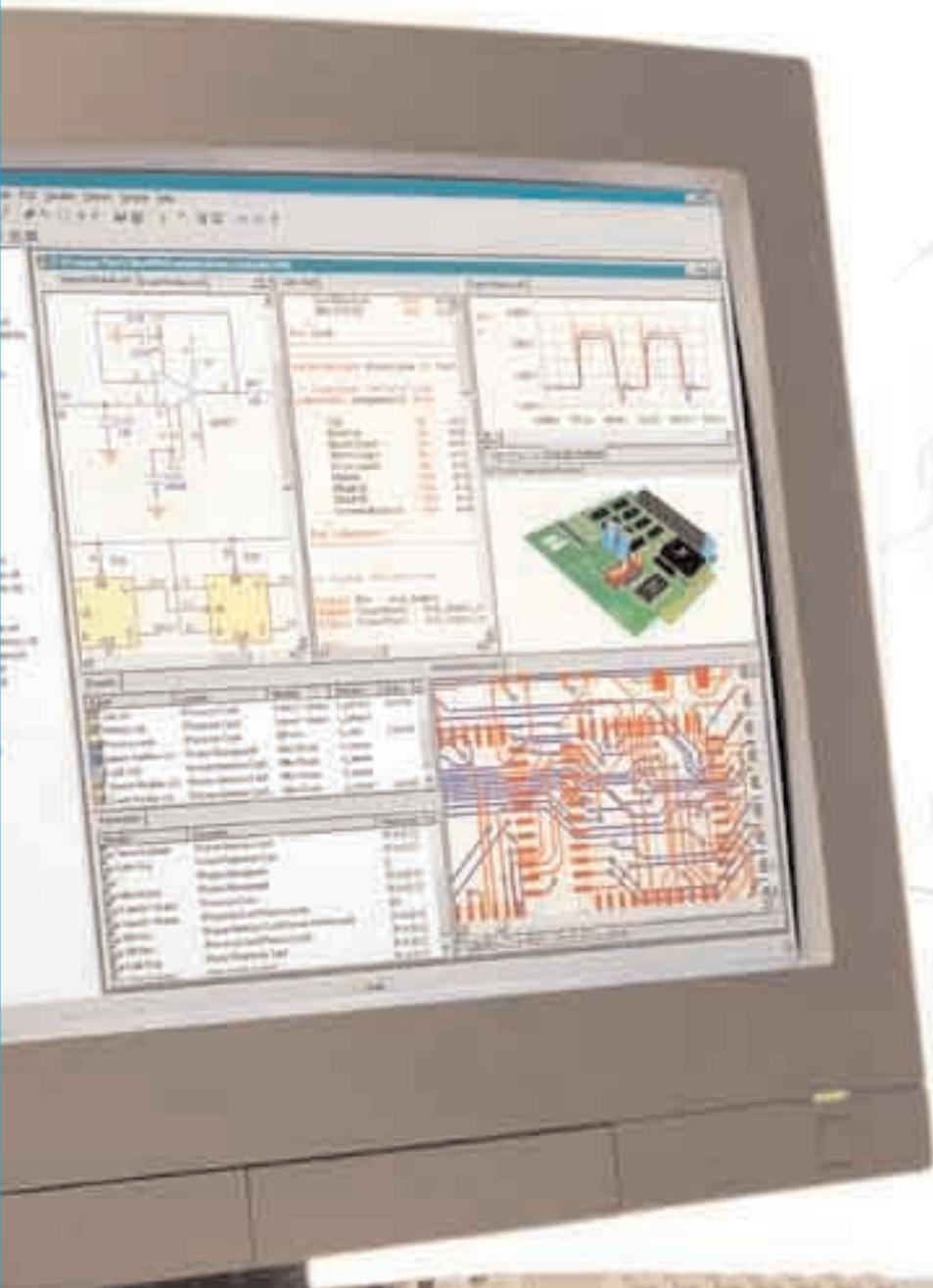


# Protel 99



**Protel**  
Making Electronic Design Easy™

# Schematic Capture. Start your design

Protel's superb schematic editor provides fast and smart multi-sheet hierarchical schematic entry with high quality output. Choose from over 60,000 supplied schematic symbols, representing the widest range of component manufacturers and categories available, and build on these symbol libraries with regular updates from Protel's website. Wire components together using the intelligent manual and automatic wiring tools then validate the design with Protel's powerful electrical rules checker. Capturing your design has never been easier.

## Fully-hierarchical design environment with advanced project management

Protel 99's schematic editor is easy to use, yet capable of managing even the largest design. There is no limit to the number of sheets, or the depth of design hierarchy. With Protel 99's schematic editor you can design anything from a simple single-sheet light dimmer, through to a 200-sheet, multi-level, complex hierarchy, 128-channel audio mixer.

Regardless of your preferred design strategy, Protel 99 puts you in the driving seat. Use standard block-diagram methodology – create and wire the top-level block diagram and Protel will automatically create the necessary sub-sheets, complete with ports, ready for you to place and wire the components. If you prefer a “bottom-up” approach, simply create the separate sections of the design, then automatically create the sheet symbols on the top sheet, complete and ready to wire. Whichever way you like to work, Protel 99's schematic editor will work with you to make the schematic capture process easy.

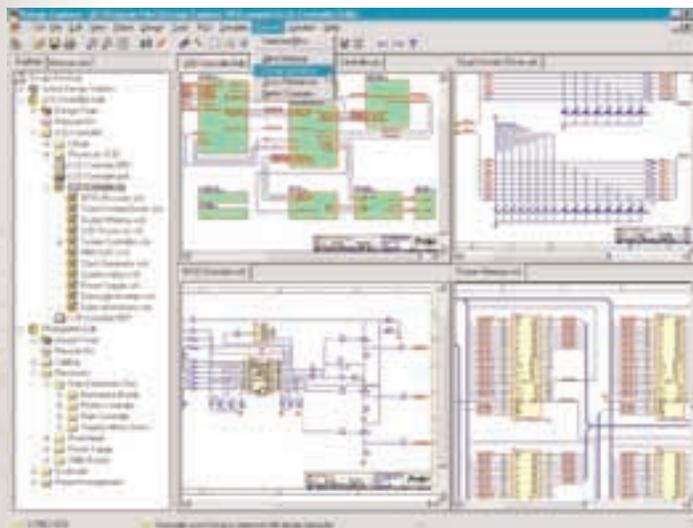
And with the Protel 99 Design Explorer navigation tree you can examine the entire structure of your circuit at a glance. Any sheet in your design is just a mouse click away. For finer control use Protel 99's powerful browsing and hierarchy navigation tools to jump to any design object or trace any signal throughout the entire design.

## Superior component and library management features

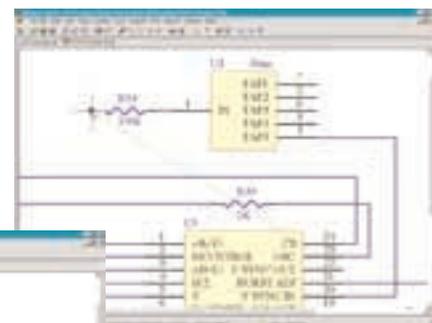
The heart of any design is the components and Protel 99 comes with one of the most comprehensive library sets available, as well as a powerful library editor that makes creating and managing your own component libraries a breeze. Create professional components using familiar graphical drawing tools and techniques, including IEEE notation. Move and copy components freely between libraries to easily set up a library structure that suits your needs. And with Protel 99's SmartTeam technology you can easily control access to your company libraries, defining the read/write privileges of each team member.

With such an extensive array of components at your fingertips you'll appreciate Protel 99's powerful library search capabilities, which allow you to search for components by name or description. With full support for wildcards and the ability to search all libraries in a single directory or sub-directories, a defined path, your entire hard disk, or across the company network, you'll quickly locate the components you need.

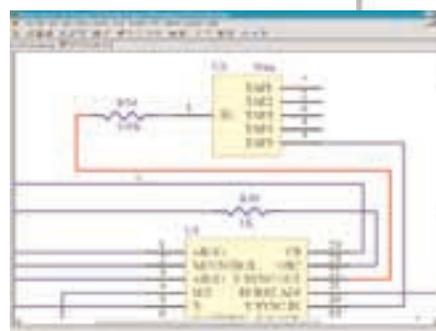
Protel 99's comprehensive libraries and powerful library management features help you build up the heart of your design quickly and efficiently.



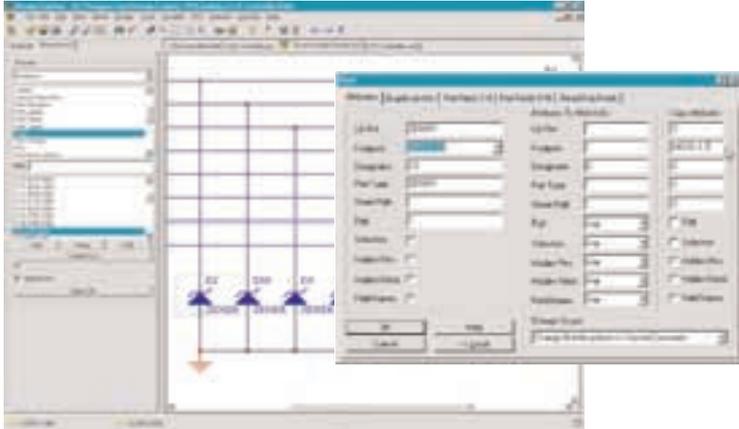
Protel's hierarchical schematic design features let you take a “bottom up” or “top down” approach, depending on your preferred methodology. Protel can automatically generate sub-sheets based on higher-level sheet symbols, or create sheet symbols based on existing sheets.



Protel's schematic auto-wiring mode makes short work of point-to-point wiring jobs. Simply click on the start and end points and the wire is automatically routed.



# with a competitive edge ...



Protel's powerful global editing features let you easily apply property changes to defined object groups on the current sheet, or across your entire schematic project.

## Easy-to-use interactive design tools

Making the right connections is easy with Protel 99's Guided Wiring and Auto-Junction features. As you wire your schematic, active electrical "hot spots" guide you to valid connection points, allowing you to accurately wire at lower zoom levels. Because you are guided to the right point each time you make a connection you'll spend less time tracing faulty wiring in your finished schematic. And there's no need to terminate a wire at each connection point. Wire across multiple component pins or the end points of existing wires and Protel 99 is smart enough to insert the connection junctions automatically.

Protel 99's Autowiring mode makes point-and-click wiring a reality. Choose your start and end points and the autowirer does the rest, plotting a suitable wiring path to make the connection.

## Powerful editing capabilities

Protel 99's schematic editor uses standard "point-and-click" and "click-and-drag" graphical editing techniques that give intuitive control over the schematic editing process. Quickly craft your design using familiar move, drag, copy and paste actions. Regardless of the complexity of your design, you'll feel right at home and be in total control throughout the entire design capture process.

Need to change the PCB footprint for all 245 resistors in your 15-sheet schematic? No problem with Protel 99's powerful global editing capabilities. Global editing is as easy as editing one component, then applying that change to a defined group. You can perform complex changes, and precisely control which objects are to be changed. You can perform global edits on virtually every type of

schematic design object, making multiple design changes simple and quick. Once you've experienced the power of Protel 99's global editing features you will wonder how you ever got by without them.

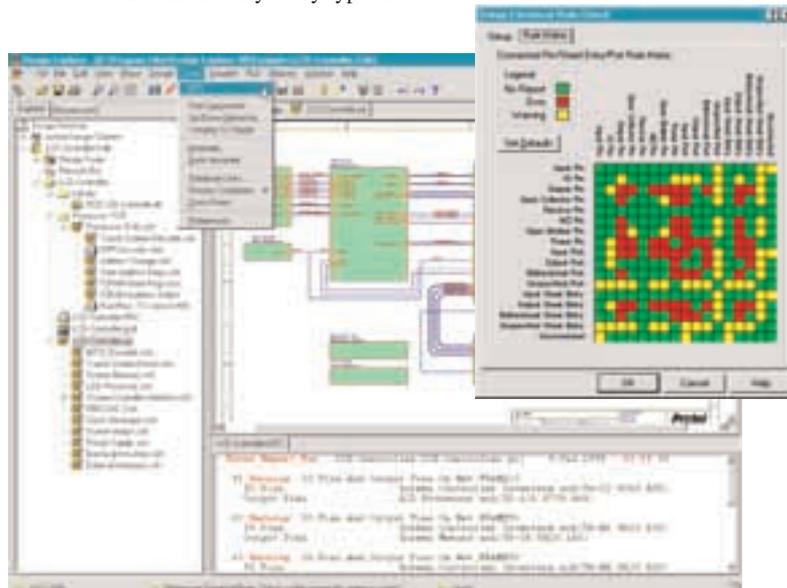
Of course it's a designer's prerogative to change his or her mind, and Protel 99's user-definable multi-level undo/redo feature lets you change your mind as often as you like. You can quickly roll back to any point in the editing process, even if you have saved your work in the meantime.

Protel 99's intuitive editing features let you focus on the real task at hand - design.

## Design verification

Make sure your schematic is accurate before starting the board layout. Protel 99's comprehensive Electrical Rules Checker (ERC) analyzes your schematic for common drafting errors, as well as invalid electrical connections. Conditions such as unconnected power objects, floating input pins, and output pins connected to input ports are flagged for your attention, and a unique electrical rule matrix makes it easy to change and define relevant electrical warning and error conditions. The ERC works on single sheets or across entire projects, with problems highlighted directly on the schematic and detailed in a comprehensive ERC report.

Have total confidence in your design with Protel 99's ERC.



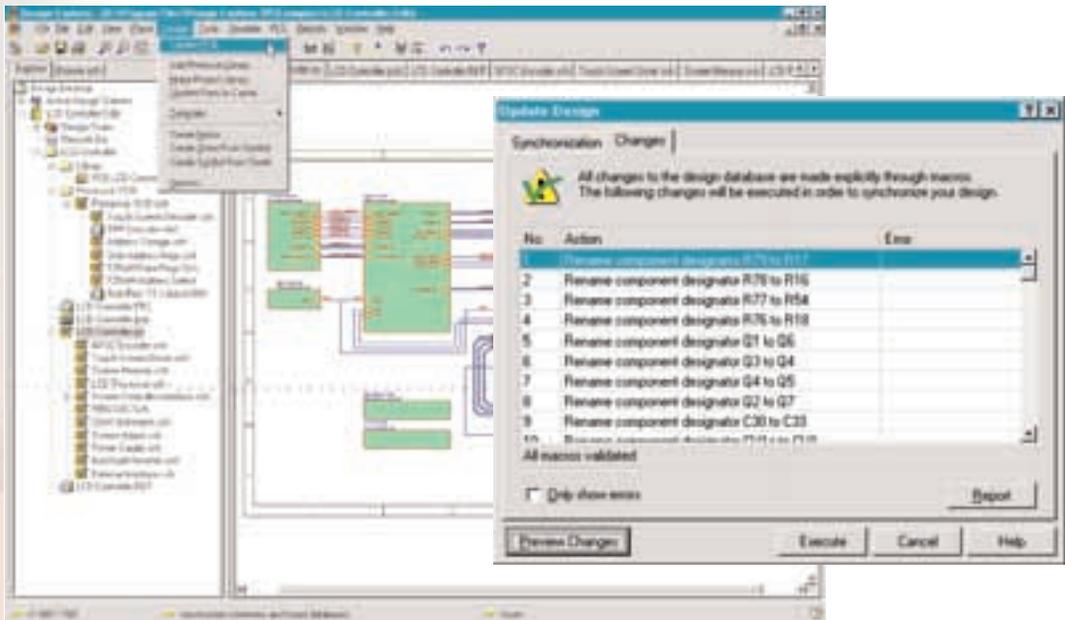
Find errors in your design fast with Protel's Electrical Rules Checker (ERC). As well as checking for common drafting errors, the ERC verifies the electrical integrity of your schematic project against definable electrical error conditions.



## Feature Highlights

- Fully-hierarchical design environment with advanced project management tools
- Supports single sheet, multiple sheet, and fully hierarchical designs including complex hierarchy
- Comprehensive component libraries with over 60,000 components
- Simultaneous multi-user library access for network installations
- Powerful library searching feature
- Standard and user-specified sheet templates supporting various image file formats
- Comprehensive design verification tools
- Powerful schematic-to-PCB and PCB-to-schematic design synchronization
- Mixed-mode simulation directly from the schematic
- Intelligent interactive guided wiring with auto-junction
- Powerful cross probing to other design documents including PCB, text reports and spreadsheets
- Dynamic linking to manufacturing component databases
- Powerful global editing capabilities - globally apply changes across the sheet, or the whole project
- Intuitive manual editing tools with flexible selection, WYSIWYG graphical editing, and easy alignment tools
- Multilevel Undo and Redo
- Fast, efficient Bill of Materials (BOM) report generation
- Comprehensive file, netlist and output format options including EDIF 2.0 and EDIF 2.0 Hierarchical
- High quality output with full Windows font support for printing and plotting
- Direct OrCAD V7.x DSN import
- Direct AutoCAD DXF & DWG import/export

Contact your local sales office today for your FREE Protel 99 30-Day Trial CD.



Protel's Design Synchronizer makes it easy to keep your schematic and PCB documents in step. You can run the Synchronizer at any time to pass design changes from the schematic to the PCB, and vice-versa.

### Direct interface to simulation and programmable logic design

Protel 99's schematic editor brings more muscle to the design capture process with seamless integration of circuit simulation and PLD design. There's no need to export your design to another application to perform mixed-mode simulations, or to design a PLD functional block. Simulation and PLD design menus and toolbars form part of the schematic capture environment.

Run mixed-mode simulations directly from your schematic at the touch of a button. Protel's simulation engine provides full SPICE 3f5/XSPICE support, as well as integrated digital component modeling using the powerful Digital SimCode language, an extension of the XSPICE standard. Protel 99 comes supplied with comprehensive "simulation-ready" schematic component libraries that let you accurately model your circuit's behavior. And Protel's simulator is easy to use, regardless of your level of simulation experience. There's no need to delve into cryptic text files to configure a simulation run – all simulation parameters are set up using a single dialog box.

Integrating programmable logic devices into your design has never been easier. Protel 99 comes supplied with an extensive PLD symbol library that lets you design your PLD as a standard schematic. Compile directly from the schematic to produce all the files necessary to

program and test the device. Protel's PLD compiler also supports direct PLD design using the industry-standard CUPL description language. With universal device support and seamless schematic integration, Protel 99 opens up the world of programmable logic to all designers.

### Tight integration with Protel 99's PCB editor

Netlisting woes are a thing of the past with Protel 99. At the touch of a button Protel's Design Synchronizer transfers your schematic information to the board design. There's no netlist import/export to deal with and you'll never have to contend with daunting text files.

During the initial transfer to the PCB design stage, the Design Synchronizer automatically adds all necessary footprints and connectivity information to the PCB document, warning you of any common transfer problems such as missing libraries. Components are arranged neatly to the side of your board area ready for placement.

After the initial transfer, the Design Synchronizer can be run at any time during the board layout process to ensure complete synchronization of your design files. Add components or modify component values in the schematic and the Design Synchronizer will automatically reflect these changes on the PCB. Similarly, designer changes made on the PCB layout can easily be fed back to the schematic.

With Protel 99, keeping your schematic and PCB documents in synchronization is as easy as pressing a button.

### Comprehensive crossprobing

There's no need to play hide-and-seek when tracing signals in Protel 99. Protel's SmartTool technology supports advanced crossprobing between editors. Select a net name on the schematic sheet using the Crossprobe tool and the corresponding net will be highlighted on the PCB. Highlight a component pin in the ERC report and the Crossprobe tool presents that component on the schematic sheet. Tracing between design documents couldn't be easier.

With Protel 99's powerful crossprobe features you can concentrate on your analysis, not which document you should look in.

### Hot-link components to your company database

Schematic components have 16 user-definable property fields that can be used to contain static text or hot-linked to fields in an external database. With this feature you can set up direct links to your company stock databases to extract company-specific part numbers, descriptions, stock status, etc.

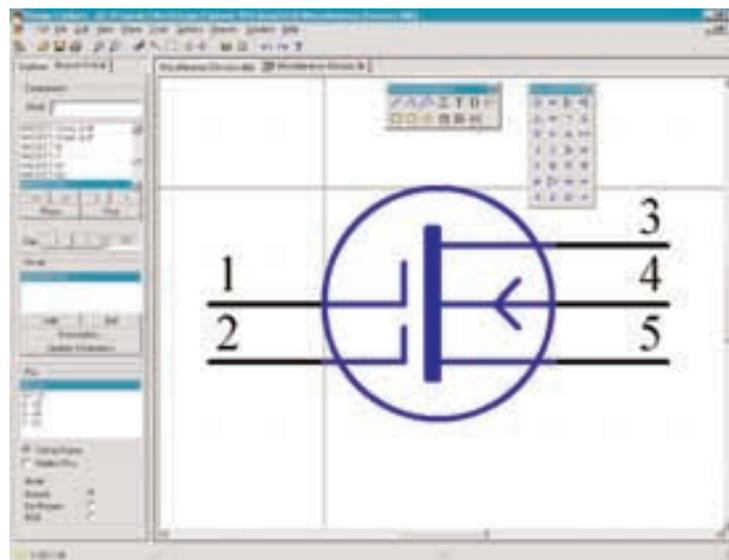
When you select an external database, Protel 99 reads in the available field names and allows you to define selection match criteria and linked fields from simple pick lists.

You could, for example, link Part Field 1 of all components to the supplier field of your company database based on the schematic symbol's Part Type. Then, when you generate a bill of materials for your design, you can automatically include the current supplier's name for each part.

Linked fields can be updated manually at any time, or set to automatically update at a selected time interval.

## Fast, efficient Bill Of Materials report generation

Creating a Bill of Materials (BOM) for your design is a common task that can take an infuriating amount of time and effort. Protel 99 makes the task easy with its versatile "BOM Wizard". Run the BOM Wizard to create comprehensive reports in just a few easy steps. Output the result to Protel 99's internal Excel-compatible spreadsheet editor, or create a file in .CSV format for use with an external spreadsheet. Alternatively, create the Bill Of Materials directly as a text file.



Protel 99's easy-to-use schematic library editor gives you full control of your components. Edit existing schematic libraries, or quickly create your own custom components.

You can create a BOM for a single sheet or an entire project and select the component properties fields you want to include in the report, including fields linked to an external database. The BOM Wizard takes care of the rest, creating and formatting your report for the selected output type.

## Define your own custom sheet templates

Customizing schematic sheets with title blocks, borders, graphics, etc, is easy with Protel 99's schematic template feature. You can use Protel 99's schematic drawing and graphics tools to create schematic sheet templates that can be used to apply custom elements to any schematic sheets.

Schematic templates can contain special text strings that link to document-specific information, such as company name, sheet number, sheet name, revision, etc. Include these special strings in your template and they will automatically insert the relevant information on your schematic.

You can change the active template for any schematic sheet at any time, allowing you to print the same schematic using different title blocks, border styles, etc. You can also set the default template to be used whenever you create a new schematic sheet, making it easy to maintain a consistent look and feel across all designs. Make a change to a template, like updating your company logo, and you can easily update any sheet based on that template.

## Get high quality results

For quality output, Protel 99's schematic editor just can't be beaten. With full support for TrueType fonts and all Windows printer and plotter drivers, Protel 99 makes producing detailed, professional-quality schematics as easy as... well, as easy as capturing your design with Protel 99.



## Specifications

**Max Sheet Size:** 64 x 64 inches

**Max. Sheet Resolution:** 0.01 inches

**Max. Sheets Per Project:** Unlimited

**Sheet hierarchy:** Unlimited depth

**Font support:**

All Windows-supported fonts

**Output Device Support:**

All Windows output devices

**Netlist Output Formats:**

Protel; Protel 2; Protel

(hierarchical); EEsof (Libra);

EEsof (Touchstone); Edif 2.0;

Algorex; AppliconBRAVO;

AppliconLEAP; Cadnetix;

Calay; Case; CBDS;

ComputerVision; EEDesigner;

FutureNet; Star Semiconductor;

HiLo; Intergraph; Mentor

Board Station 6; MultiWire;

OrCAD/PCB II; OrCAD PLD;

PADS ASCII; PCAD; PCADnlt;

RacalRedac; Scicards; SPICE;

SMASH; Tango; Telesis;

Vectron; WireList;

Xilinx XNF 5.0; VHDL

**Max. Components Per Sheet:**

Unlimited

**Max. Pins Per Component:**

Unlimited

**Max. Components Per Library:**

Unlimited

**Max. Simultaneously Open**

**Libraries:** Unlimited

**Electrical Drawing Tools:**

Bus; Bus Entry; Component

Part; Junction; Power Port;

Wire; Net Label; Port; Sheet

Symbol; Sheet Entry

**Non-Electrical Drawing tools:**

Annotation; Text Box; Arc;

Elliptical Arc; Ellipse; Pie

Chart; Line; Rectangle;

Rounded Rectangle; Polygon;

4-point Bezier; Graphic Image

**User-definable Component**

**Properties:** 16 general-purpose

text fields defined from the

schematic sheet; 8 additional

read-only text fields defined in

the schematic library editor

**Assignable Component Pin**

**Electrical Types:** Input; Output;

Input/Output; Open Collector;

Passive; Hi Z; Open Emitter;

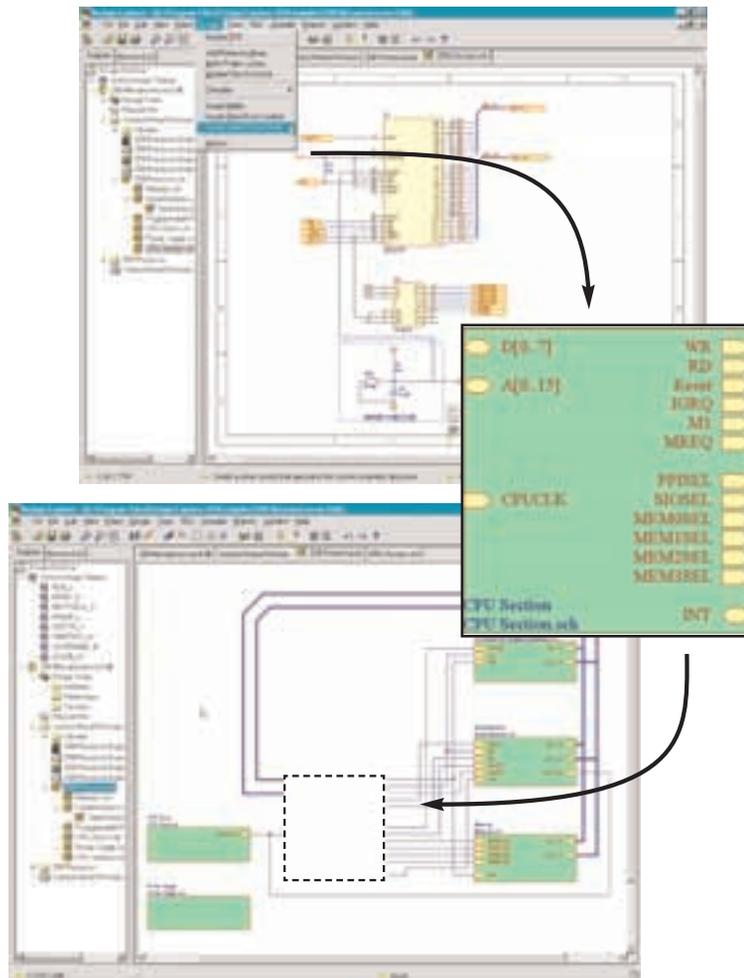
Power

**Report Generation:** Selected

Pins; Bill Of Materials; Design

Hierarchy; Cross Reference;

Netlist Compare



Protel 99's simple schematic hierarchy tools make it easy to create multi-sheet design to any level of complexity. You can automatically create sheet symbols based on existing schematic sheets, or generate a blank sheet based on an existing sheet symbol, complete with all defined IO ports.

Contact your local sales office today for your FREE Protel 99 30-Day Trial CD.