# Protel 99

SmartTeam TECHNOLOGY

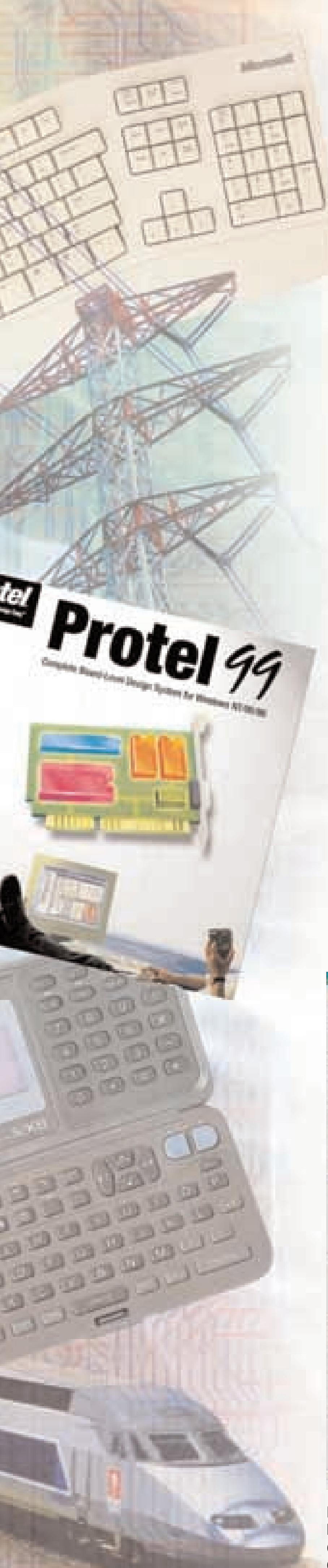
SmartDoc TECHNOLOGY

SmartTool TECHNOLOGY

Designed for Microsoft® Windows NT® Windows® 98

Protel®
Making Electronic Design Easy™

# Powerful rules-driven design accelerates




Bring a new approach to graphical printed circuit board design with Protel 99's superb PCB editor. The PCB editor combines manual and automated design in an interactive environment that supports both the new user and the experienced designer. The underlying PCB data structure is tuned for speed, and a set of powerful design rules gives you total control over the design parameters. Equipped with manual and interactive features that make it easy to layout and route the most complex mixed-technology designs, Protel 99's PCB editor is the perfect balance of power and control.

## Comprehensive design rules ensure your design meets the specs

PCB design can be a minefield of conflicting requirements. High-speed logic and smaller, more complex packaging technologies place extreme demands on the PCB designer. Net impedance, track clearances, signal reflections, path lengths, critical trace widths - all these factors and many more need to be carefully monitored and controlled to produce a successful board design. Protel 99's powerful rules-driven design approach leaves you free to focus on laying out and routing the board while your critical design requirements are automatically monitored and managed by the system.

Protel 99's PCB Editor incorporates over 25 design rule classes covering areas such as component placement, critical-net trace widths, crosstalk constraints, net impedance, signal reflections, clearances, object geometry, parallelism, routing priority and routing topology. Combined with the ability to create multiple rules, each applying to particular objects, nets, components, regions, or the whole board, Protel 99 gives you total control over the design process.

The ability to create compound rule scopes allows you to precisely target the objects a rule applies to. With simple-to-construct compound scope definitions you can route a net at one width on the top layer, and a different width on the bottom layer. Or close the solder mask over the vias on the bottom layer, but not the top layer.

## One-click PCB-to-schematic design synchronization

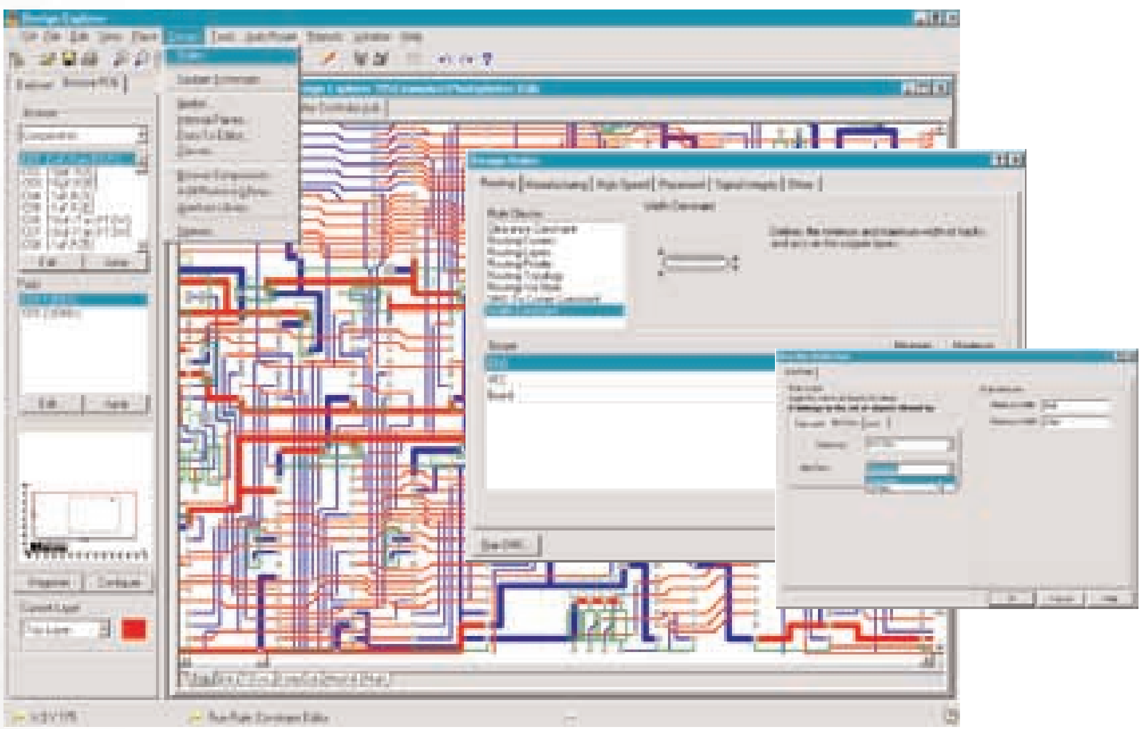
Design is not a simple linear process. Changes and updates are often made throughout the design cycle and it's all too easy for the final board layout to get out of step with the original schematics. With Protel 99's easy-to-use Design Synchronizer, keeping your schematic and PCB documents in complete harmony is a breeze. Reflecting PCB changes back to the schematic is as easy as pressing a button.

The Design Synchronizer analyzes the source PCB document and target schematic design to determine any differences. If you have changed component designators on your PCB for example, the designators on the schematic are automatically updated.

The Design Synchronizer can be run at any stage in the board layout process to ensure the total integrity of your design documentation. With Protel 99's Design Synchronizer technology you'll have complete confidence that your schematic and PCB documents represent the same circuit throughout the design process.

## Intuitive editing environment = ease of design

Protel 99's PCB editor is based on familiar Windows-based graphical editing techniques to make PCB design a natural and intuitive



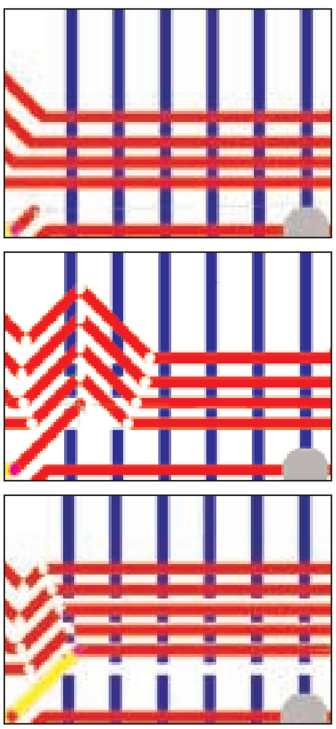
Protel 99's PCB Editor incorporates over 25 design rule classes covering areas such as routing, manufacturing, high-speed design, component placement, and signal integrity. Combined with the ability to create multiple rules, each with a different scope, Protel 99 gives you total control over the design process. The ability to create compound rule scopes allows you to precisely target the objects a rule applies to.

# board layout

process. Click-and-drag to move or resize graphical objects and double-click on an object to edit its properties. If you've ever used any Windows graphics editing packages you'll feel right at home with Protel 99. Whether you're a beginner or a power user, Protel 99's intuitive interface gives you easy control over a powerful set of PCB editing features.

Versatile global editing features let you apply property changes across any set of objects throughout your design. Double-click on a string on the overlay to change its height, for example, and you can instantly apply the height change to the single string, all strings on the overlay layer, all strings throughout the design, or any subset of strings you define. Global editing makes board-wide design changes easy.

And with Protel 99's Slider Hand, moving around your PCB design has never been easier. Simply click-and-hold the right mouse button anywhere in the PCB workspace and use the Slider Hand to 'slide' around your view of the PCB. Combined with automatic document panning and hotkey zoom control, the Slider Hand makes it easy to get the right view of your design.



Effortlessly route through tight spaces using the "push-and-shove" manual routing mode. Existing tracks are pushed out of the way in accordance with all relevant design rules.

## Unbreakable connectivity

Connectivity is the foundation of PCB design, and with Protel 99's PCB editor net integrity is paramount. Connectivity is continually monitored and analyzed as you route, with the status of each net constantly updated. As connections are completed, the corresponding ratsnest line is automatically erased. And because connectivity is monitored in "real time", you can route a net from any starting point – you're not constrained by the ratsnest connection order. The net connectivity will automatically be updated to reflect the current routing.

Also, Protel 99 monitors routing design rules as you work, instantly flagging design rule violations. With Protel 99's unbreakable connectivity and online design rule checking your routing will be fast, accurate and complete.

## Enhanced component placement tools

Component placement is the key to good board layout, and Protel 99's PCB editor is equipped with two powerful automatic placement tools, as well as a comprehensive set of intelligent interactive placement features.

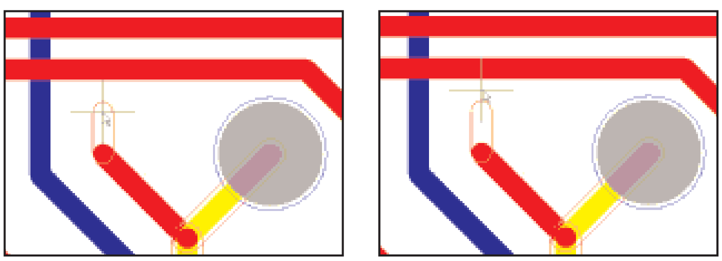
For boards with a low component count, choose Protel 99's Cluster Placer that uses a component clustering algorithm to automatically place components grouped first by connectivity (creating clusters of components), then by the component geometry.

With higher component counts, Protel 99's Statistical Placer uses an AI-based methodology called "simulated annealing". It analyzes the entire design during placement, considering the connection length, the connection density on the board, and the alignment of the components, all in accordance with the specified design rules.

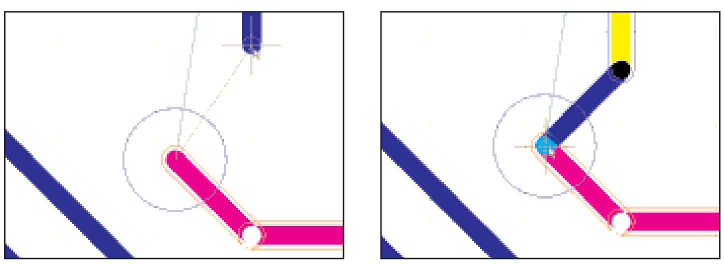
Smart interactive placement tools make short work of manual placement jobs. Use the Auto Select placement mode to quickly 'collect' components with similar footprints, then rotate, expand and contract the entire group as you move them to the required location on the board. Once your rough placement is complete, use Auto Align mode to neatly expand or contract a group of placed components with similar footprints. These and many other advanced interactive features make component placement in Protel 99 a designer's dream.

## Effortless interactive manual routing

A major challenge for today's PCB designer is working with a mix of imperial and metric technologies. Protel 99's powerful manual routing features let you easily route to any object on the board, regardless of its placement grid.



Protel 99's "avoid obstacle" manual routing mode automatically clips the track you are placing to avoid creating design rule violations. This makes it easy to "slam-and-jam" tracks together while maintaining defined clearances.

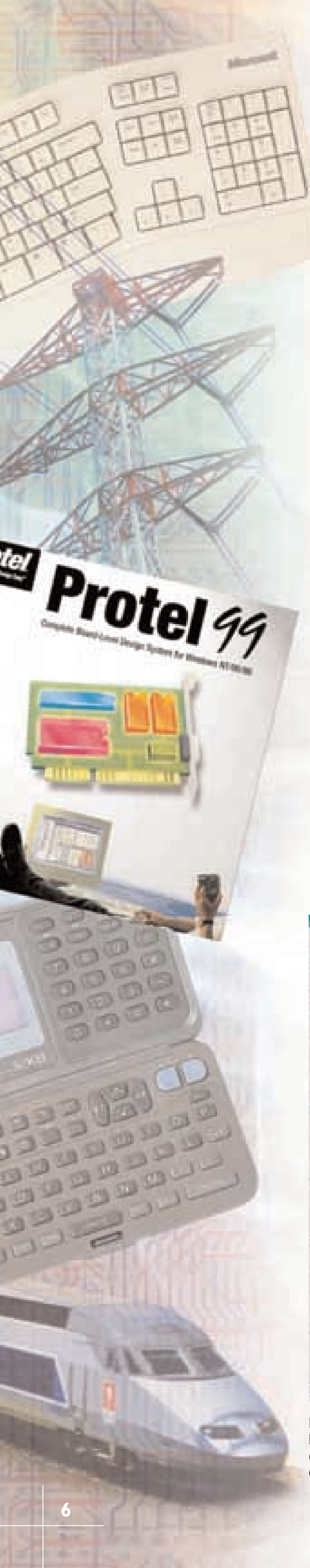


The PCB's special "electrical grid" feature snaps the cursor to valid electrical connection points, regardless of the current routing grid setting. This makes it easy to route to off-grid pads and tracks, enabling virtually gridless manual routing.

### *Feature Highlights*

- *Powerful rules-driven design environment with comprehensive design rule checking, including on-line DRC for monitoring rule violations as you work*
- *Compound, hierarchical design rule scopes that enable you to specifically target rules*
- *Direct "one-button" schematic-to-PCB and PCB-to-schematic design synchronization*
- *State-of-the-art manual and interactive routing functions, including "look ahead" track placement for easy angular and off-grid routing*
- *Intelligent software "robot" technology continually monitors and updates connectivity information as you route*
- *Sophisticated gridless manual routing, with auto-clipping and automatic "snapping" of the cursor to valid electrical hot spots*
- *Push-and-shove manual routing mode with full design rule adherence*
- *Intelligent polygon planes (copper shielding) facility with full DRC and automatic connection support*
- *Full split power plane support*
- *3D board visualization with automatic component modeling - no additional component height information required*
- *Automatic and interactive component placement features*
- *Powerful global editing features*
- *Bi-directional crossprobing to other documents, including schematics, reports and BOM's*

*Continued…*




# Powerful rules-driven design accelerates

Bring a new approach to graphical printed circuit board design with Protel 99's superb PCB editor. The PCB editor combines manual and automated design in an interactive environment that supports both the new user and the experienced designer. The underlying PCB data structure is tuned for speed, and a set of powerful design rules gives you total control over the design parameters. Equipped with manual and interactive features that make it easy to layout and route the most complex mixed-technology designs, Protel 99's PCB editor is the perfect balance of power and control.

## Comprehensive design rules ensure your design meets the specs

PCB design can be a minefield of conflicting requirements. High-speed logic and smaller, more complex packaging technologies place extreme demands on the PCB designer. Net impedance, track clearances, signal reflections, path lengths, critical trace widths - all these factors and many more need to be carefully monitored and controlled to produce a successful board design. Protel 99's powerful rules-driven design approach leaves you free to focus on laying out and routing the board while your critical design requirements are automatically monitored and managed by the system.

Protel 99's PCB Editor incorporates over 25 design rule classes covering areas such as component placement, critical-net trace widths, crosstalk constraints, net impedance, signal reflections, clearances, object geometry, parallelism, routing priority and routing topology. Combined with the ability to create multiple rules, each applying to particular objects, nets, components, regions, or the whole board, Protel 99 gives you total control over the design process.

The ability to create compound rule scopes allows you to precisely target the objects a rule applies to. With simple-to-construct compound scope definitions you can route a net at one width on the top layer, and a different width on the bottom layer. Or close the solder mask over the vias on the bottom layer, but not the top layer.

## One-click PCB-to-schematic design synchronization

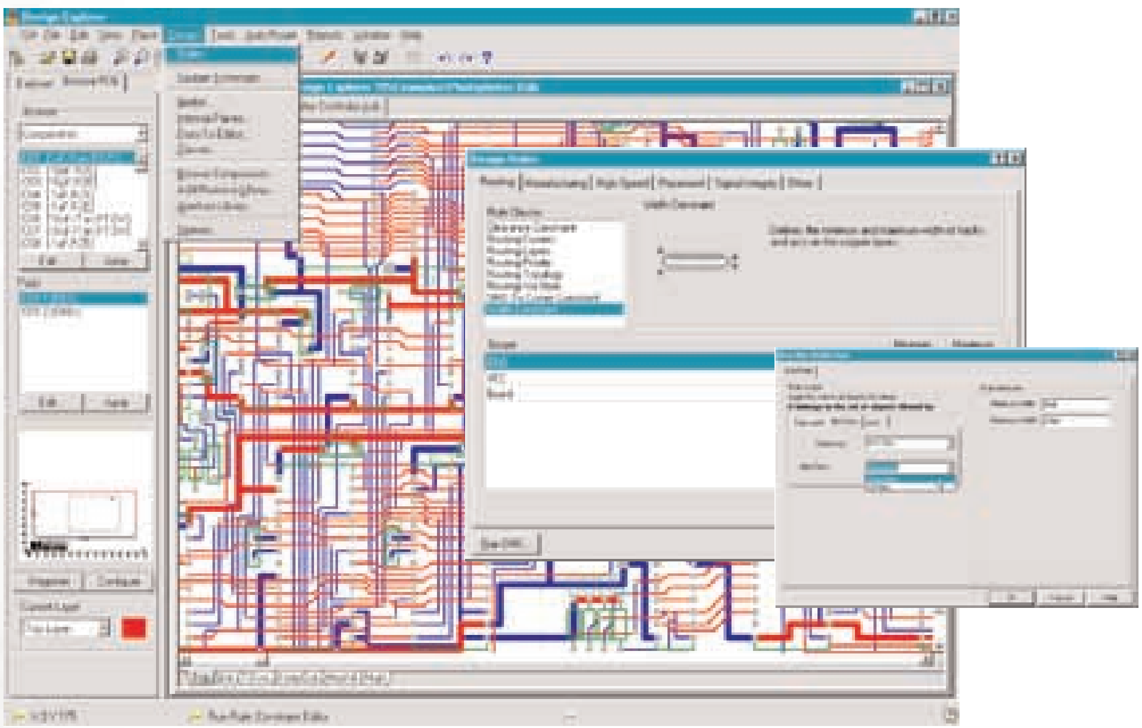
Design is not a simple linear process. Changes and updates are often made throughout the design cycle and it's all too easy for the final board layout to get out of step with the original schematics. With Protel 99's easy-to-use Design Synchronizer, keeping your schematic and PCB documents in complete harmony is a breeze. Reflecting PCB changes back to the schematic is as easy as pressing a button.

The Design Synchronizer analyzes the source PCB document and target schematic design to determine any differences. If you have changed component designators on your PCB for example, the designators on the schematic are automatically updated.

The Design Synchronizer can be run at any stage in the board layout process to ensure the total integrity of your design documentation. With Protel 99's Design Synchronizer technology you'll have complete confidence that your schematic and PCB documents represent the same circuit throughout the design process.

## Intuitive editing environment = ease of design

Protel 99's PCB editor is based on familiar Windows-based graphical editing techniques to make PCB design a natural and intuitive



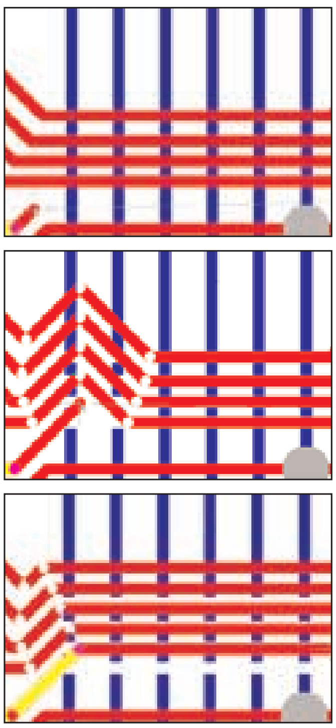
Protel 99's PCB Editor incorporates over 25 design rule classes covering areas such as routing, manufacturing, high-speed design, component placement, and signal integrity. Combined with the ability to create multiple rules, each with a different scope, Protel 99 gives you total control over the design process. The ability to create compound rule scopes allows you to precisely target the objects a rule applies to.

# board layout

process. Click-and-drag to move or resize graphical objects and double-click on an object to edit its properties. If you've ever used any Windows graphics editing packages you'll feel right at home with Protel 99. Whether you're a beginner or a power user, Protel 99's intuitive interface gives you easy control over a powerful set of PCB editing features.

Versatile global editing features let you apply property changes across any set of objects throughout your design. Double-click on a string on the overlay to change its height, for example, and you can instantly apply the height change to the single string, all strings on the overlay layer, all strings throughout the design, or any subset of strings you define. Global editing makes board-wide design changes easy.

And with Protel 99's Slider Hand, moving around your PCB design has never been easier. Simply click-and-hold the right mouse button anywhere in the PCB workspace and use the Slider Hand to 'slide' around your view of the PCB. Combined with automatic document panning and hotkey zoom control, the Slider Hand makes it easy to get the right view of your design.



Effortlessly route through tight spaces using the "push-and-shove" manual routing mode. Existing tracks are pushed out of the way in accordance with all relevant design rules.

## Unbreakable connectivity

Connectivity is the foundation of PCB design, and with Protel 99's PCB editor net integrity is paramount. Connectivity is continually monitored and analyzed as you route, with the status of each net constantly updated. As connections are completed, the corresponding ratsnest line is automatically erased. And because connectivity is monitored in "real time", you can route a net from any starting point – you're not constrained by the ratsnest connection order. The net connectivity will automatically be updated to reflect the current routing.

Also, Protel 99 monitors routing design rules as you work, instantly flagging design rule violations. With Protel 99's unbreakable connectivity and online design rule checking your routing will be fast, accurate and complete.

## Enhanced component placement tools

Component placement is the key to good board layout, and Protel 99's PCB editor is equipped with two powerful automatic placement tools, as well as a comprehensive set of intelligent interactive placement features.

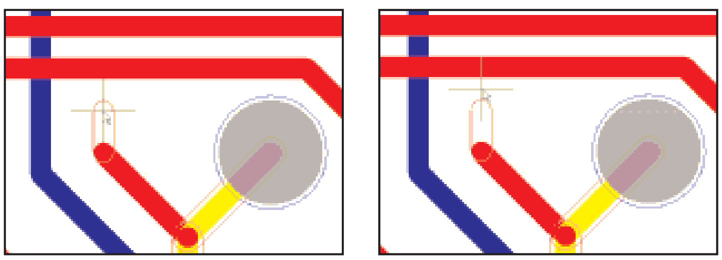
For boards with a low component count, choose Protel 99's Cluster Placer that uses a component clustering algorithm to automatically place components grouped first by connectivity (creating clusters of components), then by the component geometry.

With higher component counts, Protel 99's Statistical Placer uses an AI-based methodology called "simulated annealing". It analyzes the entire design during placement, considering the connection length, the connection density on the board, and the alignment of the components, all in accordance with the specified design rules.

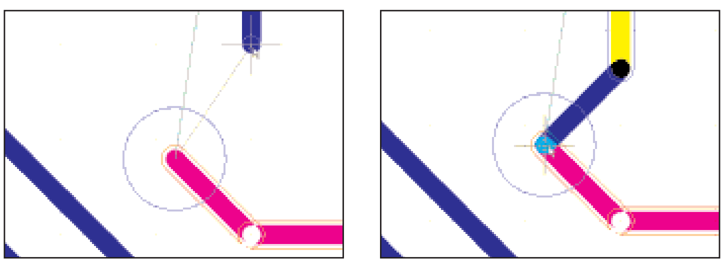
Smart interactive placement tools make short work of manual placement jobs. Use the Auto Select placement mode to quickly 'collect' components with similar footprints, then rotate, expand and contract the entire group as you move them to the required location on the board. Once your rough placement is complete, use Auto Align mode to neatly expand or contract a group of placed components with similar footprints. These and many other advanced interactive features make component placement in Protel 99 a designer's dream.

## Effortless interactive manual routing

A major challenge for today's PCB designer is working with a mix of imperial and metric technologies. Protel 99's powerful manual routing features let you easily route to any object on the board, regardless of its placement grid.



Protel 99's "avoid obstacle" manual routing mode automatically clips the track you are placing to avoid creating design rule violations. This makes it easy to "slam-and-jam" tracks together while maintaining defined clearances.

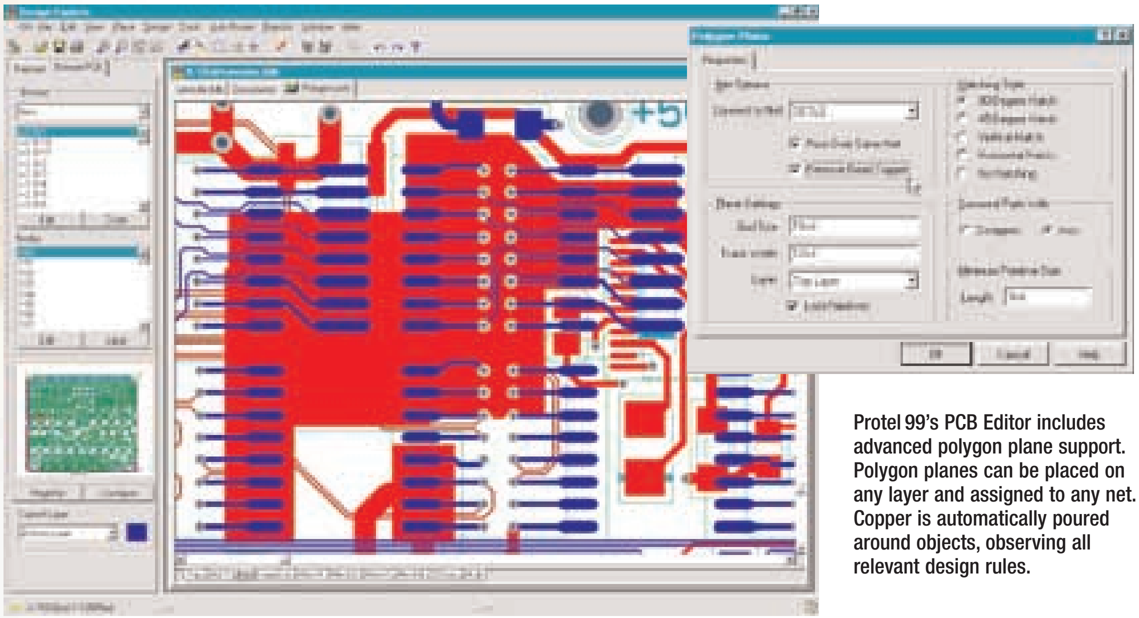


The PCB's special "electrical grid" feature snaps the cursor to valid electrical connection points, regardless of the current routing grid setting. This makes it easy to route to off-grid pads and tracks, enabling virtually gridless manual routing.

### *Feature Highlights*

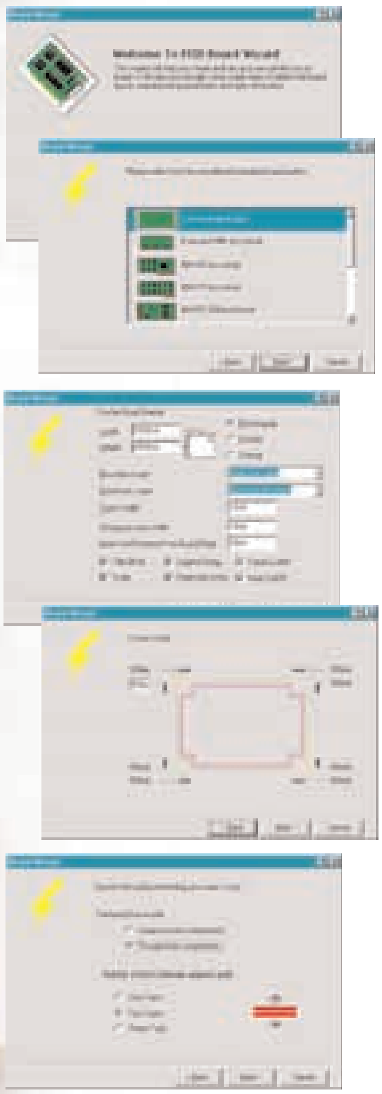
- *Powerful rules-driven design environment with comprehensive design rule checking, including on-line DRC for monitoring rule violations as you work*
- *Compound, hierarchical design rule scopes that enable you to specifically target rules*
- *Direct "one-button" schematic-to-PCB and PCB-to-schematic design synchronization*
- *State-of-the-art manual and interactive routing functions, including "look ahead" track placement for easy angular and off-grid routing*
- *Intelligent software "robot" technology continually monitors and updates connectivity information as you route*
- *Sophisticated gridless manual routing, with auto-clipping and automatic "snapping" of the cursor to valid electrical hot spots*
- *Push-and-shove manual routing mode with full design rule adherence*
- *Intelligent polygon planes (copper shielding) facility with full DRC and automatic connection support*
- *Full split power plane support*
- *3D board visualization with automatic component modeling - no additional component height information required*
- *Automatic and interactive component placement features*
- *Powerful global editing features*
- *Bi-directional crossprobing to other documents, including schematics, reports and BOM's*

*Continued...*

Protel 99's PCB Editor includes advanced polygon plane support. Polygon planes can be placed on any layer and assigned to any net. Copper is automatically poured around objects, observing all relevant design rules.

Protel 99's Board Wizard guides you through the process of creating a new board outline. Select from and customize one of the many industry standard board templates, or create your own custom templates. The Board Wizard lets you specify a title block, number layers, via types, routing technology, default track sizes and clearances, and many other board parameters. The Board Wizard then creates a new PCB document complete with board outline and default design rules.



Protel 99's configurable Electrical Grid snaps tracks to the center of electrical hot spots on placed objects as you route. This, combined with Protel's Predictive Track Placement feature, makes accurate routing to off-grid components effortless.

Protel 99's "slam-and-jam" (Avoid Obstacle) manual routing mode automatically clips tracks as you route to prevent design rule violations. In this mode you can easily route dense board areas, achieving minimal allowable clearances for all objects regardless of grid setting. And to get through those tight spots, Protel 99's new Push-and-Shove routing mode intelligently moves existing tracks to make way for a new track, while preserving design rule compliance.

For the ultimate in rerouting and cleanup, Protel 99's Loop Removal feature automatically and intelligently removes redundant track segments as you work. Simply reroute any portion of an existing track and Protel 99 will automatically delete the unnecessary segments.

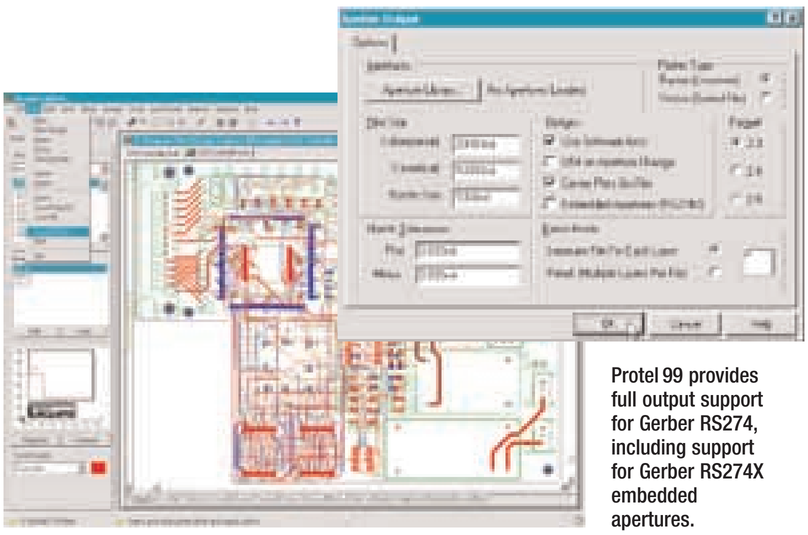
## Seamless autorouter integration

When it comes to autorouter integration, Protel 99's PCB editor sets the standards. There's no configuration file editing or netlist import/export, simply click a button and Protel 99's advanced shape-based autorouter goes to work directly on the PCB layout.

Because the autorouter works directly in the PCB editing window, you can stop the autorouter at any time and instantly work manually on your design. Protel 99's PCB editor makes the dream of one-button autorouting a reality.

## Expert Wizards at your fingertips

Protel 99's PCB Editor includes a number of intelligent design Wizards that step you through common design tasks. For example, use the Board Wizard to select from and customize one of the many industry standard board templates, or create



Protel 99 provides full output support for Gerber RS274, including support for Gerber RS274X embedded apertures.

your own custom templates. The Board Wizard lets you specify a title block, number layers, via types, routing technology, default track sizes and clearances, and many other board parameters. The Board Wizard then creates a new PCB document complete with board outline and default design rules.

Building components is a breeze with the Component Wizard. With a few simple steps you can create any component from a two-pin surface mount resistor, right through to a Pin Grid Array with several hundred pins. The Component Wizard guides you through every step in the component creation process.

## Easy component footprint editing and management

Protel 99 comes with a comprehensive array of PCB footprint libraries, and new and updated libraries are available for direct download from Protel's website at *www.protel.com*. With Protel 99, you'll have all the component footprints you need at your fingertips.

Protel 99 also includes integrated component editing and library management tools that make it easy to create and organize your own custom libraries. You can have any number of libraries open at the same time, and easily browse components in open libraries.

## Intelligent polygon planes

Protel 99's PCB editor takes the work out of generating polygon planes. Solid or latticed polygon planes can be placed on any layer, with optional automatic connection to a specified net. Polygon shapes can be defined using line and arc perimeters, and vertices can be moved, added or deleted after generation. Copper 'pours' automatically, observing all rules and wrapping around all placed objects, including arcs and non-orthogonal shapes. Polygons can be re-poured around new obstacles and you can redefine the connected pad relief styles at any time.

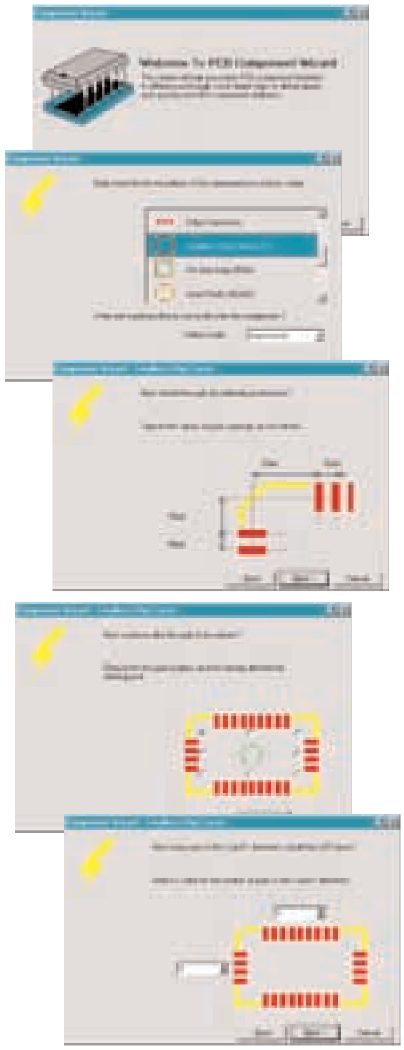
## Bulletproof design verification

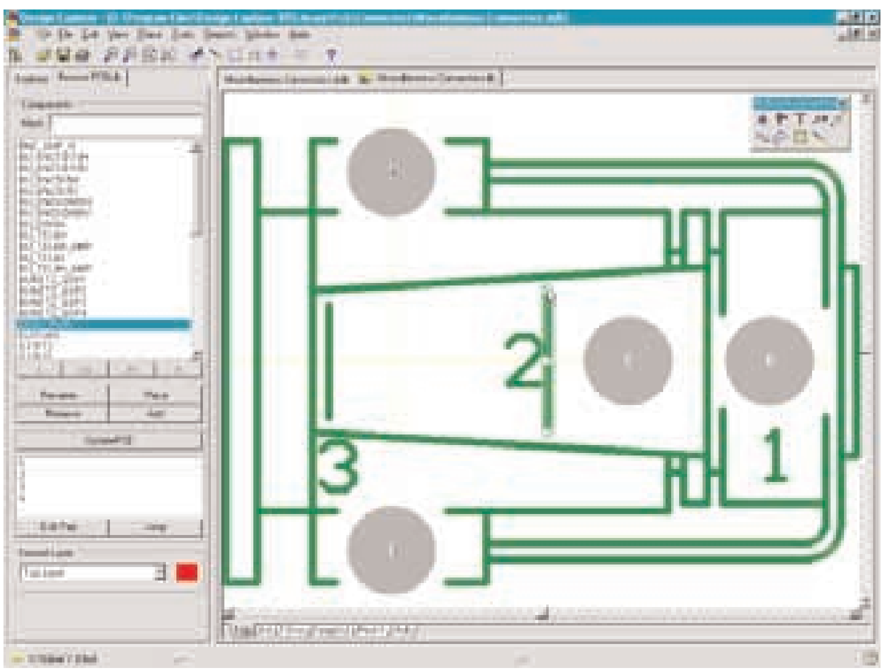
Make sure your board performs the way you expect with Protel 99's comprehensive design rule checking and reporting features. The batch-mode Design Rule Checker (DRC) thoroughly analyzes your design in respect of all specified design rules. A comprehensive DRC report flags any design rule violations. Rule violations are also highlighted directly on the board to allow you to easily focus on problem areas.

The DRC links directly to Protel 99's Signal Integrity Analyzer, giving true verification of crosstalk, reflection, impedance and other signal integrity-related design rules. Any violations of these rules can be further investigated using Protel 99's integrated Signal Integrity Simulator, which provides comprehensive crosstalk and impedance analyses, and termination simulation.

With Protel 99's design verification features you can be confident in your board's performance before it leaves your desktop.

**Protel 99's Component Wizard generates new PCB component footprints in no time. Choose from a comprehensive range of component types and customize the important parameters of your footprint in a set of easy-to-follow steps. With Protel 99's Component Wizard, you'll never be without a crucial footprint again.**

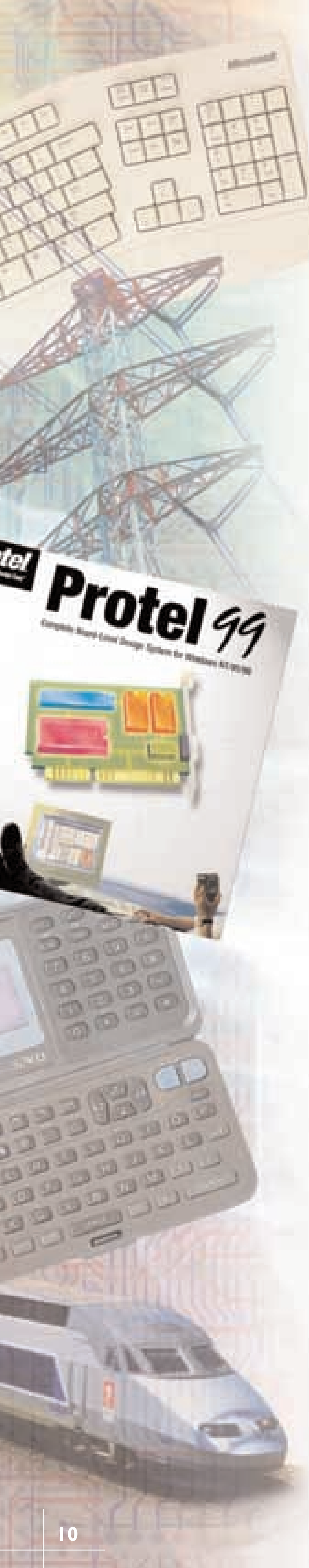




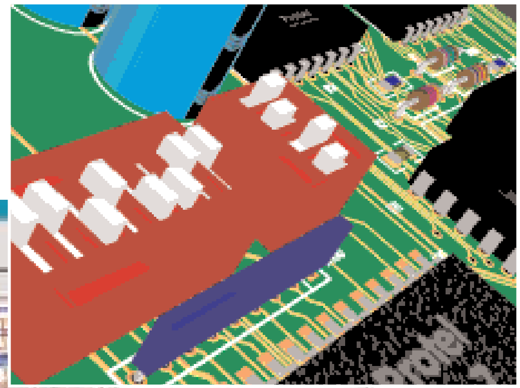

**Quickly and easily create your own custom component footprints in Protel 99's powerful PCB Library Editor. Using familiar PCB design tools you can create footprint designs of any complexity, allowing you to quickly build custom component libraries. With Protel 99, you'll never be stuck without the right footprint.**

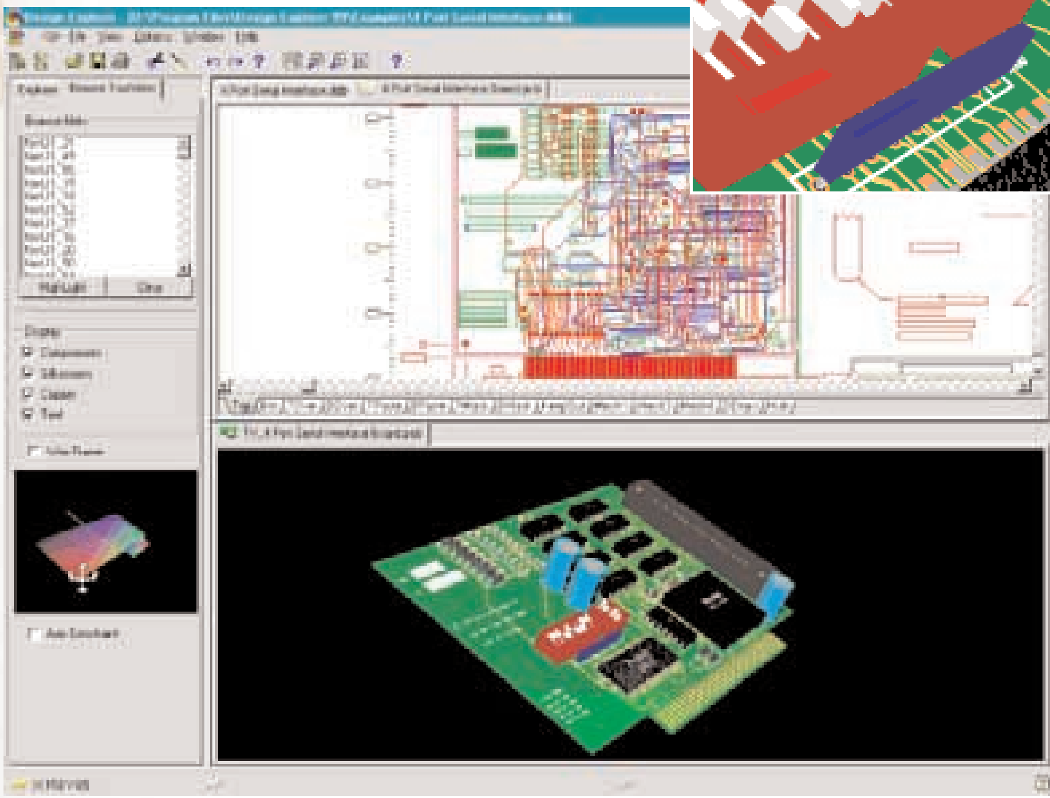
## *Feature Highlights (continued)*

- *32-bit design database, providing a resolution of 0.001 mil in a 100x100 inch workspace*
- *Full component rotation with an angular resolution of 0.001 degrees*
- *16 signal layers, 4 power planes and 4 mechanical layers*
- *Board creation Wizard with industry-standard board templates, and the ability to define your own board outline templates*
- *Multiple library support with no limit on the number of libraries open at any one time*
- *Fully-featured footprint library editor with component creation Wizard*
- *Automatic Gerber and NC drill file generation*
- *Import and export DXF and AutoCAD DWG format files up to and including R14*
- *Full support for Windows printer and plotter drivers*
- *Unlimited Undo and Redo*
- *Automatic positional component re-annotation*
- *Easy switching between imperial and metric units*
- *Export design objects to spreadsheet for direct property editing*
- *Fully-customizable toolbars, menus and shortcut keys*
- *Free hole size editor add-on available*
- *Free testpoint finder add-on available*
- *Free overlapping hole detector add-on available*



Protel 99's unique 3D visualization feature lets you see your finished board before it leaves your desktop. Sophisticated 3D modeling and extrusion techniques render your board in stunning 3D without the need for additional height information. Rotate and zoom to examine every aspect of your board, and show or hide components, copper, silkscreen or text.





## 3D board visualization mode

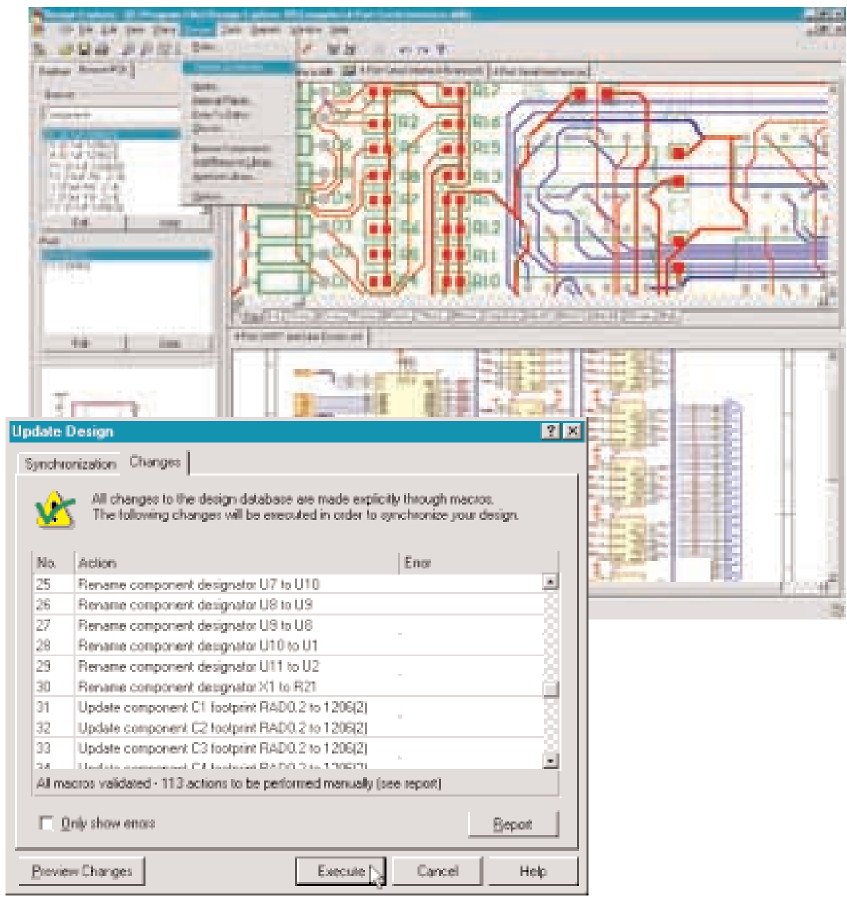
Why wait for your board to be manufactured to see what it looks like? Protel 99 includes sophisticated 3D board visualization technology that lets you preview your assembled board design as a stunning 3D image. Run-time component modeling techniques automatically extrude and render the components on your board without the need to define additional height information. Simply select the 3D option from the PCB View menu and your board will be instantly brought to life.

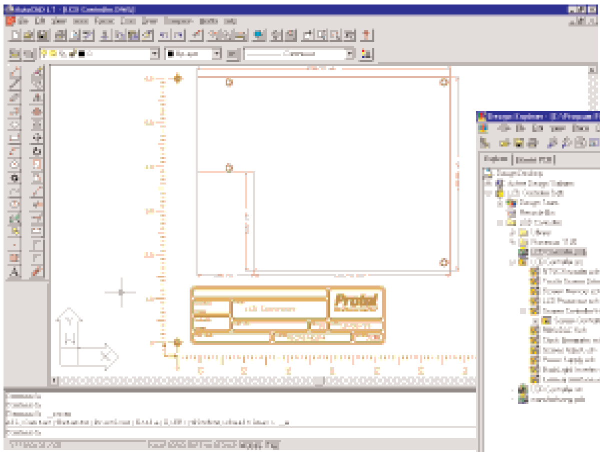
You can render your board populated, or hide the components for a bare board view – very useful for finding text over holes. You can also show or hide the copper, text and silkscreen layers, and even highlight individual nets. You have full rotational and zoom control over the 3D image, so you can view your board from any angle.

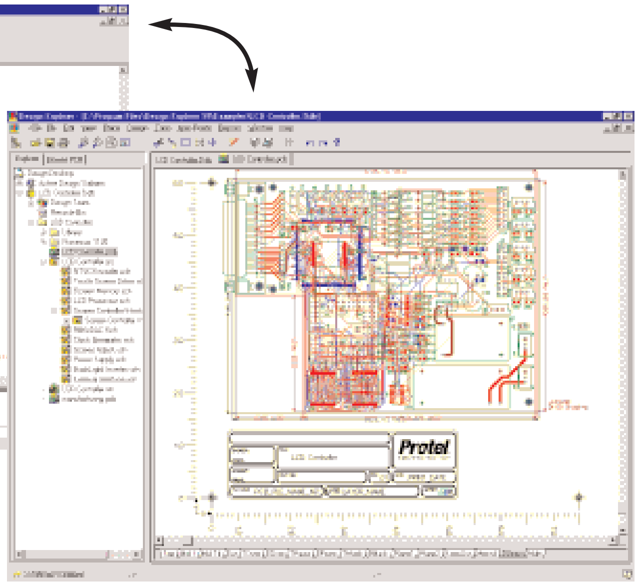
For unknown or custom board elements, Protel 99's 3D viewer includes a simple 3D component modeling language that allows you to easily define an object's extrusion parameters.

Add a new dimension to your board design with Protel 99's 3D visualization technology.

Reflecting modifications made on the PCB, such as designator and footprint changes, is a simple one-step operation with Protel 99's Design Synchronizer. No messing with "Was-Is" files, simply run the Synchronizer at any time in the design process to make sure your schematic and PCB documents are in perfect harmony.







Effortlessly interface from Protel 99's PCB Editor to AutoCAD drawings. Whether you want to import a mechanical outline drawing to use as the board boundary in Protel, or export your Protel board to AutoCAD, Protel 99's AutoCAD translator makes it easy. The translator imports/exports both DXF and DWG files, as well as supporting user-defined layer mapping and PCB component – AutoCAD block transforms.

## Versatile output generation

Protel 99 provides full support for industry-standard Gerber RS274 and NC drill file generation to give maximum manufacturer compatibility. Load manufacturer-specific aperture files, or let Protel generate an aperture table from your current PCB design. Protel supports embedded apertures using the Gerber RS274X standard.

And to complete your design documentation, Protel 99 supports all Windows printer/plotter drivers. Print or plot your board layer-by-layer, or create composite print outs of all or selected layers.

Protel 99 leads the field for fast, efficient, intelligent desktop PCB design.

## Bi-directional AutoCAD interface

Need to give your mechanical designers a board outline in AutoCAD format? No problem with Protel 99's bi-directional AutoCAD interface. You can import or export both DXF and native DWG files directly from Protel 99's PCB Editor, making mechanical design easier. The interface supports all AutoCAD versions up to Release 14. User-definable layer mapping allows you to easily relate PCB and AutoCAD layers, and there is full PCB component – AutoCAD block conversion support. There is also support for ungrouping components during translation and displaying rounded track ends in AutoCAD.

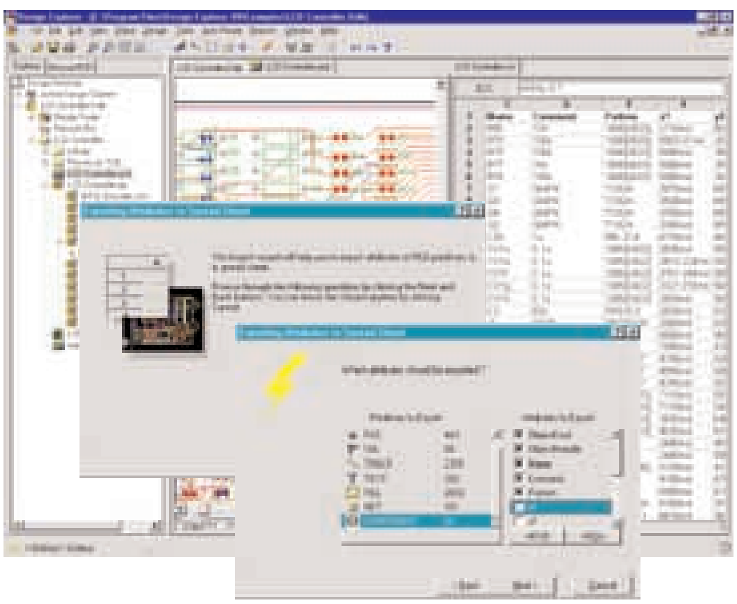
Protel 99's powerful AutoCAD translator takes the work out of mechanical design issues.

## Spreadsheet export for fast PCB object editing

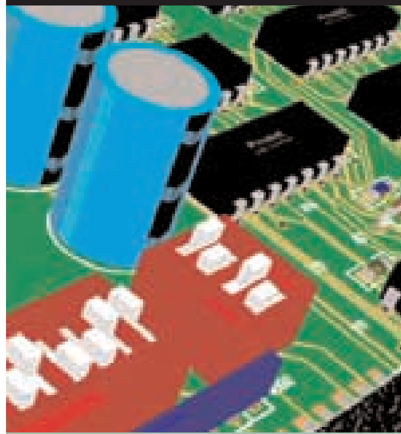
Protel 99 includes a powerful PCB - spreadsheet editing feature that gives you a different perspective on your design. An easy-to-use Spreadsheet Export Wizard guides you through the process of exporting selected PCB object types. All objects of the types defined, along with their properties, are then displayed in Protel 99's integrated Excel-compatible spreadsheet editor.

Use powerful spreadsheet editing functions to view and edit the properties of your PCB objects. Search and replace component designators, reassign polygon layers – edit virtually any property of any object on your board. When you're finished simply update the PCB from the spreadsheet and your changes are incorporated into the PCB document.

Protel 99's versatile spreadsheet export feature gives you fine control over your design objects and allows you to view your design in an entirely different way. It's just one more feature that makes PCB design with Protel 99 easy.



Easily export PCB design objects to Protel 99's Excel-compatible spreadsheet editor using the PCB spreadsheet Wizard. Design object properties can be viewed and edited using powerful spreadsheet functions and features, and any changes used to update the PCB document. Protel 99's PCB spreadsheet export feature gives you a whole new perspective on your design.



## Specifications

***34 Layers:*** *16 x Signal; 4 x Mechanical; 4 x Internal Plane; 2 x Solder Mask; 2 x Paste Mask; 2 x Silkscreen; 2 x Drill (Drill Guide & Drill Drawing); 1 x Keep Out; 1 x Multi-Layer (spans all signal layers)*
***Max. board size:*** *100 x 100 inches*
***Max. Resolution:*** *0.001 mil linear displacement; 0.001o angular rotation*
***Import File Formats:*** *Netlist (Protel and Protel 2 formats); AutoTrax; DOS PCB 3; AutoCAD DXF; Gerber - batch or single; Protel PCB 2.8 - ASCII and binary*
***Export File Formats:*** *Netlist (Protel format); AutoCAD DXF; HyperLynx; Protel PCB 2.8 - ASCII*
***Output Device Support:*** *Gerber RS274 (internal driver, supports embedded apertures); HPGL Plotter (internal driver); all Windows printer & plotter drivers*
***Measurement Units:*** *Metric and Imperial*
***Max. Net Count:*** *Unlimited*
***Report Generation:*** *Board Information Summary; Selected Pins; Bill Of Materials; Design Hierarchy; Netlist Status; Signal Integrity; NC Drill (Excellon binary and ASCII); Pick and Place; Create Netlist (from copper)*
***Pads Styles:*** *Round; Rectangular; Octagonal*
***Pad Size:*** *0.001 to 99999 mils*
***Pad Stacks:*** *Different shapes definable for top, bottom and mid layers*
***Via Styles:*** *Through-hole; Blind & Buried (Layer Pairs); Blind & Buried (Any Layers)*
***Track Widths:*** *0.001 to 99999 mils*
***Track Placement Modes:*** *Oblique; $45^{o}$; $45^{o}$ with arcs; $90^{o}$; $90^{o}$ with arcs*
***Interactive Routing Modes:*** *Ignore Obstacles; Avoid Obstacles; Push-and-Shove*
***Polygon Plane Styles:*** *$90^{o}$ hatched; $45^{o}$ hatched; Vertical hatch; Horizontal hatch; Solid*
***Plane Connectivity:*** *Assignable to any net*
***Power Planes:*** *All plane layers can be assigned to any net. All plane layers can have multiple splits*
***Max. Components Per Board:*** *Unlimited*
***Max. Pins Per Component:*** *Unlimited*
***Max. Components Per Library:*** *Unlimited*
***Max. Simultaneously Open Libraries:*** *Unlimited*