager

Select **Design » Netlist Manager** from the PCB Editor menus to pop up the **Netlist Manager** dialog. The Netlist Manager is used to perform functions such as: comparing the PCB netlist to an external netlist; generating a netlist from the routed copper; exporting the internal netlist from the PCB; adding, modifying and deleting nets; and updating the net attributes of routing primitives from the net names on the component pads.

Other New Features

Autopanning

Adaptive autopanning provides smooth controllable panning, regardless of the board size, display contents and zoom level. Select the Adaptive mode in the Autopan Style option in the **Preferences** dialog.

Panel to Workspace Tracking

When you click on a net name in the Browse panel the net is automatically highlighted on the board. Similarly, when you click on a net on the board it is also highlighted in the panel.

Status Bar Reporting

As you move the cursor over objects in the workspace the attributes of that object are displayed on the Design Explorer Status Bar. This option can be disabled in the Display Tab of the **Preferences** dialog.

The message zones on the Status Bar can also be resized – position the cursor over the arrow symbols on the status bar, when the cursor changes to a double-headed arrow click and drag to resize a message zone.

Ungrouping Polygons, Dimensions and Coordinates

Dimensions, coordinates and polygons can be ungrouped, select **Tools » Convert » Group/Ungroup** to ungroup any of these objects. Use this when creating metric dimensions in an imperial workspace, moving a coordinate string without changing the coordinate location, or to be able to copy and paste polygon primitives onto a plane layer.

Cleaning Routed Nets

The **Cleanup Net Objects** and **Cleanup All Net Objects** options in the **Tools** menu examine the routing and break track segments at T-junctions, pads and vias, and remove redundant track segments (where one track segment exactly overlays another track segment).

Placing Circles

Circles (closed arcs) can be placed by selecting **Place Circle** from the menus. Click once to define the center, move the cursor, then click a second time to define the radius.

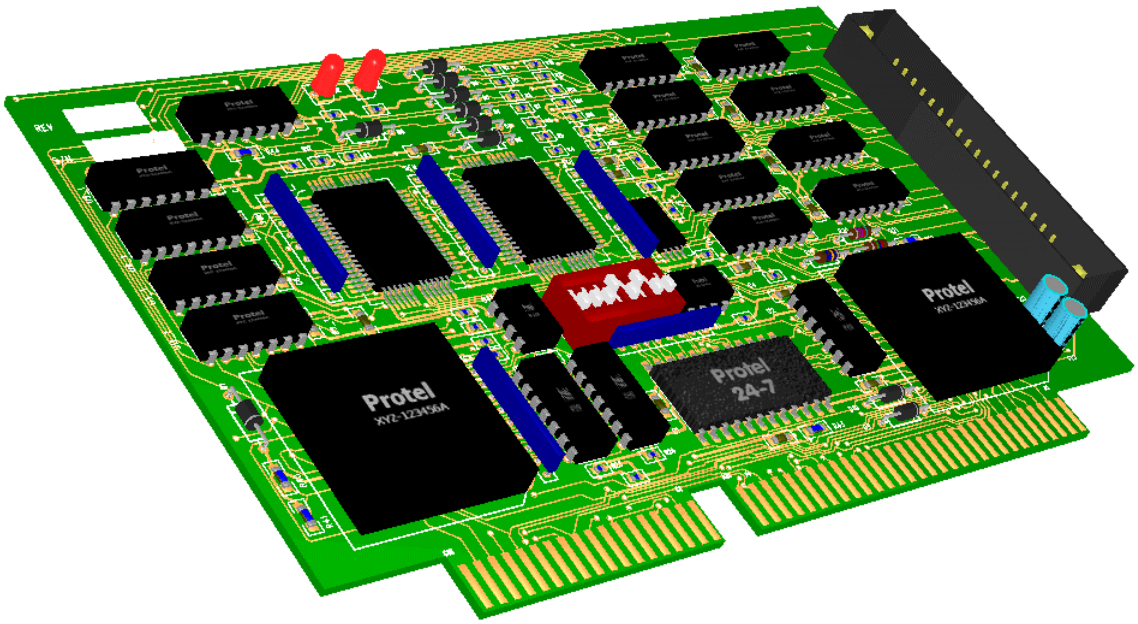
Placing Non-Electrical Lines

Track placement on electrical layers has been separated from track placement on non-electrical layers with the addition of a new **Place » Line** item in the **Place** menu. Use this when you are placing tracks on any non-electrical layers, including mechanical, keepout, and drill layers.

Net Name Length

Net names can be up to 255 characters in length in the PCB Editor, matching the net name length limit in the Schematic Editor.

3 Dimensional PCB Visualization



The 3D Viewer is a visualization tool that allows you to preview and print a 3D image of your PCB. The 3D Viewer is built around an OpenGL-based rendering engine, a standard graphics language supported by most graphics cards. It uses a run-time component modeling algorithm that uses the component designator prefix, footprint and outline shape to automatically select model and texture information and construct a suitable component model. Components that can not be recognized are automatically extruded.

Creating a 3D View of the Board

To create a 3 dimensional view of your board select **View » Board in 3D** from the PCB Editor menus. The board is analyzed and a 3D view is created in a new Window.

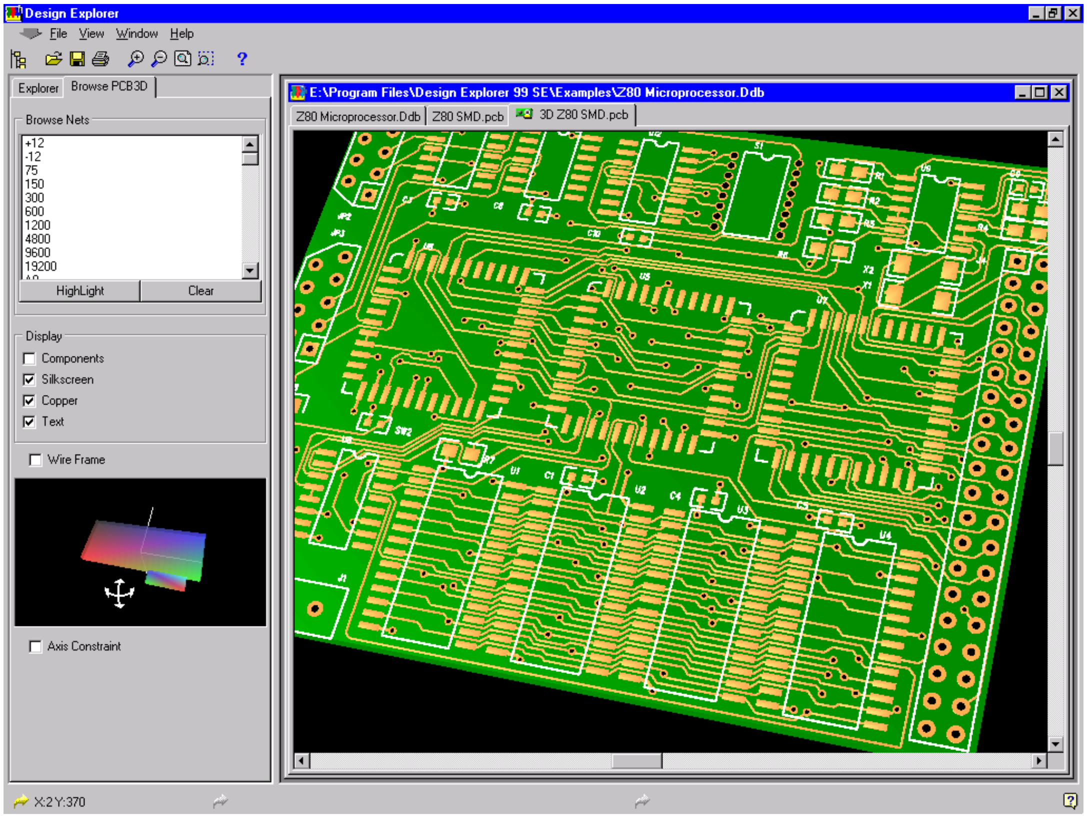
Changing the View of the Board

The 3D Viewer supports full rotational and zoom control, making it possible to display the board at any angle. The board can be rotated by clicking-and-dragging in the MiniViewer window in the panel. The standard PCB Editor display shortcuts are also supported – press the PAGEUP and PAGEDOWN keys to zoom in, the END key to redraw the view, and right-click and drag on the 3D image to display the slider hand and slide the 3D view around.

> The rendering process can be canceled at any time by left-clicking on the 3D image as it is being redrawn.

You can also selectively hide the components, silkscreen outlines, copper, and text strings. These options can be enabled in the panel, or the **Preferences** dialog.

The Browse PCB 3D panel also includes a highlight feature, click on a net name and click the Highlight button to highlight that net on the board. There is also an Animate option, which flashes the net that is being highlighted. The highlight color and the animate feature are set up in the **Preferences** dialog.



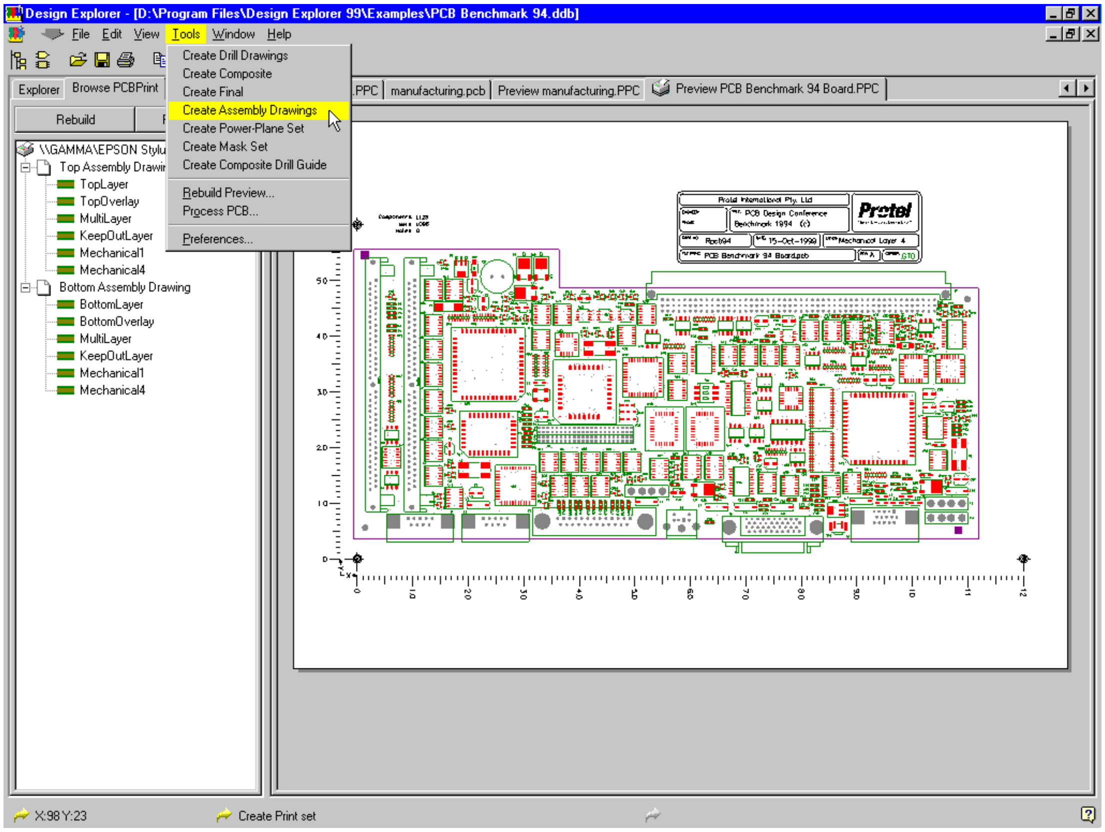
Use the display options in the panel to control what is shown on the board

Printing the 3D View

The 3D view can be printed by selecting **File » Print** from the 3D Viewer menus. This will print what is currently displayed in the 3D window.

Three print qualities are supported, Draft, Normal and Proof. The print quality is selected in the **Preferences** dialog (select **View » Preferences**).

Printing to a Windows Printing Device



Preview and configure the printouts before printing

An essential part of the design process is producing printed documentation about the PCB design. This could include a manufacturing drawing detailing the fabrication information, check plots for verifying the contents of each fabrication layer, and assembly drawings detailing component location information and loading order.

In Protel 99 SE printed output is created by preparing a preview of the required printouts, then printing them with the new Power Print feature. Using this approach you can define precisely what mix of PCB layers you want to print, set the scaling and orientation, and see exactly how it will look on the page before you print it.

Protel 99 SE's new Power Print engine also supports printing the current screen area and copying the current preview to the Windows clipboard, making it easy to include PCB information in your documentation.

The Power Print feature works by creating a Power Print Configuration document (**.PPC*). This PPC document details: which PCB is being previewed, the target printer, the set of printouts, and the PCB layers to include on each printout. When you open a PPC document this setup information is read, the PCB is analyzed, and the previews of the PCB are displayed in a separate Tab in the database window. You can then print the printouts as required.

Because the actual PCB data is not stored in the PPC document it must be extracted from the PCB when you create, modify, or open a PPC document. This analysis happened automatically – if you prefer you can disable automatic rebuilding in the Power Print **Preferences** dialog and then use the Rebuild button (when you change the preview configuration), or Process PCB button (when you modify the PCB) in the Browse PCBPrint panel to update the previews.

Setting up a Print Preview

To perform a preview of a PCB select **File » Print Preview** from the PCB editor menus. A PPC document will be created and opened, displaying the PCB as it will appear on the printed page(s). The default PPC document name for a PCB called *MyDesign.PCB*, is *Preview MyDesign.PPC*. Note that you can rename this document in the same way you rename any document in the Design Explorer.

> If a Power Print document with the default name of *Preview MyDesign.PPC* already exists when you select **File » Print Preview** it is automatically opened. To create a second PPC document rename the default PPC document.

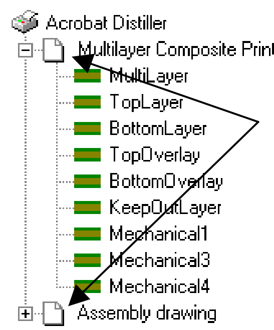
Once the preview appears, click on the **Browse PCBPrint** Tab in the Design Manager to display the current configuration of printouts. The default PPC is configured for a single composite printout, targeting your default Windows printer.

What is a Printout?

A Printout is a set of one or more PCB layers that will be printed as a single print job. Depending on the scaling, this may be on a single piece of paper, or tiled (spread) over a number of pieces of paper.

Each printout is represented as a page icon in the **Browse PCBPrint** panel.

Any combination of PCB layers can be included in a printout, and you can define as many printouts as you like in each PPC document.



This PPC document includes 2 printouts

Changing the set of Printouts

When you select **File » Print** from the PCB editor menus and the preview pages appear, the default configuration is a *composite* printout. A composite printout is a drawing with all the main layers overlaid on one another, simulating the real PCB.

There are a number of predefined printout sets, click on the **Tools** menu to display the list. When you select a different printout set the current configuration is replaced with the new configuration. You can easily create new printouts, select **Edit » Insert Printout** to add a new printout to the current Print Preview document. By default a new printout includes the top layer.

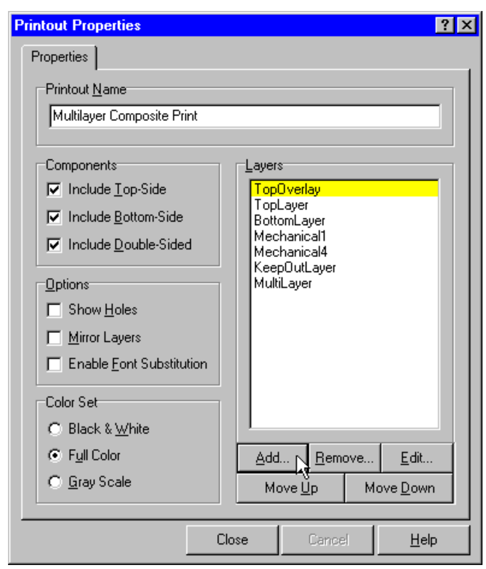
Specifying the Layers in a Printout

To display the set of layers in a printout click once on the small + symbol next to the printout icon. The order the layers are displayed in the panel is the order they are rendered on the actual printout.

To add and remove layers from a printout right-click on the printout icon and select **Properties** from the small floating menu. The **Printout Properties** dialog will pop up. The Layers region of the dialog displays the current set of layers that will be included in this printout. Use the buttons at the bottom of the Layers region to modify the layer set.

To add a new layer:

1. Click the Add button to display the Layer Properties dialog.
2. Select the layer you would like to add in the drop down Print Layer Type list.
3. Set the display mode for the primitives as required. Refer to the topic Configuring the Layer Properties for more information on the primitive display mode.



Add layers to a printout in the Printout Properties dialog

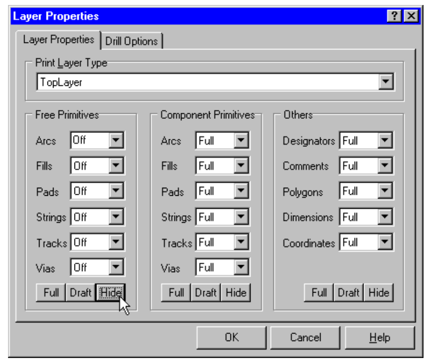
4. Click OK to close the Layer Properties dialog.

The new layer is added at the bottom of the layer list. This means that this layer will be drawn first in the printer's memory when the image is rendered. Each layer above is then rendered on top in turn. Use the Move Up and Move Down buttons to change its position in the render order.

> As well as adding mechanical layers individually to a printout, you can also automatically include them in all printouts. Enable the required mechanical layers in the Mechanical Layers Tab of the **Properties** dialog.

Configuring the Layer Properties

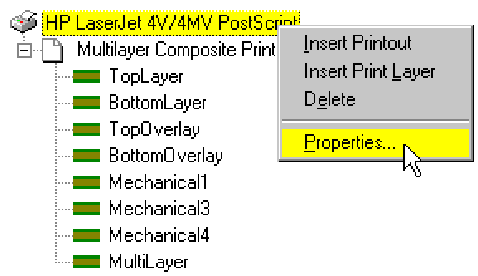
As well as being able to include any PCB layer on a printout, you can also control the way the primitives on that layer are displayed on the printout. This gives you complete control over what appears on the printed page. For example, consider an assembly drawing which includes the component overlay, the top layer (for the surface mount pads) and the multilayer (for the thruhole pads). The top layer would be configured to display the component primitives, so the surface mount pads were visible, and hide the free primitives, so the routing is not visible. The multilayer would also be configured to display the component primitives, so the throughhole component pads were visible, and hide the free primitives so that the vias were not visible.



Control the display of primitives in the Layer Properties dialog

Changing the Target Printer

When you create a new PPC document it is automatically configured to target the default Windows printer. To change the target printer, select **File » Setup Printer** from the Print Preview menus. Alternatively, right-click on the printer icon in the panel and select **Properties** from the popup menu that appears.



Right-click on the printer icon to select a different printer

Setting the Paper Orientation, Scaling and other Printer Options

Printing options such as paper orientation, scaling and margins are configured in the **PCB Print Options** dialog (right-click on the printer icon and select **Properties**).

There are 3 default scaling modes, click the Print What arrow to display these.

Standard Print – print the board at the scaling defined by the current Print Scale amount.
Whole Board on Page – automatically scale the board to fill the page.
PCB Screen Region – fit the current screen region onto the page. Note that any spare space on the page will be filled as well.

Printing

Once the printouts are correctly configured in the preview window you are ready to print. There are a variety of printing options available, click on the **File** menu to display these different options. The options include:

Print All – select this option to print all printouts in the current PPC document. Each printout is sent to the printer as a separate print job, with the same name as the printout.

Print Job – select this option to print all printouts in the current PPC document, with all printouts sent in the same print job. The print job has the same name as the PPC document.

Print Page – print the current page. If the printout is tiled over a number of pages a dialog will appear prompting you to type in the page number, or the page range. As a reference, each page of a multi-sheet tile printout includes a small red preview page number at the top left of the page. The numbers are not included on the printout, they can also be turned off by disabling the Preview page numbers option in the **Preferences** dialog.

Print Current – print all pages in the current printout.

Tip: printouts can be reordered by clicking and dragging the printout in the PrintPCB Panel.

Copying from the Preview Window to other Applications

Select **Edit » Copy** from the Power Print menus to copy the current printout to the Windows clipboard. You can then paste the image into other applications that support the Windows clipboard, such as Microsoft Word.

Note: The image is copied from the preview to the clipboard as an enhanced metafile. You can also export the printouts as either standard or enhanced WMF files.

Exporting the Printouts as WMF files

To export the current set of printouts as WMF files select **File » Export** from the Power Print menus. The **Export Options** dialog will pop up. The WMF files are saved in a folder in the design. There are 2 output folder options: overwrite folder, where the files are written into the same folder each time you export; or create time stamped folder where the files are written into a new folder each time you export.

You can also save a copy of the PCB into the folder each time you export by enabling the Save a Copy of the PCB File option. This will save a copy of the last saved version of the PCB.

To export the files directly onto the hard disk enable the Export Copy To option. Use the enhanced metafile option to specify if the metafiles are to be written as standard metafiles (16 bit), or enhanced metafiles (32 bit).

Color, Font and other Preferences

To set the layer color, font substitutions and other Power Print options select **Tools » Preferences** from the menus to display the **Preferences** dialog.

Color and Gray Scale Assignments

Each layer in a printout can be assigned a different shade of gray (or color). To change the gray shade for a layer, click once on the gray box next to the layer name, then select the new gray shade from the dropdown palette. To change the color for a layer, click once on the color box next to the layer name to display the **Choose Color** dialog.

Note: Use the Color Set options in the **Printout Properties** dialog to select Gray Scale or Color for each printout.

Font Options

Each of the 3 PCB Editor fonts can be substituted for a different Windows font on each printout. Use these options to specify the font that you want substituted for each of the PCB Editor's 3 fonts (Default, Serif, and San Serif).

Once the substitute fonts are specified and substitution is enabled, open the **Printout Properties** dialog to enable font substitution for that printout.

Overlap for Tiled Print

This option defines the amount of printed information that is duplicated on adjacent pages when a printout is tiled (printed over a number of pages).

Automatic Rebuild Option

Each time you change the setup options in one of the Power Print dialogs the data is re-analyzed to ensure that the previews are accurate. Disable this option to stop automatic rebuilding.

Select **Tools » Rebuild Preview** (or click on the Rebuild Preview button in the panel) to force a rebuild if you have changed a setup in one of the dialogs. Select **Tools » Process PCB** to force a rebuild if you have modified the PCB (or click on the Process PCB button in the panel).

Mechanical Layers

As well as adding mechanical layers individually to a printout, you can also automatically include them in all printouts.

Generating the Manufacturing Files

Completing the PCB layout is only the first part of the process that culminates in the fabrication and assembly of your PCB. The link between your design and the finished board are the print, Gerber and NC drill fabrication files, as well as the Bill of Materials, testpoint report, and pick and place assembly files. Apart from printed output, all the PCB manufacturing output files are generated by the CAM Manager.

For background information on photoplotting refer to the topic *What is Artwork?* later in this chapter. For information on printing from the PCB editor refer to the topic *Printing to a Windows Printing Device* earlier in this supplement.

Working With the PCB Manufacturer

When you start designing, you should have a clear idea of the output requirements of the PCB technology and production methods you will be using.

If you intend to use the services of a plotting bureau or PCB manufacturer take the time to consult with them before you start generating artwork. Bureaus and manufacturers often have specific requirements that must be reflected in the files or artwork that you submit. For example, you may wish to either “step and repeat” or panelize your Gerber files for efficient quantity fabrication.

To accomplish this, you have to know the film size accepted by the photoplotter, clearance requirements, etc, as well as the manufacturing tolerances involved. Planning for Numeric Control (NC) drilling, requires similar consideration.

In some instances, the bureau or fabrication facility will prefer to work directly with “raw” Gerber files (or even PCB files) rather than panelized Gerbers. Understanding these requirements will help you to plan the entire design process for efficient and trouble-free completion.

Generating Manufacturing Output with the CAM Manager

The CAM Manager is an independent editor that operates in the Design Explorer, alongside the PCB Editor, Schematic Editor, and other editors. The CAM Manager gives you total control over the setup and creation of the manufacturing output files, including Gerber, NC drill, pick and place, Bills of Materials, testpoint reports and DRC reports. The setups are saved in the design in a CAM Output Configuration document (.CAM) and can be modified at any time. CAM documents can also be copied from one design to another, making it easy to transfer your preferred manufacturing output file configurations.

The CAM Manager includes a CAM Output Wizard, which takes you through the steps to create each of the different output setups.

All enabled outputs can be generated at any time with a single command. The output files that are generated are written to a separate CAM Outputs folder, which can be timestamped if required. There is also an option to automatically export the CAM outputs directly to a disk drive.

The CAM Manager supports the following output types:

- Gerber files
- NC Drill files
- Testpoint reports
- Pick and Place files
- Bill of Materials (BOM)
- Design Rule Check (DRC) reports

Creating a new CAM Document

To create a new CAM document for the current PCB select **File » CAM Manager** from the PCB Editor menus. A blank CAM document will be created, and the Output Wizard will automatically start. The Wizard can be used to create an output setup for each type of supported output. You can also create the various output setups directly by selecting the required type in the **Edit** menu.

How the CAM Documents are Named

When you select **File » CAM Manager** from the menus the new CAM document is automatically named *CAM Outputs for MyDesign.CAM*, for a PCB named *MyDesign.PCB*. If a CAM document with this name already exists then this document is reopened. To create a second CAM document, first rename the original CAM document, then select **File » CAM Manager** from the PCB Editor menus.

Adding Output Setups to a CAM Document

There are 2 ways to add a new output setup to the current CAM document, you can use the CAM Output Wizard, or select the required output type in the CAM Manager **Edit** menu.

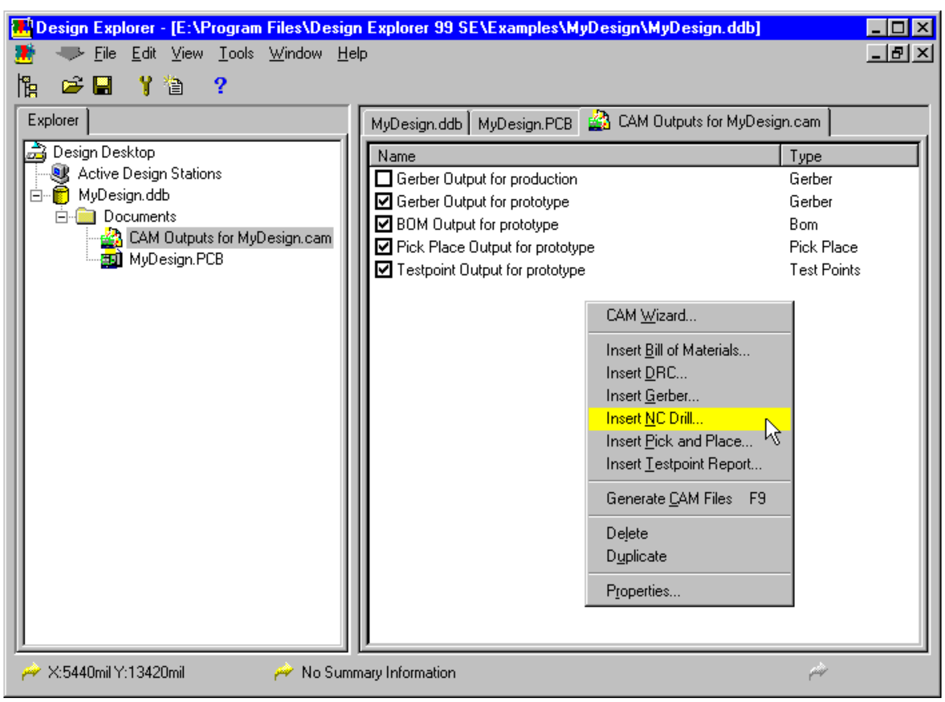
Using the CAM Output Wizard

The Wizard can be run by selecting **Tools » CAM Wizard** from the menus, or by right-clicking in the CAM document and selecting **CAM Wizard** from the floating menu. All of the output types can be set up from the Wizard, select the required type on the second page of the Wizard. If you are unsure of an option as you step through the Wizard leave it on the default setting – any setting can be changed later by double-clicking on the output setup in the CAM document to display the setup dialog. You can then use the What's This help icon (at the top right of each dialog) for information on a specific option in the different setup dialogs.

Directly Adding an Output Setup

Each CAM document can store multiple output setups, including multiples of the same kind of output setup. For example, one CAM document might include the Gerber and NC drill output setups for the prototype run, as well as a second set of output setups for the production run.

To add a new setup select the required type from the CAM Manager's **Edit** Menu, or from the right-click menu. The **Setup** dialog for that type will appear, configure the setup as required and click OK to close the dialog.



Right-click in the CAM document to add a new output setup

Modifying, Duplicating and Deleting an Output Setup

Double-click to modify an existing output setup. To delete or duplicate a setup right-click and select the appropriate choice from the floating menu.

Configuring the CAM Generation Options

CAM options are configured in the **CAM Options** dialog, select **Tools » Preferences** to display the dialog. Each option is described below.

CAM Output Folder

CAM outputs are written to a sub-folder that is created in the same folder as the PCB. If the Overwrite folder option is enabled, the folder will be called *CAM for MyDesign* (for a PCB called *MyDesign.PCB*), and the outputs written to this folder each time you select **Generate CAM Files** from the menus.

If the Create Time Stamped Output Folder option is enabled the date and time is appended to the end of the folder name, and a new folder created each time you select **Generate CAM Files** from the menus.

CAM Output Files Destination

If the One Folder For All Outputs Option is enabled all output documents are written to the main CAM folder. If the Separate folder for each output type option is enabled sub-folders are created within the main CAM folder to store the separate output types in.

Archive PCB File Option

If the Save a copy of the PCB option is enabled the last-saved version of the PCB is copied to the CAM folder when the CAM outputs are generated.

Export CAM Outputs

Enable the Export CAM Outputs option to automatically export the output files to the specified location on the hard disk.

Generating the Output Files

Output can be generated for any of the current output setups at any time. To generate output, first enable the setups you require by clicking on the small square next to the setup name in the CAM document. When the required outputs are enabled select **Tools » Generate CAM Files** from the menus (shortcut: F9). The outputs will be created according to the current setups in the **CAM Options** dialog.

Gerber File Output Setup

What are Gerber Files?

A PCB is fabricated as a series of layers that the manufacturer assembles into a board through a variety of chemical and mechanical processes. To fabricate each physical layer in the PCB the manufacturer uses an image of that layer – this image is referred to as a phototool. A phototool is a piece of clear film, with black lines, circles and other shapes forming exactly the same patterns as the content of that layer in Protel 99's PCB Editor.

Once the PCB design process is complete and the design has passed all the design rule checks, the Gerber files are generated, one for each layer needed in the fabrication process. The Gerber language is the standard language format used to transfer PCB layout data from the PCB design software to the phototool creation process. These Gerber files are then sent to the manufacturer, who loads them into a photoplotting machine and creates the phototools.

Each phototool is created by exposing the film to build up the image required for that layer. The information needed to form the image includes the shape and size of the objects on that layer, and the coordinates of these objects. The shapes are specified in the Gerber file as apertures, and typically these apertures are created from the board and included in each Gerber file, when they are referred to as embedded apertures. If the apertures are not embedded then they must be stored in a separate aperture file and supplied with the Gerber files.

Setting up the Gerber File Options

To include a Gerber setup in a CAM output configuration document select **Edit » Insert Gerber** from the CAM Manager menus to pop up the **Gerber Setup** dialog. Click on the What's This help icon at the top right of the dialog for detailed information about each of the options in this dialog.

Note that when you enable the Embedded Apertures option in the Apertures Tab of the **Gerber Setup** dialog the aperture list is created automatically, each time you generate the Gerber files. The apertures are then embedded in the Gerber files, according to the RS274X standard. This feature means that you do not need to worry if the current aperture list includes all the required apertures – unless your PCB manufacturer does not support embedded apertures it is highly recommended that you use this option. For more information on apertures refer to the topic *What is Artwork?* later in this chapter.

To ensure that the finished PCB meets your design and manufacturing requirements it is important that you contact the fabrication house and discuss their requirements before generating the Gerber files.

Some of the requirements you should discuss include:

Any apertures restrictions – most modern photoplotters are raster plotters which can accept any size aperture. Generally they also accept Gerber files with embedded apertures. In this situation enable the **Embedded Apertures** option in the **Apertures** Tab of the **Gerber Setup** dialog.

Mask expansions – mask expansions are required for the solder and paste mask layers. The solder mask layer defines where the manufacturer must apply a thin layer of solder to the bare copper on the board, typically on component pads. The paste mask layers define where solder paste is applied to the board during the assembly process, and is normally only required for surface mount components. Solder and paste mask expansions are specified in the **Manufacturing** Tab of the **Design Rules** dialog in the PCB Editor.

Power plane clearances – if the design includes internal power planes the manufacturer will specify the clearance required for through hole component pads and vias that do not connect to the plane. The physical connection parameters used for pads and vias that do connect to a plane must be set to suit the requirements of the design. Power plane clearances and connection styles are also set up in the **Manufacturing** Tab of the **Design Rules** dialog in the PCB Editor.

The units and format of the Gerber files – The units can be either inches or millimeters. The format specifies the precision of the coordinate data, this must be selected to suit the placement precision of the objects in the PCB workspace. For example, the 2:3 format has a resolution of 1 mil (1 thousandth of an inch). If your design has objects placed on a sub 1 mil grid then this format will not be adequate. Conversely, the higher precision formats may be more difficult and expensive to photoplot and manufacture.

If the plots should be centered on the film – the Gerber data can be automatically centered on the specified film by enabling the Center Plots On Film option in the **Gerber Setup** dialog. Note that Gerber coordinates are referenced from the absolute origin if the center plots on film option is turned off.

The drilling requirements – the drilling information is normally supplied in the form of NC drill files, refer to the *NC drill Output Setup* topic for more information.

File Extensions used to Identify each Gerber File

When you generate the Gerber output a series of files are created, each one corresponding to one of the layers enabled in the Gerber setup. These files are then loaded into a Gerber photoplotter, which produces the necessary phototools for PCB manufacture.

Each Gerber file is given the name of the PCB document, with a unique extension that identifies that layer and plot type. For example, the Top layer Gerber file for a PCB called *MyDesign* will be saved as *MyDeisgn*.GTL, to indicate "Gerber Top Layer". Because each design normally generates numerous Gerber files, these extensions help identify each file.

We recommend that you follow this convention which conforms to general industry practice. The following table shows the extensions that are used:

Top Overlay	.GTO
Bottom Overlay	.GBO
Top Layer	.GTL

Bottom Layer	.GBL
Mid Layer 1, etc	.G1, .G2, etc
Power Plane 1, etc	.GP1, GP2, etc
Mechanical Layer 1, etc	.GM1, .GM2, etc
Top Solder Mask	.GTS
Bottom Solder Mask	.GBS
Top Paste Mask	.GTP
Bottom Paste Mask	.GBP
Drill Drawing	.GDD
Drill Drawing – Top to Mid 1, Mid2 to Mid 3, etc	.GD1, GD2, GD3, etc
Drill Guide	.GDG
Drill Guide – Top to Mid 1, Mid 2 to Mid 3, etc	.GG1, GG2, GG3, etc
Pad Master, Top	.GPT
Pad Master, Bottom	.GPB
Keep Out Layer	.GKO
Gerber Panels	.P01, .P02, etc

NC Drill Output Setup

Numeric Control (NC) drill file requirements are configured in the **NC Drill Setup** dialog. This dialog appears when you edit an existing NC drill setup in a CAM output configuration document, or when you select **Edit » Insert NC Drill** from the CAM Manager menus to add a new NC drill setup to the current CAM output configuration document.

NC drill files are used to program a drilling machine with the information required to drill the holes in the blank PCB during the PCB fabrication process. Drill files specify the drill sizes, drill tool assignments and hole locations. Drill hole coordinates are referenced from the user-defined relative origin.

Three types of drill files are produced:

MyPCB.DRR - drill report, detailing the tool assignments, the hole sizes, hole count and the tool travel.

MyPCB.TXT - ASCII format drill file. For a multilayer PCB which uses blind and/or buried vias a separate drill file for each layer pair is created, with a unique file extension.

MyPCB.DRL - binary format drill file. For a multilayer PCB which uses blind and/or buried vias a separate drill file for each layer pair is created, with a unique file extension.

Setting the NC Drill Options

The NC Drill files should be created with the same format, or precision, as the Gerber files. For example, if the Gerber setup has been configured to use the 2:4 format, then the corresponding NC drill setup should use the same format. Click on the What's This help icon at the top of the **NC Drill Setup** dialog for information on a specific feature in the dialog.

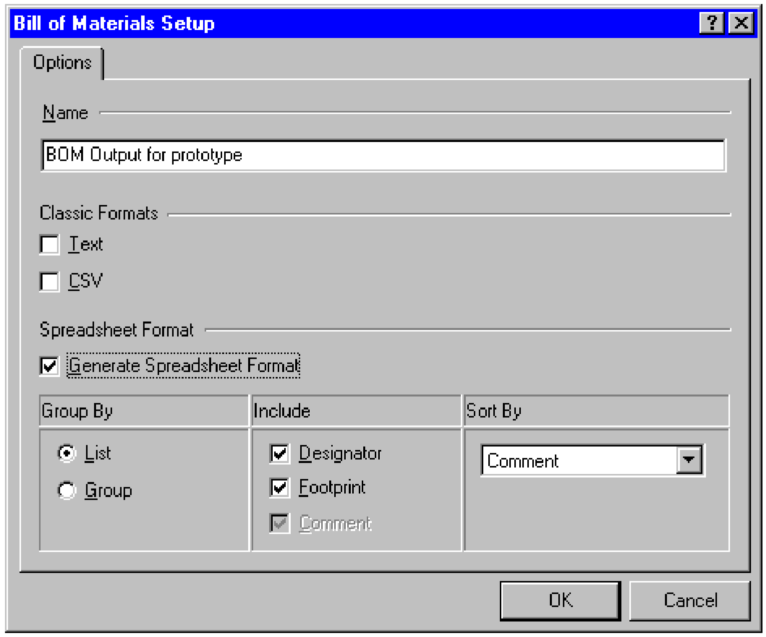
Bill of Materials Output Setup

Bill of Materials (BOM) file requirements are configured in the **Bill of Materials Setup** dialog. This dialog appears when you edit an existing Bill of Materials setup in a CAM output configuration document, or when you select **Edit » Inset Bill of Materials** from the CAM Manager menus to add a new Bill of Materials setup to the current CAM output configuration document.

Three BOM formats are supported, Text, CSV (Comma Separated Values) and Spreadsheet. The Text and CSV formats group the BOM listing by component comment and value. The Spreadsheet option allows you specify the format of the BOM.

Setting the Bill of Materials Options

Enable the output format that you want the BOM created in. More than one format can be enabled, each of the 3 BOM formats is given a different file extension. Click on the What's This help icon at the top of the **Bill of Materials Setup** dialog for information on a specific feature in the dialog.



Set up the Bill of Materials output requirements in the Bill of Materials Setup dialog

Pick and Place Output Setup

Pick and place file requirements are configured in the **Pick and Place Setup** dialog. This dialog appears when you edit an existing pick and place setup in a CAM output configuration document, or when you select **Edit » Insert Pick and Place** from the CAM Manager menus to add a new pick and place setup to the current CAM output configuration document.

Pick and place files are used to program machines that automatically load components onto the PCB during assembly. They are called pick and place because they *pick* the required component from a feeder tube and *place* it in the correct location on the PCB. Once a PCB has the components loaded it is then passed through another machine that solders all the connections.

The pick and place file includes the following information for each component:

- Designator
- Footprint
- Location - expressed in 3 formats; by geometric center, component reference point, and pad 1 location
- Side of board
- Rotation
- Component comment

Pick and place component location files can be generated in Spreadsheet, CSV and Text formats, and in imperial or metric units. Pick and place coordinates are referenced from the user-defined relative origin.

Setting the Pick and Place Options

Enable the output format that you want the Pick and Place file created in. More than one format can be enabled, each of the 3 formats is given a different file extension. Click on the What's This help icon at the top of the **Pick and place Setup** dialog for information on a specific feature in the dialog.

Testpoint Report Output Setup

Testpoint report requirements are configured in the **Testpoint Setup** dialog. This dialog appears when you edit an existing testpoint setup in a CAM output configuration document, or when you select **Edit » Insert Testpoint** from the CAM Manager menus to add a new testpoint setup to the current CAM output configuration document.

Testpoints are identifiable points on the board that are used to analyze the board. There are typically 2 types of analyses carried out on a board: bare-board testing, to test for shorts between the routed tracks; and assembled board testing, which tests for correct performance of the finished board. These tests are carried out by placing the board on a 'bed of nails', where testpoint probes (nails) contact the testpoints on the board.

A PCB testpoint is simply a pad or via that has one of its testpoint attributes enabled. Pads and vias can be specified to be a top layer, bottom layer, or both top and bottom layer testpoint.

Testpoints can be defined manually, searched for by the Testpoint Find feature, or placed by the autorouter. They are placed according to, and tested against, the Testpoint Style and Testpoint Usage design rules. The Testpoint Style rule reports testpoints that do not comply with the required physical parameters (size, etc), and the Testpoint Usage rule reports those nets which have not had a testpoint correctly assigned. Refer to the *Design Rules* chapter for information on setting up these rules.

The testpoint report is then used to find all pads and vias that have one or both of their testpoint attributes enabled. The testpoint report includes:

- Net name
- Testpoint name
- X and Y coordinates – referenced from the user-defined relative origin
- Side of board
- Hole size
- Testpoint type – thru-hole or surface layer

Setting the Testpoint Options

Enable the output format that you want the Testpoint report created in. More than one format can be enabled, each of the 4 formats is given a different file extension. Click on the What's This help icon at the top of the **Testpoint Setup** dialog for information on a specific feature in the dialog.

Design Rule Check Output Setup

Design Rule Checking requirements are configured in the **DRC Setup** dialog. This dialog appears when you edit an existing DRC setup in a CAM output configuration document, or when you select **Edit » Insert DRC** from the CAM Manager menus to add a new DRC setup to the current CAM output configuration document.

DRC testing and reporting can be performed directly in the PCB workspace by selecting **Tools » Design Rule Check** from the menus. The CAM Manager DRC report runs the same testing routines, it is included to simplify the process of preparing the necessary output files when the design phase is complete and the design is ready for release to manufacture. For detailed information on design rule checking refer to the chapter *Verifying the PCB Design* in the Protel 99 Designer's Handbook.

Setting the DRC Options

Enable the rules that you want tested in the **DRC Setup** dialog. Click on the What's This help icon at the top of the **DRC Setup** dialog for information on a specific feature in the dialog.

Transferring the CAM Document to Another Design

CAM documents can be copied from one design to another like any other design document. Close the CAM document prior to copying, right-click on its icon in the tree or the folder it is stored in to display the floating menu, and select **Copy**. Right-click in the target folder in the second design and select **Paste** from the floating menu. Like the Windows File Explorer you can also use drag and drop to copy from one design to another.

When you copy a CAM document from one design to another you may need to reset the target PCB. This can be done at any time when a CAM document is open by selecting **Tools » Set Target Board** from the CAM Manager menus.

Note: If you do change the target PCB and you are not using the Embedded Apertures option for the Gerber setup you must ensure that the aperture list is appropriate from the new PCB prior to generating the Gerber files.

What is Artwork?

Artwork is the name give to the pieces of film that are used by the PCB manufacturer to fabricate the PCB. These pieces of film, one for each fabrication layer, are normally created by a “photoplotter”, from the Gerber files generated from the PCB.

Artwork can also be generated by high-resolution Postscript “imagesetters”, that are used by graphic design and typesetting bureaus. These machines are capable of producing film positives at resolutions at 2540 dpi (dots per inch) or higher.

However, users should be aware that there are some limitations to using this approach for PCB artwork. The resolution of these systems does not necessarily translate into positional accuracy or linearity, particularly when measured over a large area. There are also film size restrictions. Postscript output files can be generated by the PCB Editor’s Power Print feature.

Photoplotted Artwork

Gerber format photoplotting provides the highest resolution output and is generally considered the method of choice for production PCB tooling as it provides the best quality artwork for board production. Photoplots will be required when the design is either large in total area, or of high-density with fine line details. Gerber outputs are generated by the PCB Editor’s CAM Manager.

About Photoplotters

Photoplotters are similar to pen plotters in many ways, the primary difference being that photoplotters use light to plot directly onto photosensitive film. The many advantages of this approach has led to the widespread adoption of photoplotting in the electronics industry.

Because the etching of printed circuit boards is generally based upon photographic techniques, the production of positive and negative photo-tools (or films) is an inherent part of the process. When the original artwork is a pen plot, a number of intermediate steps have to be performed to produce the final tools. Pen plots are generally plotted at least 2:1 scale to achieve reasonable accuracy and then photographically reduced.

Photoplotters provide sufficient accuracy to generate a precision 1:1 plot in a single operation. Photoplotting bureau services are widely available and all designers should carefully consider its advantages. To make the best use of photoplotting, it’s helpful to understand some key concepts.

Vector vs. Raster Plotters

Photoplotters fall into two general categories, vector and raster.

Vector plotters generally use an aperture “wheel” or “slide” to create the combination of “flashes” and “strokes” to “draw” an image. These make images in much the same way as pen plotters. They select a drawing instrument (or aperture) and describe a vector in the drawing space. The result can be seen as a line the width of which is defined by the aperture. Apertures are a collection of defined shapes which allow the plotter to plot varying track widths, pad shapes, etc. Flashes occur when there is no movement of the light source, strokes occur whenever there is movement while the light source is on. Some plotters use separate apertures for strokes and flashes in order to maintain consistent exposure. Others control the light intensity – all apertures serving for both uses.

Raster plotters do not use a system of fixed apertures. They read the Gerber file, storing an “image” of the whole plot, which is then scanned onto the film, line-by-line, not unlike a television image. Raster photoplotters can synthesize a virtually unlimited variety of different apertures, providing a great amount of flexibility to the designer.

Some photoplotters use the Postscript language. Photoplot files for these devices are prepared using an appropriate Postscript driver. For information about Postscript printing, see the *Postscript Printing Tips* topic later in this chapter.

You will want to know something about the "target" photoplotter, in order to make efficient use of its capabilities when you design.

> Contact your photoplot bureau before generating any photoplots. Matching available plotting options at the edit level can save considerable time and expense when generating Gerber phototools.

Photoplotter Languages

Nearly all photoplotters are controlled by a vector-based plotting language, developed specifically for this task, generically referred to as "Gerber" – a registered trademark of the Gerber Scientific Company. This language has become an industry standard (also known as RS-274). While the language has evolved to accommodate changes in both plotting equipment and design tasks, a number of potential difficulties and limitations must be considered by the designer when planning a job for Gerber output.

A Gerber format file describes a plot as a series of draft codes (or commands) and coordinates. The draft codes control the aperture to be used, turning the light "on" or "off" and so on. Coordinates define the position of the various flashes and strokes on the plot. This information is stored as an ASCII text file.

The structure of Gerber files can vary due to a number of "optimizations" that have been added to the format over time, to address the changing capabilities of plotting hardware. Your photoplot bureau may need to know details regarding Protel's use of Gerber format, so we have described it in some detail below.

Protel Gerber files are divided into individual commands, followed by carriage return code then a line feed code. Each record is terminated by the character "*".

The records may refer to an absolute location or a draft code which changes apertures. Thus a record might be "X800Y775*" which instructs the plotter to move to a particular coordinate or "D16*" which is a draft code or command, such as a new aperture selection.

Some plotters reserve draft codes D01–D09 for uses other than aperture selection, for example:

- D01 Turns the light source on.
- D02 Turns the light source off.
- D03 Flashes the light source.

On some older plotters the special code "G54" needs to be sent before each change of aperture code. The last Gerber record is terminated by the special record M02*, which is followed by another block, containing the character "hex 08," then 509 "spaces" (hex 20), then a carriage return and a line feed.

You can inspect any Gerber file with a text editor or word processor capable of loading an "un-formatted" text file.

About Apertures

All Gerber format photoplotters use apertures. Apertures describe the available tools used to draw on film. In the case of a vector plotter these apertures correspond to various sizes and shapes of holes in an aperture wheel or slide. Light is projected through these apertures onto the film emulsion.

Raster plotters are not limited to a set of specific aperture sizes and shapes. Raster imaging systems interpret the aperture information in the generated Gerber file and the entire plot image is synthesized and represented by a bitmap and plotted line-by-line, not unlike a television image.

Using Apertures

The apertures that will be used to translate your PCB file into a set of Gerber files are stored in a file with the extension .APT. Apertures can be regarded like plotter pens. Aperture descriptions include a shape, such as a 50mm square, and use – flash, stroke or anything (either flash or stroke).

Before you can generate a Gerber file, you can either load an aperture file that matches the capabilities of the target plotter, or you can let the PCB Editor automatically create an aperture file, extracted from the primitives (tracks, pads, etc) in the current PCB file. When targeting a vector plotter, the apertures in the .APT file must correspond to the apertures available on the actual aperture wheel or slide to be used. The photoplotting bureau will supply the aperture table to suit their vector plotter. Raster plotters use the aperture file to translate draft codes directly into an image “map”. If the target plotter is a raster device, you can generate the apertures from the PCB and supply the generated aperture table with the Gerber files. Your photoplotting bureau will supply the required file generation details.

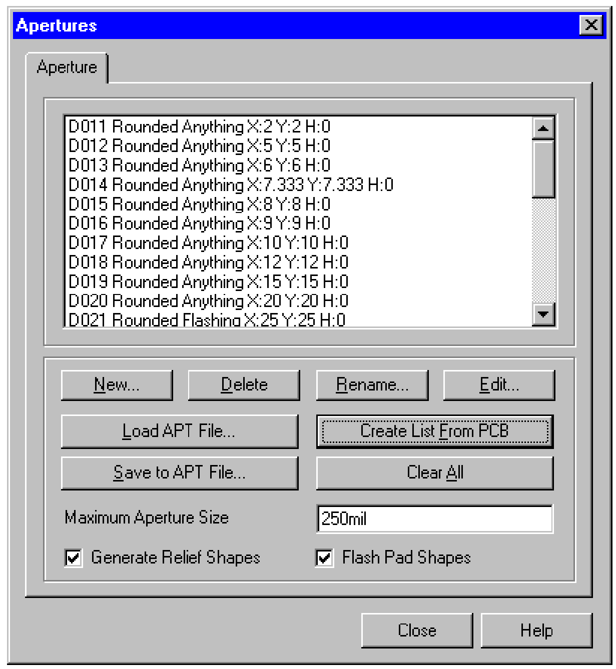
When you use an existing aperture file, the PCB Editor scans the primitives (tracks, pads, etc) in the PCB file and matches these with aperture descriptions in the loaded .APT file. If there is no exact match of aperture to primitive, the PCB Editor will automatically “paint” the primitive with a suitable smaller aperture. If there is no aperture suitable to “paint” with, a .MAT match file will be generated listing the missing apertures and Gerber file generation will be aborted.

If targeting a vector plotter, use primitives (track and pad sizes and shapes) for which there is a matching aperture. If the designer is familiar with the aperture set supported by the target photoplotter and tailors the choice of objects placed on a PCB design accordingly, the photoplotter will be able to faithfully reproduce the file in the most efficient manner.

Loading and Editing Apertures

Select the **Design » Aperture Library** menu item to load, create, or edit the apertures used by the photoplot output routines. When you choose this menu item the **Apertures** dialog will pop up. Any currently loaded apertures will be listed. These apertures would be used if you generated a photoplot at this time. If there are no apertures loaded, the list is empty. Options allow you to work with new or existing aperture files. Changes are applied to the aperture file currently loaded into memory. These changes do not become permanent until you use the Save to APT File button.

> You can define up to a maximum of 1000 different draft codes, in the range D00-D9999, although some of these codes (usually D00-D09) may be “reserved” when targeting some plotters, so use of these codes is not generally recommended.
>
> When creating new apertures do not include the “D” in the **Draft Code Number** dialog.



All aperture manipulation is done in the Apertures dialog

Postscript Output

High-resolution Postscript “imagesetter” output is now widely available from graphic design and typesetting bureaus. This equipment is capable of producing film positives at resolutions as high as 2540 dpi (dots per inch) and can provide a low-cost alternative to Gerber plots.

However, users should be aware that there are some limitations to using this approach for PCB artwork. The resolution of these systems does not necessarily translate into positional accuracy or linearity, particularly when measured over a large area. There are also film size restrictions. Postscript output files can be generated by the PCB Editor’s Power Print feature, refer to the topic *Printing to a Windows Printing Device* for information on printing from the PCB Editor.

Postscript Printing Tips

Postscript printers and “imagers” generally produce output between 300 and 2540 dpi. Because of the high resolution obtainable from these devices, many users are interested in producing artwork quality Postscript prints as a lower-cost alternative to Gerber plots. However, there are a few limitations which should be considered before printing.

High-resolution laser imagers print directly onto film or sensitized paper. While these devices are quite accurate horizontally, they do not always achieve consistent linearity, particularly on devices where the film or paper moves off a roll, then through the printing mechanism via a series of rollers.

Some typesetting / graphic arts bureaus now use Postscript imagers that use cut, rather than roll film, mounted on a large drum. These imagers suffer much less from linearity problems and may provide a suitable alternative to Gerber plots for non-critical designs.

To test any Postscript device, create a file with vertical and horizontal tracks of known length and carefully measure the output with a rule of known accuracy. This will allow you to apply a correction factor scale setting to either axis, which should minimize the problem. The amount of linearity error may not always be constant, so you should check each final artwork print for accuracy before committing the art to fabrication.

Another problem with 300 or 600 dpi “desktop” laser printers is the “overspray” and “bleed” effects created when the toner is fused to the paper. Small particles adhere to the paper on either side of lines, etc, creating the potential for unwanted effects in your artwork.

When designing for laser print artwork, you should keep the clearances generous, and again, print at a reasonable scale to minimize scale and bleed effects.

The print quality obtainable with a laser printer is largely determined by the paper. A number of special papers are currently available (primarily for the graphics arts trades) which reduce this toner “bleed” into the paper, hence making the outline sharper. Some of these special papers are slightly heavier and treated to resist the waxes and glues used for paste-up, making them easier to handle. Be especially careful to keep these paper laser prints clean.

Postscript compatible photo-typesetting equipment has the advantage of being able to provide output at very high resolutions (up to 2540 dpi). These devices can also print a direct film positive to A3 (or “B”) size.

However, the concern with linearity, described above, applies to these devices as well. The problem of linear accuracy will already be familiar to imagesetting bureaus who provide color separations to the graphics arts industry.

Some Postscript printers will “time out” and discard the current data when they do not receive the end of page marker within a specified time. This can cause problems where you seem to be missing pages from your plots. If you experience this problem using a Postscript printer or any other printing device then you should go to the Control Panel, select the printer icon, select the printer and click the Configure button. Change the Transmission Retry to 500 seconds, or some larger number. This will allow the printer sufficient time to catch up before the Print Manager gives up.

Pen Plotting

You can plot to a pen plotter via a Windows plotter driver. This should not be a problem for newer devices that use raster, rather than vector plot routines, such as the newer large format “ink jet” type plotters. There are also true vector plotter drivers available, contact your plotter supplier for further information.

File Importers and Exporters

The PCB Editor can import from and export to a variety of other file formats. Exporting options include older version Protel file formats and AutoCAD® DWG/DXF. Importing options include older version Protel file formats, Gerber files and Orcad Layout® PCB files.

To import a file select **File » Import** from the PCB Editor menus. Select the required format in the drop down file type list at the bottom of the **Import File** dialog.

Working with Older Protel Formats

The PCB file format has changed in Protel 99 SE, the file format is now referred to as PCB 4.0. To save a PCB back to the previous format (PCB 3.0) which was used by Advanced PCB V3, Protel 98, Protel 99 and Protel99 SP1, select **File » Save Copy As** from the PCB Editor menus.

Protel 99 SE can import all older version Protel PCB files. Import the file into the design database in the normal way (shortcut: drag and drop the file from the Windows Explorer into the Design Explorer), then double-click on it to open it.

Mechanical CAD Interface

Protel 99 SE includes full support for importing from and exporting to both DWG and DXF format files. Select **File » Import** or **File » Export** from the PCB Editor menus and set the file type to DXF/DWG at the bottom of the dialog.

Summarizing the Mechanical CAD Interface features:

- Full support for DWG/DXF import/export to the PCB editor, all versions from 2.5 to R14.
- User-definable layer mapping on import. The **Import from AutoCAD** dialog will appear during the import process, where the layer mapping is defined.
- Import from AutoCAD model space or paper space.
- Automatic import scaling if the import data is larger than the PCB workspace.
- Component to block and block to component translation.
- Supports metric or imperial units.

Use the Help button and What's This help in the dialogs for more details on the import/export options.

Orcad Layout to Protel PCB Interface

Protel 99 SE can import binary format Orcad Layout V9.x design files. Select **File » Import** from the PCB Editor menus and set the file type to MAX at the bottom of the dialog.

Summarizing the Layout Importer features:

- Directly load Orcad Layout (V9.x) PCBs into the PCB editor.
- User-definable layer mapping in the **Orcad Layout Importer** dialog. This dialog appears during the import process.
- Comprehensive, multi-level import reporting.
- Import Layout (*.LLB) libraries into Protel 99 SE's PCB Library Editor.
- Import a library directly from a Layout PCB file.

Use the Help button and What's This help in the dialogs for more details on the import options.

Schematic Capture

Design entry enhancements include – new schematic to PCB design integration features, a powerful user-controlled design annotation system, new file import/export options, improved library editing and library management, and numerous workspace editing enhancements.

Schematic Editor Workspace Enhancements

Schematic to PCB Design Integration

As a designer you often need to move design information back and forth between the 2 different views of your design – the schematic and the PCB. This process, known as *design synchronization*, is an integral part of the design process, and may need to be carried out many times through the course of the design cycle. Performing a design synchronization can be done at any time by selecting **Design » Update PCB** from the Schematic Editor menus, or **Design » Update Schematic** from the PCB Editor menus. When either of these menu items are selected the **Update Design** dialog appears.

The options in the **Update Design** dialog are used to control the synchronization process. Three new options have been added in Protel 99 SE.

Creating Component Classes and Room Definitions

On many designs the way the components are grouped on the schematic sheets directly reflects the way they should be grouped on the PCB. If this is the case for your design you can create PCB component classes for the components on each sheet when you synchronize the design. These component classes can then be used for setting up design rules, for selection, and during placement.

To create PCB component classes from each schematic sheet enable the Generate Component Classes and Placement Rooms for All Schematic Sheets in Project option in the **Update Design** dialog. Each component class is given the same name as the schematic sheet it is created from, with any spaces removed. Multipart components that span more than one sheet are included in the class of the sheet that contains the first part of the component.

A PCB placement room is also created for each component class. These placement rooms are spread across the board, ready for positioning. For information on working with placement rooms refer to the topic, *Working with Placement Rooms,* in the *PCB Design* section of this supplement.

Creating Net Classes from Buses

A PCB net class can be created for each schematic bus. To do this enable the Generate Net Classes for all Buses in Project option in the **Update Design** dialog.

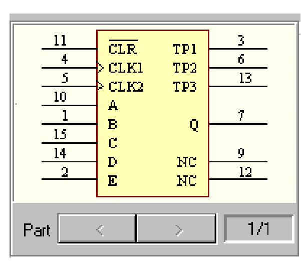
Updating the Net Names on the Routing

If the design has changed so that the net names on the routing no longer match the net names on the PCB component pads, enable the Assign Net to Connected Copper option in the **Update Design** dialog to automatically reapply the net names on the pads to all the connected routing.

Browsing Schematic Libraries

Browsing in the Browse Schematic Panel

Use the MiniViewer window to Browse through the component libraries in the current library list. Set the Browse mode in the panel to Libraries to display the MiniViewer.



Use the MiniViewer to browse the libraries

Browsing with the Browse Libraries dialog

You can also browse through the component libraries in the current library list using the **Browse Libraries** dialog. Select **Design » Browse Library** to pop up the dialog.

Multipart Component Numbering

Multipart components can use either a numeric or alpha part identifier suffix, for example U1:1, U1:2, etc, or U1A, U1B, etc. Set the multipart suffix option in the Schematic Editor **Preferences** dialog. Note that this is an environment setting, it applies to all currently open sheets.

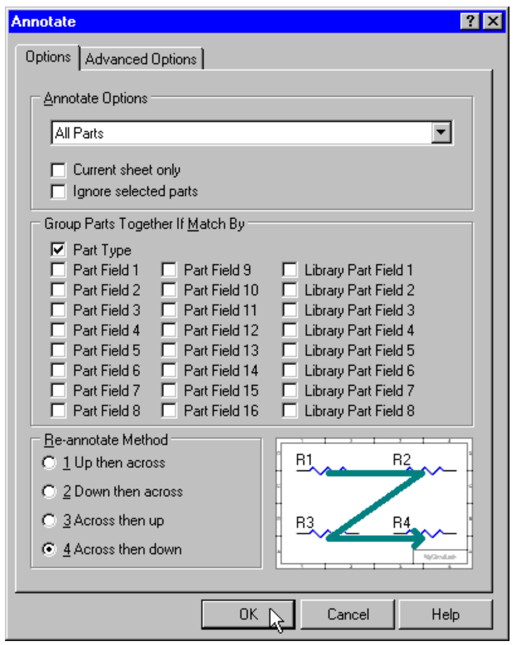
Pasting and Moving Selections

An automatic keep-within-sheet feature ensures that objects are not placed outside the sheet boundary. When a selection is being moved, or pasted into a schematic sheet, it is automatically kept within the sheet borders. If you attempt to paste into a sheet that is too small for the clipboard contents a warning message will appear, asking if you wish to continue.

Positional and User-Controlled Component Annotation

Positional Annotation

Annotation is the process of setting the designator for each schematic component to be unique. Schematic components are annotated based on their position on the sheet. The position of each component is determined by the location of its designator within the reference zones around the edge of the sheet. To change the number of sheet border regions enable the Use Custom Style option in the **Document Options** dialog, and change the X Ref Region Count and the Y Ref Region Count settings.



Select Tools » Annotate to annotate the schematic

Controlled Sheet-by-Sheet Annotation

Schematic component annotation can also be selectively controlled for each sheet, with a user-defined designator range and/or suffix. To do this disable the Current Sheet Only option in the Options Tab, then click on the Advanced Options Tab. This Tab will list all the sheets in the project. Enable the check box next to each sheet that requires controlled annotation (any sheet that is not checked is annotated in the normal way according to the settings in the **Annotate** dialog's main Options Tab).

There are 2 controlled annotation methods, by Range and by Suffix. To annotate by range enter a From and To value for each sheet, for example From=1, To=1000. Note that the annotation process does not permit duplicate designators, you must include a unique suffix if the range is repeated for more than one sheet.

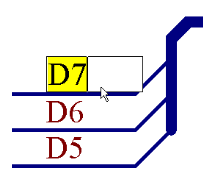
The Suffix options allow you to repeat the range for multiple sheets. Both alpha suffixes (A, B, C, etc) and numeric suffixes (_1, _2, _3, etc) are supported. Use these annotation options if you are annotating a multi-channel design and you want common in-channel numbering across all channels, for example, R1_1, R2_1, C1_1, etc, for channel 1, then R1_2, R2_2, C1_2, etc, for channel 2, and so on.

Excluding Components from the Annotation Process

Components can be excluded from the annotation process by selecting them. Enable the Ignore Selected Parts option in the **Annotate** dialog to exclude these parts from the annotation process.

On-Sheet Text Editing

Text strings can be edited directly on the schematic sheet. Click once to focus a text string, click a second time to edit the string directly on the schematic sheet. This behavior can be disabled by turning off the Enable In-place Editing option in the **Preferences** dialog.



Including the Document Name on the Schematic

The name of the current document can be automatically included on the schematic by placing the special string .Filename_No_Path. The text **Annotation** dialog includes a list of all special strings in the drop down list.

Enhanced On-Sheet Editing

Rotating and Mirroring Ports

Ports can be rotated and mirrored when they are being moved. Press the SPACEBAR to rotate a port, press the X or Y key to mirror it along the X or Y axis.

Sheet Entries

Sheet entries can be placed on all 4 sides of a sheet symbol, click and drag a sheet entry to change its position. Sheet Entries can also be copied and pasted between sheet symbols.

Autopanning

Whenever you are working with an editing cursor (a crosshair cursor) you can autopan across the schematic by moving the cursor up to the window frame. The autopanning speed is controlled by a sliding speed control in the **Preferences** dialog, and is independent of sheet size and sheet contents.

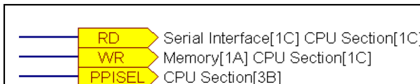
Cross-sheet Port Referencing

You can include sheet-to-sheet port referencing on your schematic, in one of two ways. The two options are in the Schematic Editor **Reports** menu and include:

Add Port References (Flat) – select this to include a string next to each port, detailing the sheet name and grid reference of all other ports of the same name in the project.

Add Port References (Hierarchical) – select this to include a string next to each port, detailing the sheet name and grid reference of the sheet entry that this port connects to.

Port references can be removed at any time by selecting **Reports » Remove Port References** from the menus. Port references are a calculated attribute of the port, they can not be edited and are not stored with the design. Their location is determined by the location of the port on the sheet and the position of the connecting wire.



Include port referencing to track connections between sheets

Naming Net Identifiers

Buses

Buses can be named either ascending or descending, for example D[0..15], or D[15..0].

Power Ports

There are 7 styles of power ports available in the schematic editor. Four of these are generic styles that can be used for any type of net, and include the net name in their presentation on the sheet. Three of the styles are dedicated ground or earth style symbols. For these 3 styles the net name can be pre-defined in the **Preferences** dialog.

Net Names

Net names can be negated with a single leading backslash ("\") character. Enable this option in the **Preferences** dialog.

Output Generation

Printing Markers and Directives

Error markers, no ERC markers, and directives can be excluded from the printout by disabling the Include on Printout options in the **Schematic Printer Setup** dialog.

Bill of Materials

Settings in the Bill of Materials Wizard are remembered between editing sessions, and lists of designators are logically sorted.

Workspace Shortcuts

Placing Components

Components can be placed on the sheet from the Browse Schematic panel, or by selecting **Place » Part** from the menus to pop up the **Place Part** dialog. The **Place Part** dialog includes fields for specifying the designator, part type (component value) and footprint. There is also a Browse button, which pops up the **Browse Libraries** dialog.

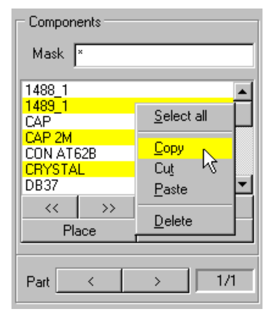
New and Updated Shortcut Keys

- Hold the CTRL shortcut key to temporarily suspend the snap grid when the cursor is a crosshair. Use this when you are positioning text, or some other object that needs to be moved off-grid.
- The CTRL shortcut key is also used to initiate a drag operation (keep the wires attached as a component is moved). To do this hold CTRL, click and hold on the component, then release CTRL and start dragging the component. You can also hold the CTRL key down during a drag to suspend the snap grid.
- Press the BACKSPACE shortcut key to delete the last-placed vertex during wire, bus, or line placement.
- Hold the ALT or ALT+SHIFT shortcut keys to constrain object movement to the X or Y direction.

Schematic Library Editor Enhancements

Copying Components

Components can be copied and pasted between libraries, and from a schematic to a library. To copy between libraries select the component(s) in the Library Editor panel using the standard Windows selection keys (left-click, SHIFT+click and CTRL+click). Once the components have been selected click the right mouse button to pop up the floating menu and select **Copy**. Change to the target library, right-click in the Library Editor panel, and select **Paste** to add them to the target library.



Right-click in the Panel to copy the selected components

Component Rule Check

Component Rule Check feature reports any pins missing in a sequence of pins.

Mechanical CAD Interface

Protel 99 SE includes full support for importing from and exporting to both DWG and DXF format files. Select **File » Import** or **File » Export** from the Schematic Editor menus to start the process.

Summarizing the Mechanical CAD Interface features:

- Full support for DWG/DXF import/export to the Schematic editor, all versions from 2.5 to 2000.
- Component to block and block to component translation.
- Import scaling option.
- Option to include the schematic template during export.

Use the Help button and What's This help in the dialogs for more details on the import/export options.

Orcad Capture® to Protel Schematic Interface

Protel 99 SE's Schematic Editor can import Orcad Capture binary format schematic DSN files. To import a Capture DSN file first import it into a folder in the design (right-click in a database folder and select **Import** from the floating menu). Once the DSN file has been imported double-click on it to start the translation process. Once the translation process is complete the sheets in the design are displayed in the Design Explorer panel. Follow the same steps to import a Capture library file.

- Import Orcad Capture (V7.x and V9.x) binary DSN schematic design files.
- Import Orcad Capture (V7.x and V9.x) binary OLB schematic library files.

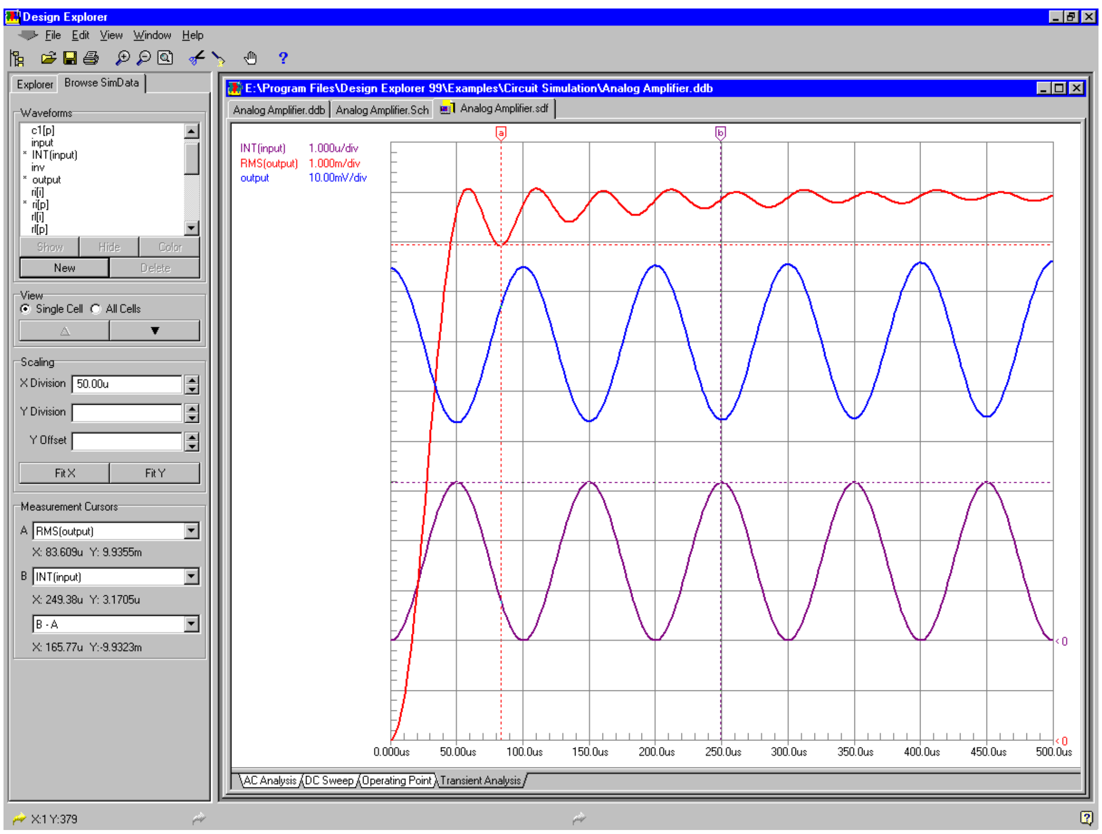
Circuit Simulation

Protel 99 SE includes a powerful mixed-signal circuit simulator. The Circuit Simulator uses an enhanced version of Berkeley SPICE3f5/Xspice, allowing you to accurately simulate any combination of analog and digital devices.

Mathematical Functions and Waveforms

As part of the analysis of your design you may want to perform a mathematical operation on one or more of the simulation signals, and view the resultant waveform. This feature is an integral part of the simulator's waveform viewer, you can construct a mathematical expression based on any signal available in the simulated circuit.

To define a mathematical function click the New button in the Browse SimData Panel, which pops up the **Create New Waveform** dialog. There are 3 parts to the dialog; a list of available Functions, a list of currently available Waveforms, and an Expression building field. The Expression can be constructed by either typing it in directly, or by clicking to select a function in the Functions list, then clicking to select the signal that you want to apply that function to.



Create mathematical expressions and display the results with the signal waveforms

Listing of Functions and Operators

Formulae can be based on any available waveform and support the following operators and functions:

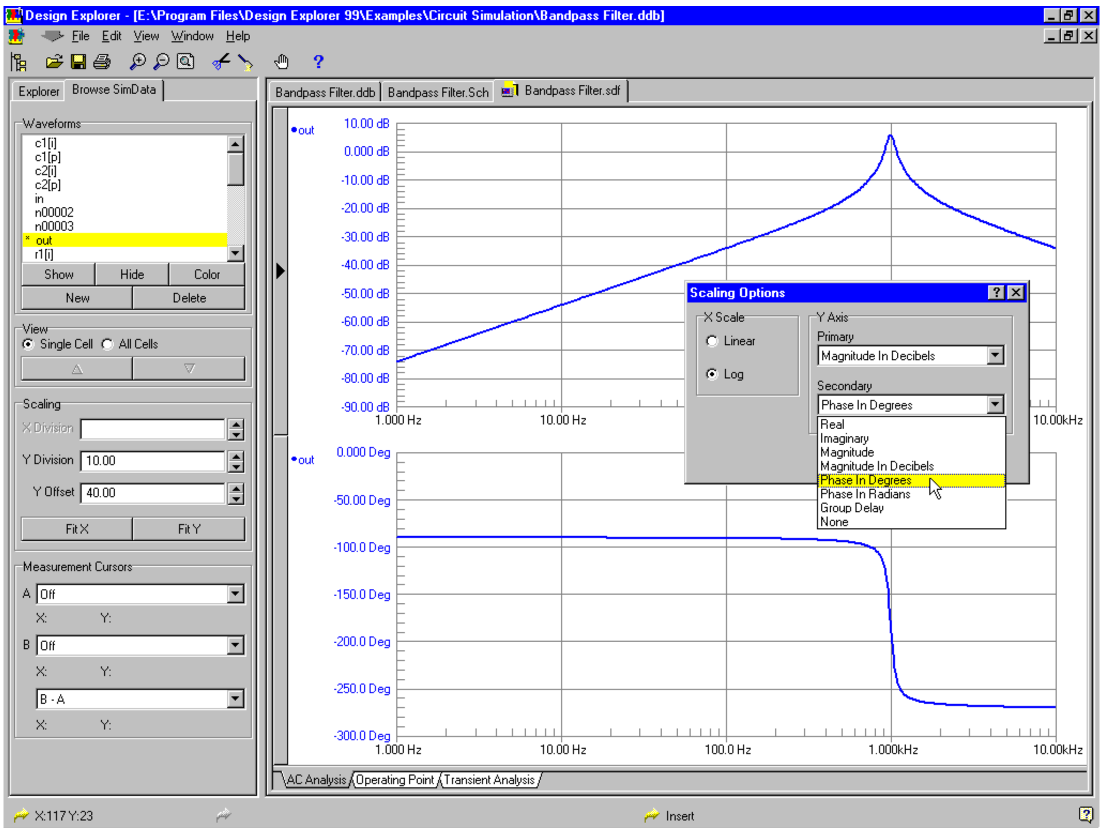
Operator/ Function	Description
+	Addition operator.
-	Subtraction operator.
*	Multiplication operator.
/	Division operator.
^	Power operator, y^x returns the value of "y raised to the power of x." Same as PWR(,).
( )	Precedence Indicators. Use to set precedence of math operations. Operations contained within ( ) will be performed first.
ABS( )	Absolute value function. ABS(x) returns the value of $\lvert x \rvert$.
ACOS( )	Arc cosine function.
ACOSH( )	Hyperbolic arc cosine function.
ASIN( )	Arc sine function.
ASINH( )	Hyperbolic arc sine function.
ATAN( )	Arc tangent function.
ATANH( )	Hyperbolic arc tangent function.
AVG( )	Average function. Returns the running average of the wave data.
BOOL(,)	Boolean function. In the expression BOOL(wave, thresh), wave would be the name of a waveform, thresh would be the switching threshold. Returns a value of one for wave arguments greater than or equal to thresh and a value of zero for wave arguments less than thresh.
COS( )	Cosine function.
COSH( )	Hyperbolic cosine function.
DER( )	Derivative function. dx/dt. Returns the slope between datapoints.
EXP( )	Exponential function. EXP(x) returns the value of "e raised to the power of x", where e is the base of the natural logarithms.
INT( )	Integral function. Returns the running total of the area under the curve.
LN( )	Natural logarithm function. Where LN(e) = 1.
LOG10( )	Log base 10 function.
LOG2( )	Log Base 2 function.
PWR(,)	Power function. Same as ^ operator. PWR(y,x) returns the value of "y raised to the power of x."
RMS( )	Root-Mean-Square function. Returns the running AC RMS value of the wave data.
SIN( )	Sine function.
SINH( )	Hyperbolic sine function.
SQRT( )	Square root function.
TAN( )	Tangent function.
TANH( )	Hyperbolic tangent function.
UNARY( )	Unary minus function. UNARY(x) returns -x.
URAMP( )	Unit ramp function. Integral of the unit step: for an input x, the value is zero if x is less than zero, or if x is greater than zero, the value is x.
USTEP( )	Unit step function. Returns a value of one for arguments greater than zero and a value of zero for arguments less than zero.

Simultaneous Waveform Display

Often you need to analyze and simultaneously display the same waveform scaled in different ways, for example frequency and phase, or frequency and group delay.

To do this first change the waveform display to single cell, then right-click and select **Scaling** from the floating menu. Set the primary Y axis scaling as required, then add the second scaling by setting the Secondary field as required. The single cell window will be split horizontally displaying the 2 versions of the waveform.

Measurement cursors can be added in the normal way, either through the panel, or by right-clicking on a waveform name in the window and selecting from the floating menu. There is also a selection region on the left of the waveform window, click on this to set the currently active view.



View the frequency and phase response at the same time