Protel *99 SE* **Trial Version**

Discover how Protel 99 SE increases your board design productivity

SmartDoc
TECHNOLOGY

SmartTeam
TECHNOLOGY

SmartTool
TECHNOLOGY

Designed for
Microsoft®
Windows NT®
Windows® 98

LATEST
SERVICE PACKS
AVAILABLE AT
www.protel.com

Protel®

Making Electronic Design Easy™

Contents

What is Protel 99 SE?

Welcome to Protel 99 SE — a complete board design system for Windows 95/98/NT4. Protel 99 SE provides an integrated design environment that includes a full suite of advanced design entry and PCB layout tools, integrated design document management, and support for design team collaboration across a network. Key features of Protel 99 SE include:

- Full 32-bit code optimized for Windows NT4/9x
- SmartTool technology integrates all design tools under a single environment
- SmartDoc technology and Protel's unique Design Explorer platform stores your integrated design within a single design database, or as stand-alone files under the Windows File System
- SmartTeam technology allows multiple designers to safely work on the same design across a network, with team management facilities built into Design Explorer
- Integrated schematic capture, including libraries containing over 60,000 symbols
- Integrated mixed-mode simulation directly from a schematic sheet, with full SPICE 3f5 support
- Integrated schematic or CUPL-based programmable logic design with universal device support
- Integrated rules-driven PCB design with automatic on-line design rule checking
- Up to 32 signal, 16 plane and 16 mechanical layers with definable layer stack and drill pairs
- Integrated shape-based PCB autorouting that produces routing results equivalent to that of an experienced board designer, even on complex boards
- Dual auto-placement technologies for both simple and complex designs, with enhanced interactive placement modes

- Enhanced interactive routing modes, including "push-and-shove" routing
- Enhanced PCB design rules set, including signal integrity checking
- Export/import of design rule sets
- Board signal integrity checking detects potential impedance, propagation delay and overshoot problems, and allows full reflection and crosstalk analyses to be performed
- Comprehensive manufacturing output management - produce Gerber files, NC Drill files, BOMs, DRC reports, Pick-&-Place files and Testpoint reports with the click of a button
- Advanced PCB printing with full print previewing
- 3D board viewer with automatic component extrusion and modeling
- Full AutoCAD® DXF and DWG file import and export up to R14
- OrCAD® Capture V7 and V9, and Layout V9 import
- Full "one button" schematic-PCB design synchronization
- Comprehensive component libraries, with continual updates available from the Protel Library Development Center
- Create new Schematic sheets using standard or user-defined templates, with support for customized title blocks
- Choose from over 60 industry-standard PC board outlines, or create customized board templates
- Integrated Wizards guide you through complex tasks, such as setting up a PCB outline or generating a Bill of Materials from a schematic
- Protel 99's open architecture allows third-party developers to integrate servers into the environment, with full API access to the schematic and PCB document editors
- Integrated Macro programming support using the Client Basic programming language

How to install Protel 99 SE

1. Insert the Protel 99 SE Trial CD into your CD ROM drive. On most systems the setup program will automatically start shortly after the CD drive tray is closed.

2. If the installation program window does not appear automatically, run the **Setup.exe** program in the **Protel 99 SE** directory of the installation CD.

Once the setup program initializes, a Wizard will guide you through the installation process.

Note: If you are installing Protel 99 SE under Windows NT, you will need local Administrator privileges to successfully complete the installation.

After installation, visit the Downloads section of the Protel website at www.protel.com to download and install the latest Protel 99 SE service pack.

System requirements

Minimum requirements:
- Pentium-class PC running Windows NT4/95/98
- 32MB RAM
- Display: 1024 x 768 resolution, 256 colors
- 200MB free hard disk space

Recommended system:
- Pentium II 300MHz or better PC running Windows NT4
- 64MB RAM
- Display: 1024 x 768 resolution, 16-bit color
- 300MB free hard disk space for full install

Starting Protel 99 SE

To start Protel 99 SE, select the **Protel 99 SE** icon in the **Programs » Protel 99 SE** folder of your Windows Start menu.

Notes on the trial version

The trial CD includes a full working version of Protel 99 SE, time-limited for a period of exactly 30 days from the first time the software is run. Re-installing the software will not increase the length of the trial period.

Warning: The trial version of Protel 99 SE can only be installed once on a particular computer. If you attempt to extend the trial period by changing the system clock, re-installing the software or moving it to a different directory, you will render the trial version unusable. If anything goes wrong during the installation do not uninstall, simply reinstall to the same directory.

Defragmenting your hard disk: If you defragment your hard disk with Norton's SpeedDisk, it must be configured to make the following file types unmovable — *.ENT, *.KEY and *.RST. Failure to do this will render the trial version unusable.

Notes for existing Protel users

Installing Protel 99 SE will not affect existing Protel installations. Protel 99 SE will run concurrently with previous versions of Protel software, including Protel 99.

Protel 99 SE uses a database structure to store design documents. Data storage can be in the form of a single Access design database, or as stand-alone files under the Windows File System. Existing Protel 99, Protel 98 and v3 files can be directly imported into a Protel 99 SE design and edited. Documents exported from a Protel 99 SE design will be editable in Protel 99, Protel 98 and v3 products, but some advanced features may be lost.

To open a Protel 99 SE PCB document in previous versions you must first save the file in **PCB 3.0 Binary File** format (select **File » Save As** from the PCB menu).

Please note: As with Protel 99, Protel 99 SE's simulator is not compatible with Protel 98 or earlier simulation files. Protel 99 simulation files, however, will run under Protel 99 SE without modification.

How to access Protel on-line

Protel – Making Electronic Design Easy

At Protel we recognize the importance of immediate access to resources and information. That's why we strive to make our website a valuable resource for the entire Protel community. From catching up on the latest Protel news and searching for technical information, through to downloading the latest library symbols, it's all at your fingertips at **www.protel.com**.

Secure On-line Ordering

If you would like to order a fully-licensed copy of Protel 99 SE on-line, visit the Protel website for product pricing and distributor information, as well as a secure on-line ordering service.

Extensive Library Support

Protel 99 SE comes supplied with comprehensive component libraries, and new libraries are continually being developed by the Protel Library Development Center. Visit **www.protel.com** to download the latest libraries free-of-charge.

Comprehensive Technical Resources

Need an answer to a technical question at 2.00am? Visit the Support pages at www.protel.com to get the latest technical information. Read Articles, check out What's New, or browse through the FAQs. You can also download Add-Ons and macros, get the latest service packs, read Protel's valuable "Tip Of The Week", or find out about the latest products from one of the affiliated third-party developers.

About this booklet

This booklet has been designed to show you some of the productivity-enhancing features of Protel 99 SE. Each section is based around the example files supplied and installed with the Protel 99 SE Trial version. As you go through each example we would encourage you to examine and explore the various dialogs and menus you encounter. While this booklet points out some of the main options and features it is by no means an exhaustive look at all that Protel 99 SE has to offer.

After reading through this booklet we invite you to continue your exploration of Protel 99 SE by using the example files in conjunction with Protel 99 SE's comprehensive help system to examine more of Protel 99 SE's powerful features, such as the 3D PCB Viewer and the Signal Integrity simulator.

Also, visit the support section of the Protel website at www.protel.com where you will find information about any current issues to do with Protel 99 SE, as well as downloadable add-ons, pre-release software, and tutorials that can be used with your Trial version.

1 **Design Manager.** This panel displays the Design Explorer navigation tree, which gives a Windows Explorer-like hierarchical view of open design databases and the documents they contain. Click the **+/−** icons to expand/contract subfolders. Click on a document or folder to open or focus it in the main design window. Right-click on a document or folder to popup a menu of options. Also displayed in the Design Manager panel are specific document editor panels which appear as tabs when editing the various document types.

2 **Active Design Stations.** Click the **+** symbol next to this icon to show a list of all computers on the network that currently have Protel 99 SE running.

3 **Design Team.** This is a special folder within each design database that allows you to configure and display information about the document sharing properties for the design.

4 **Design Window.** Each open design database has an associated design window used to display open design documents and folders.

5 **Design Tabs.** Open documents and folders are shown as tabs within the design window. Click on a tab to make that document or folder active. Right-click on a tab to popup a menu of options, including the ability to split the window to display multiple tabs simultaneously.

6 **Design Explorer Menu.** Clicking the arrow head in the left corner of the menu bar activates the Design Explorer menu, which gives various system options.

7 **Help Advisor.** Click this icon to get help by simply typing in your questions. Protel 99 SE's Natural Language Query system analyzes your question and presents a list of relevant help topics.

Useful shortcuts

The following keyboard shortcuts can be used to navigate the Design Explorer:

Keys	Action
Ctrl+TAB	Cycle through open documents in the design window.
Ctrl+F4	Close the currently active document in the design window.

When you are working on a schematic or PCB document in Protel 99 SE, use the following shortcuts to change your view:

Keys	Action
V, D	View entire document.
V, F	Zoom to fit all objects on the document.
PgUp	Zoom in centered on the cursor position.
PgDn	Zoom out centered on the cursor position.
End	Redraw the screen.
Esc	Terminates the current action.
Ctrl	Hold the Ctrl key down when opening a design to stop the automatic document opening procedure.

Use the following mouse controls on objects in schematic or PCB documents:

Mouse	Action
Single-Click	Focuses an object. Editing handles appear, allowing you to change the object's shape.
Shift-Click	Selects or deselects an object. Selected objects can be cut, copied, deleted, etc. Shift-Click is cumulative, ie multiple objects can be selected.
Double-Click	Opens an object's properties dialog for editing.
Right-Click	Terminates the current action.

Understanding the editing cursors

When you activate any process in Protel 99 SE which acts on an object or document, the cursor will change to a crosshair to allow accurate positioning. In general, once you have entered an editing mode you will remain in that mode until you right-click or press the Esc key to return to the normal Windows cursor. This allows you to repeat actions, such as placing multiple objects, without having to re-invoke the process.

Also, Protel 99 SE is a "re-entrant" editing environment — you can launch a task while in the middle of performing another. For example, while placing an object you can change your view using the shortcut keys without leaving placement mode.

Getting help

Protel 99 SE includes a natural language query system, the Advisor, that allows you to get help by asking questions in "plain English". To open the help Advisor, click the yellow question mark icon on the right-hand side of the status bar. Type your question into the Advisor interface and press the Search button to search for relevant topics in Protel 99 SE's main help system. Click on a topic in the list to open the main help window at that topic. The Advisor interface will remain open, allowing you to easily select other topics in the search results list.

To open the main help system directly, select an option from the Help menu. From this menu you can open the help system at the main contents page, or get help on topics relevant to the currently open document type.

Dialog-level "What's This" help is also available within Protel 99 SE dialogs. Simply click the question mark icon in the top-right corner of the dialog, then click on the control you would like help with. A popup explanation of the control will appear. Click anywhere within the popup to clear it.

Document management with Protel 99 SE

### How do I open and navigate a design database?

Protel 99 SE includes a number of example design files, which can be found in the ***Examples*** folder in your Protel 99 SE installation directory. To examine some of the features of Protel 99 SE's Design Explorer, open the **Photoplotter** example file:
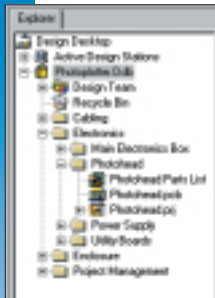
1. Select **File » Open** from the menus and navigate to the ***Design Explorer 99 SE\Examples*** folder.

2. Click on **Photoplotter.ddb** in the file list and then click the **Open** button. The design will open and be shown in the navigation tree.

3. Click the **+** symbol next to the design name in the navigation tree to expand the folder and show its contents.

4. Click the **+** symbols to expand the **Electronics** folder, and then the **Photohead** folder. This folder contains the PCB layout, parts list and schematics for a photoplotter photohead.

5. Click the **Photohead.pcb** file icon in the navigation tree. The PCB layout will open in the design window.

6. Click the **Photohead.prj** file icon. The top level schematic for the circuit will open. You can switch between open documents by clicking on the tabs at the top of the design window.

7. Continue exploring this example design. To close an open document, right-click on its tab in the design window and select **Close** from the popup menu (shortcut **CTRL+F4**). To close the entire design and remove it from the navigation tree, select **File » Close Design** from the menus.

How can I view multiple design documents simultaneously?

1. Open **Photoplotter.ddb** found in the *Design Explorer 99 SE*
   Examples folder.

2. Click the **+** symbol next to the design name in the navigation tree to
   expand the folder, and then expand the **Electronics** and **Photohead**
   folders. Click the **Photohead Parts List** icon to open the document in the
   design window. Do the same for
   Photohead.pcb and **Photohead.prj**.

3. Right-click on the **Photohead Parts
   List** tab in the design window and
   select **Split Horizontal** from the
   popup menu. The design window will
   be split, with the **Photohead Parts
   List** shown in one half and the other
   open documents in the other.

4. Right-click on the **Photohead.prj** tab
   in the design window and select
   Split Vertical from the popup menu.
   The design window will be split
   again. You can now view all three
   documents simultaneously. You can move the border of a split by moving
   the mouse over the split axis until the cursor changes to a double-headed
   arrow. Click-and-drag to move the split axis.

5. To remove the window splits, right-click on any tab in the design window
   and select **Merge All** from the popup menu.
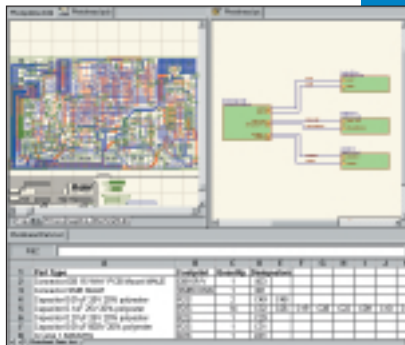
How do I create a new design or create new documents within a design?

1. First create a new design by selecting **File » New Design** from the menu (if no design is currently open simply select **File » New**). In the resulting dialog, you can set the **Design Storage Type** to **MS Access Database** to store your new design as a single design database, or **Windows File System*** to create stand-alone design files. For this example choose **MS Access Database**. Type a name for your design in the **Database File Name** field, and click the **Browse** button to select a location in which to store the new design. Once you have finished, click **OK** to create the new design. Your new blank design will be opened in the workspace.

2. The design window will show the contents of the design, which includes a default document folder called **Documents**. Double-click on the **Documents** icon to open the folder as a new tab in the design window. You will be presented with a blank folder.

3. Let's add a schematic to the design, stored in the **Documents** folder. Right-click anywhere in the blank Documents pane in the design window and select **New** from the popup menu.

4. The resulting dialog will show a list of document types that can be created. Check the **Show all document kinds** to show documents from all OLE-compliant applications. Select **Schematic Document** and click **OK**. A blank schematic sheet will be created in the current design folder.

5. Type a name for the sheet and press **ENTER** to save the name.

6. Double-click on the schematic sheet icon to open the sheet in the design window. You are now ready to start designing your circuit. Protel 99 allows you to store any number of documents in a design. Folders are special documents that contain other documents or folders, allowing you to organize your documents in a hierarchical fashion within the design

database. You can "drag-and-drop" documents and folders in the design window and navigation tree to rearrange the structure of the database.

As well as native Protel documents, Protel 99 SE lets you store any type of document within a design database. When a folder is active, right-click in the design window and select **Import** to import a document into the design.

*Note: Setting the data type to **Windows File System** requires less disk space to store your design, however in this mode you will not be able to use Protel 99 SE's document sharing and some advanced document management features.

Design Capture with Protel 99 SE

How does Protel 99 SE support multiple schematic sheets?

1. Open **LCD Controller.ddb** found in the **\Design Explorer 99 SE\ Examples** folder.

2. Click the **+** symbol next to the design name in the navigation tree to expand the folder, and then expand the **LCD Controller** folder to view its contents.

3. Click the **LCD Controller.prj** icon to open the schematic in the design window. This is the top level schematic sheet. The green rectangles are called "sheet symbols" and each one represents a schematic sub-sheet.

4. Click the sheet navigation button on the main schematic toolbar. The cursor will change to a crosshair.

5. Click on a sheet symbol. Protel 99 SE will take you to the appropriate sub-sheet.

6. Right-click or press **ESC** to exit sheet navigation mode.
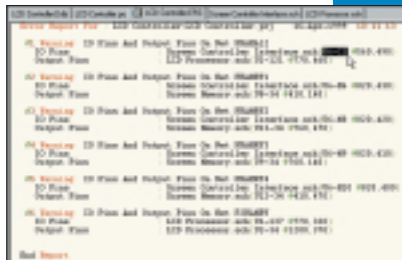
   The navigation tree (click the **Explorer** tab at the top of the Design Manager panel to view the tree) shows a hierarchical view of the schematic project. Protel 99 SE supports unlimited sheets and hierarchy depth.

   Protel 99 SE's hierarchical schematic environment allows you to adopt a modular "top down" or "bottom up" approach to design. You can create your top level schematic first, using sheet symbols to represent schematic sub-sheet modules, then select **Tools » Create Sheet From Symbols** to automatically generate sub-sheet templates. Alternatively you can create your sub-sheets first and select **Tools » Create Symbol From Sheet** to generate a sheet symbol representing the schematic.
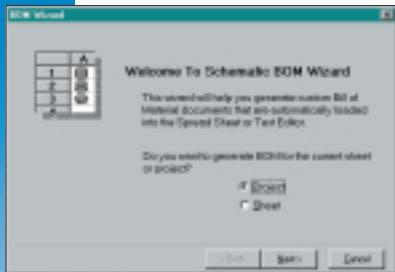
How do I check the electrical integrity of my schematic?

1. Open **LCD Controller.ddb** found in the *\Design Explorer 99 SE \Examples* folder.

2. Click the **+** symbol next to the design name in the navigation tree to expand the folder, and then expand the **LCD Controller** folder to view its contents. Click the **LCD Controller.prj** icon to open the schematic in the design window.

3. Protel 99 SE helps you find problems with your design using a powerful Electrical Rules Check (ERC) feature.
Select **Tools » ERC**. The ERC includes options for checking for a number of common drafting-type errors. As well, the **Rule Matrix** tab of the ERC setup dialog allows you to set invalid electrical conditions, such as an output pin connected to another output pin.

4. Leave all options in the Setup Electrical Rules Check dialog set to their defaults and click **OK** to run the ERC. The ERC report **LCD Controller.ERC** will be automatically created and displayed.

5. In the ERC report note that several warnings are shown regarding IO pins connected to Output pins, a condition set in the Rule Matrix. Use the mouse to highlight one of the pin numbers in the report, for example "U6-C2" (highlight just the component designator and pin number text).

6. Click the cross probe button on the main toolbar. The relevant schematic sheet will be activated with the highlighted component pin centered on the screen. This allows you to easily navigate to the problem areas of your schematic, regardless of which sheet contains the error.

How do I create a bill of materials for my design?

1. Open **4 Port Serial Interface.ddb** found in the **\Design Explorer 99 SE\Examples** folder.
2. Click the **+** symbol next to the design name in the navigation tree to expand the folder, and then expand the **4 Port Serial Interface** folder to view its contents. Click the **4 Port Serial Interface.prj** icon to open the schematic in the design window.

3. Select **Reports » Bill Of Material** from the schematic menus to start the BOM Wizard. The BOM Wizard leads you through all the steps necessary to generate a bill of materials in either text format, or as a Microsoft Excel-compatible spreadsheet.
4. Step through the BOM Wizard leaving all settings at their default values. When the Wizard is finished, a bill of materials for the project will be generated as an Excel-compatible spreadsheet and displayed with Protel 99 SE's built-in spreadsheet editor.

How can I simulate my circuit?

1. Open **Bandpass Filter.ddb** found in the **\Design Explorer 99 SE\Examples\Circuit Simulation** folder. The schematic shows the circuit for 1kHz bandpass filter, and the simulation results for this circuit are shown in the lower part of the design window.
2. Click the **AC Analysis** tab in the simulation results to view the frequency response for the circuit.

3. To better view this response, right-click in the cell containing the **out** waveform and select **View Single Cell** from the popup menu. Right-click on the waveform and select **Scaling** from the popup menu. In the resulting dialog set the X Scale to **Log**, the Primary Y Scale to **Magnitude in Decibels**, and the Secondary Y Scale to **Phase In Degrees**. Click **OK** to close the dialog. The waveform will now be displayed as a familiar Bode plot, with both magnitude and phase responses shown.

4. Click anywhere on the schematic to make it the active document. Select **View » Fit All Objects** from the schematic menu to zoom in on the circuit.

5. Double-click on the symbol for the capacitor **C1** and in the resulting dialog change the value in the **Part Type** field of the **Attributes** tab to be **0.2uF** then click **OK** to close the dialog. Repeat this for **C2**.

6. Select **Simulate » Setup** from the schematic menu. In the **General** tab of the resulting dialog, click the **Keep Last Setup** radio button and then click the **Run Analyses** button to rerun the simulation.

7. Examine the **AC Analysis** tab in the simulation results and you will see the center frequency for the circuit has moved from 1kHz to 100Hz.

Protel 99 SE comes supplied with a comprehensive set of "simulation ready" schematic symbols, which can be found in the libraries contained in **\Design Explorer 99 SE\Library\Sch\Sim.ddb**. Simply construct your schematic using components from these libraries and you can easily run analog and digital simulations straight from the schematic.

How can I include programmable logic in my design?

1. Open **LCD Driver.ddb** found in the *\Design Explorer 99 SE\Examples \PLD* folder.

2. Click the **+** symbol next to the design name in the navigation tree to expand the folder, and then expand the **LCD Driver** folder to view its contents. Click the **LCD.sch** icon to open the schematic in the design window. The schematic shows the functional circuit for an LCD driver to be implemented in a G22V10 device.

3. Select **PLD » Configure** from the schematic menu. The resulting dialog shows the various compiler options and the available output files that can be generated. Leave all settings at their defaults. Note that the G22V10 device has been selected as the target device. Click **OK** to close the dialog.

4. To create the files necessary to program the device, select **PLD » Compile** from the schematic menu.

5. When the compiler has finished, ensure that the **View Files** option is checked in the PLD compiler message box, then click **Close** to close the dialog. The compiler output files will be created and opened in the design window. Click the tabs to view each file. The **LCD.JED** document is the JEDEC programming file for the device.

   When you design a circuit using the logic symbols in Protel 99 SE's PLD symbols libraries (found in *\Design Explorer 99 SE\Library\ Sch\PLD.ddb*), you can compile the circuit directly from the schematic to produce an industry-standard JEDEC file ready for download into the device programmer.

   You can also use Protel 99 SE's integrated text editor to create PLD files directly in the device-independent CUPL Hardware Description Language, using traditional truth tables, state machines and Boolean equations.

PCB Design with Protel 99

How do I start to layout the board?

1. Open **4 Port Serial Interface.ddb** found in the *\Design Explorer 99 SE\Examples* folder.

2. Click the **+** symbol next to the design name in the navigation tree to expand the folder, and then click on the **4 Port Serial Interface.ddb** folder icon to view its contents in the design window. Note that the design already contains a finished PCB, but we will start a new layout for the circuit.

3. Select **File » New** from the menu. In the **Wizards** tab of the **New Document** dialog select **Printed Circuit Board Wizard** and then click **OK**.

4. Step through the Wizard and make the following selections: IBM XT bus format; XT short bus; Two Layer - Plated Through hole and No power/ground planes; Through hole Vias only; Through hole Components and Two tracks between pads; default track, via and hole sizes and clearances. Click the **Finish** button to create the new board.

5. Before transferring the schematic information you must load the necessary footprint libraries, which in this example have been stored within the design. Select **Design » Add/Remove Library** from the PCB menu. In the PCB Libraries dialog navigate to and select the **4 Port Serial Interface.ddb** found in the *\Design Explorer 99 SE\Examples* folder. Click the **Add** button to add the database to the libraries list, then click **OK** to close the dialog. This makes all the libraries in the database available to the PCB editor.

6. In the navigation tree, click the **4 Port Serial Interface.prj** icon to open the schematic project.

7. With the schematic sheet active select **Design » Update PCB** from the schematic menu. The resulting dialog will list all possible target PCB documents. Select your newly-created PCB from the list and click the **Apply** button.

8. In the **Update Design** dialog, uncheck the two **Classes** options, leave all other options at their default values and click the **Execute** button. Protel 99 SE's design synchronizer will analyze the schematic and PCB documents and determine any differences between them.
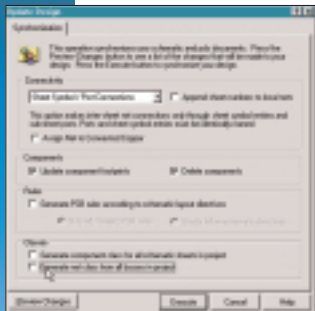
9. Because this is the first time the schematic has been synchronized with this board, the **Confirm Component Associations** dialog will open showing any matched components on the new PCB. Notice that the connector P1 on the schematic has been matched with the XT edge connector on the board. All other components are listed as unmatched because they have not yet been placed on the board. Click **Apply** to update the PCB.

10. When the synchronizer has finished updating the design, click the tab for your new PCB to make it active. All component and connectivity information from the schematic has been transferred to the PCB ready for layout and routing.

Protel 99 SE's Printed Circuit Board Wizard allows you to choose from over 60 industry-standard outlines, or create a custom outline with the option of saving the outline as a template for future use.

Protel 99 SE's design synchronizer can be re-run at any time in the design process to reflect design changes between schematic and PCB documents.

How do I define the layer stackup of my board?

1. Open **Z80 Microprocessor.ddb** found in the ***Design Explorer 99 SE*
Examples** folder.

2. Click the **+** symbol next to the design name in the navigation tree to
expand the folder, and then expand the **Surface Mount Revision** folder to
view its contents. Click the **Z80 Processor board.pcb** icon to open the
board in the design window.

3. From the PCB menus select **Design » Layer Stack
Manager**. The **Layer Stack Manager** allows you to
fully define the makeup of your board. The
diagram shows the current layer stackup of the
board, which consists of top and bottom signal
layers, and two internal planes.

• Double-click on the layer names to the left of the
diagram to edit the properties of a layer, or to
rename that layer.

• Double-click on the names to the right of the diagram to edit the
properties of the layer substrate.

• Use the buttons on the top right of the dialog to add, delete or
rearrange layers.

• Use the dropdown list on the top left of the dialog to select the
appropriate layer technology for your board.

4. Click the **Drill Pairs** button on the bottom right of the dialog. The **Drill
Pair Manager** dialog allows you to set up allowable drilling layer pairs. The
board is currently set for through-hole drilling from the top to bottom
layers only.
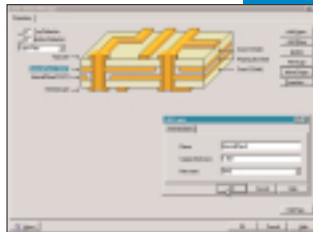
5. Right-click in the drill pairs list and select **Create Drill Pairs From Layer Stack** from the popup menu. Answer **Yes** to the confirmation dialog and Protel 99 SE will create a set of drill pairs based on the available layers defined in the Layer Stack Manager. These drill pairs are used by the Layer Pairs design rule to detect invalid blind and buried vias.

Protel 99 SE's Layer Stack Manager and Drill Pair Manager give you full control over the construction of your board.

How do I ensure power tracks have a certain minimum width?

1. Open **Z80 Microprocessor.ddb** found in the **\Design Explorer 99 SE\Examples** folder.

2. Click the **+** symbol next to the design name in the navigation tree to expand the folder, and then expand the **Z80 Processor** folder to view its contents. Click the **Z80 Processor board.pcb** icon to open the board in the design window.

3. Select **Design » Rules** from the PCB menu and activate the **Routing** tab in the Design Rules dialog.

4. Click on **Width Constraint** in the **Rule Classes** list to select it. In the list of rules you will notice that the power nets for the board have special rules applied that individually set their track widths. You can set up multiple design rules of the same class and apply them to different objects by setting their scope. Protel 99 SE applies rules hierarchically depending on scope. In this case the width rule set for individual nets overrides the board width rule of 10mil. You can disable individual design rules from this dialog. Double-click on a rule to edit it.

5. Click **Close** to close the Design Rule dialog.

6. To run a design rule check, from the PCB menu select
Tools » Design Rule Check. On the **Report** tab ensure that the **Create Report File** option is checked and then click the **Run DRC** button to generate a report listing all selected rule violations.

Protel 99 SE's board design environment is fully rules-driven. You can set up a comprehensive set of design rules covering routing, manufacturing, high speed design, signal integrity and more.

Design rules can be monitored on-line as you work, with violations instantly highlighted. Also, Protel 99 SE's sophisticated manual routing features force compliance with relevant design rules, such as width constraints, as you route, minimizing the chance of design errors.

Design rule sets can be exported and then imported into other designs, allowing you to create reusable rule groups.

How do I manually route my board?

1. Open **Board1.ddb** found in the **\Design Explorer 99\Examples\PCB Auto-Routing** folder.
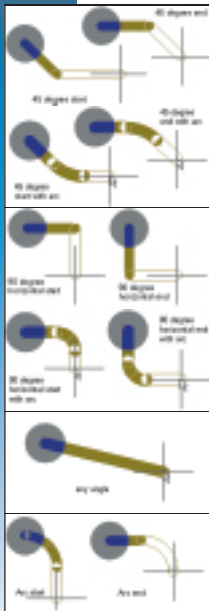
2. Click the **+** symbol next to the design name in the navigation tree to expand the folder, and then expand the **Board 1** folder to view its contents. The folder contains identical PCB layouts – one routed, the other unrouted. Click the **Board1.pcb** icon to open the unrouted board in the design window.

3. The board should show connection lines between the pins of the placed components. If connection lines are not visible, press the **L** key to open the Document Options dialog at the **Layers** tab and ensure the **Connections** option is checked. Click **OK** to close the dialog.

4. Position the cursor over the center of the board and press the **PGUP** key to zoom in on the board, centered on the cursor position. Repeat this until the pin numbers and net names are visible on the pads.

5. Click on either the **Top** or **Bottom** layer tabs at the bottom of the board design window to select the layer.

6. Press the **P** key to popup the placement menu and then press the **T** key to select the **Interactive Routing** item. The cursor will change to a crosshair. Position the cursor over a pad and left-click to start routing.

7. Left-click to anchor each track segment as you route. Press **BACKSPACE** to rip up the last track segment you placed. Right-click or press **ESC** to finish placing a track. You will remain in track placement mode, allowing you to begin another route. Right-click or press **ESC** again to drop out of track placement mode.

Protel 99 SE includes several track placement modes: 45 degree, orthogonal, oblique or arc. Press **SHIFT+SPACEBAR** during track placement to cycle through these modes. Within each mode, press **SPACEBAR** to toggle between start and end angle modes.

To change layers when routing, press the "*****" asterisk key on the numeric keypad to cycle through the available signal layers. Protel 99 SE will automatically insert a via where necessary.

Protel 99 SE uses a powerful connectivity algorithm that continually re-analyzes the design as you route, optimizing and adjusting the connection lines as each track is placed. This allows you to route connections along any path, without regard to the current connection line.

An electrical grid automatically overrides the snap grip, allowing you to easily route connections to off-grid component pins. Holding down the **CTRL** key during routing temporarily disables the electrical grid.

During manual routing, Protel 99 SE's unique "look ahead" feature enables you to easily position tracks, without guesswork.

Protel 99 SE's "Avoid Obstacle" routing mode automatically monitors your routing and prevents you from inadvertently violating a design rule by placing a track across an electrical object not on the same net. Holding down the **ALT** key during routing temporarily suspends this feature.

Protel 99 SE's "Push Obstacle" mode allows you to automatically push tracks out of the way as you route. To change routing modes, select **Tools » Preferences** from the PCB menu. In the **Options** tab of the resulting dialog, select an option from the **Interactive Routing Mode** dropdown list.

How do I automatically route part or all of my PCB?

1. Open **Board1.ddb** found in the *Design Explorer 99 SE\\Examples\\PCB Auto-Routing* folder.

2. Click the **+** symbol next to the design name in the navigation tree to expand the folder, and then expand the **Board 1** folder to view its contents. The folder contains identical PCB layouts – one routed, the other unrouted. Click the **Board1.pcb** icon to open the unrouted board in the design window.

3. Position the cursor over the center of the board and press the **PGUP** key to zoom in on the board, centered on the cursor position. Repeat this until the pin numbers and net names are visible on the pads.

4. To automatically route a particular net, select **Auto Route » Net** from the PCB menu. The cursor will change to a crosshair. Left-click on any pad on the relevant net and Protel 99 SE will automatically route the entire net. Continue to select other nets to route, or right-click or press **ESC** to exit net autorouting mode.

5. You can automatically route a single connection within a net, all pads on a particular component, or all connections within a defined area by repeating step 4 above and selecting the relevant option (Connection, Component, Area) from the Auto Route menu.

6. To automatically route the entire board, select **Auto Route » All** from the PCB menu. The Autorouter Setup dialog will open. If you wish to keep pre-routed connections on the board, check the **Lock All Pre-routes** option. Click the **Route All** button to start autorouting.
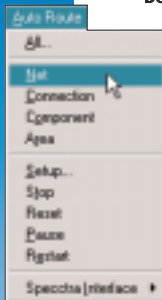
Protel 99 SE includes a powerful shape-based autorouter that uses automated board analysis techniques to produce routing results comparable with that of an experienced board designer. Protel 99 SE automatically analyzes your board design and chooses the most efficient routing algorithm.

Unlike maze-type routers which work with rectangular blocks, Protel 99 SE's shape-based autorouter analyzes the board using complex polygon shapes, ensuring true diagonal routing.

How do I generate manufacturing output?

1. Open **Z80 Microprocessor.ddb** found in the **\Design Explorer 99 SE\Examples** folder.

2. Click the **+** symbol next to the design name in the navigation tree to expand the folder, and then expand the **Z80 Processor** folder to view its contents. Click the **Z80 Processor board.pcb** icon to open the board in the design window.

3. From the PCB menu select **File » Cam Manager**. This will start the **Output Wizard**. The **Wizard** guides you through the process of configuring the various manufacturing outputs for your design.

4. Click the **Next** button to go to the second page of the Wizard. On this page select **Gerber** from the list.

5. Click the **Next** button to step through each page of the Wizard, selecting the Gerber settings you want (you may simply leave everything at the default settings for now). On the last page of the Wizard click **Finish** to create the output configuration file.

A CAM Outputs document is created in your design. This document can contain any number of different output types such as BOMs, NC Drill files, etc. The document that you have just created contains one output – that for producing Gerber files as specified. To add more outputs simply select **Tools » CAM Wizard** to re-run the Wizard, or choose an output type directly from the Edit menu.

• To edit the properties of output, double-click on the name of the output in the list.
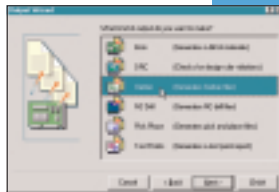
- To enable/disable a particular output within an output file, check/uncheck the box next to the output name.

6. To generate all specified output files, select **Tools » Generate CAM Files** from the CAM Manager menu.

The CAM outputs will be created and stored in an outputs directory within your design.

How do I create assembly drawings for my board?

1. Open **LCD Controller.ddb** found in the *Design Explorer 99 SE\Examples* folder. Click the **+** symbol next to the design name in the navigation tree to expand the folder, and then expand the **LCD Controller** folder to view its contents. Click the **LCD Controller.pcb** icon in the navigation tree to open the board in the design window.

2. From the PCB menu select **File » Print/Preview**. A printer document will be created in your design and opened in the workspace. This shows a default composite printout of your board.

3. From the preview menu select **Tools » Create Assembly Drawings** and answer **Yes** to the confirmation prompt. Protel 99 SE will create assembly drawing print previews for your design.

4. If the PCB Print Browse panel is not active, click the **Browse PCBPrint** tab at the top of the navigation pane. The Browse Panel shows you the current printouts. The top level of the tree shows the different printouts that will be produced.

- Click on a printout name to preview that printout in the main window.
- Click the **+** symbol next to a printout to see the layers that it contains.
- Right-click on a printout name in the Browse Panel to edit the properties of the printout or insert a new printout.

- Right-click on a layer in the Browse Panel to insert or delete layers in a current printout, or edit the properties of a layer.
- To change the print settings (such as the orientation or page size), select **File » Setup Printer** from the menu.

5. When you are ready to print, select one of the Print options from the **File** menu.

6. By default your current print settings are stored in a file called **Preview *filename*.PPC**. To store the print settings for the assembly drawings you have just created so that they won't be overwritten, select **File » Save Copy As...** from the menu and save a copy of your printout with a different name. You can store any number of printouts in a design. To re-use or edit a particular printout, simply open the appropriate printout document in the design window.

Protel 99 SE's PCB Power Print gives you full control of the printing process and allows you to easily create assembly drawings, drill drawings, etc. Print settings are stored with your design so you can reprint at any time, and you can store any number of different printouts within a single design.

Component Library Management with Protel 99 SE

How are libraries stored in Protel 99 SE?

In Protel 99 SE both schematic symbol and component footprint libraries are stored within designs. Any number of libraries can be stored within a single design.

Protel 99 SE comes supplied with a comprehensive set of symbol and footprint libraries. These can be found in the library design databases in the

\Design Explorer 99 SE\Library\Sch folder (for schematic symbols) and the

\Design Explorer 99 SE\Library\PCB folder (for PCB component footprints).

To use symbols and footprints contained within libraries in a schematic or PCB document you do not need to open the relevant library design databases. You must, however, tell the schematic or PCB editor the names of the designs that contain the libraries you wish to use.

1. When any schematic or PCB document is active in the design window, click the **Browse Sch** or **Browse PCB** tab at the top of the navigation pane to activate the browse panel.

2. Set the **Browse** mode to **Libraries** using the dropdown list. Any available libraries will be shown in the display list.

3. To add a library to the list, click the **Add/Remove** button on the browse panel. In the resulting dialog you can browse for designs. Select a design and click the **Add** button to add it to the list of available files. All libraries of the relevant type contained within the database will be made available in your documents. Click **OK** to close the dialog.

Once libraries are loaded into the schematic or PCB editor, the symbols or footprints they contain can be added to your design.

To edit the contents of a library, you must open the database containing the library in the Design Explorer. If the library is currently loaded into the schematic or PCB

editor, you can select a component from the library in the browse panel list and click the **Edit** button to open the library database directly for editing.

How do I find a schematic symbol for a 22V10 PLD?

1. Open any of the design examples found in the **\Design Explorer 99 SE\ Examples** folder.

2. Use the **+** symbols to expand the design folders and open any schematic document by clicking on its schematic icon in the navigation tree.

3. Select **Tools » Find Component** from the schematic menus.

4. In the **By Library Reference** text field, type ***22V10***, including the asterisk wildcard symbols.

5. Ensure that the search **Scope** is set to **Specified Path**, and that the **Path** to the schematic libraries folder is correct.

6. Click **Find Now** to start the search.

Protel 99 SE will list any components matching the search description and the libraries they are stored in. You can then place a component directly onto your schematic, edit its schematic symbol or load the libraries into Protel 99 SE's current libraries list to browse the other components they contain.

You can search for components by library reference or by their description fields. Protel 99 SE will search currently loaded libraries, a specified path, or all drives on your system (including mapped network drives) for references to the specified component. The component search string supports the ***** and **?** wildcard symbols, allowing you to take into account different manufacturer's naming conventions.

Protel maintains an active Library Development Center which continually updates libraries and creates new ones. The latest libraries are available for download at Protel's extensive website at **www.protel.com**.

How do I create a new PCB footprint?

1. Open **LCD Controller.ddb** found in the **\Design Explorer 99 SE\Examples** folder. Click the **+** symbol next to the design name in the navigation tree to expand the folder, and then expand the **LCD Controller folder** to view its contents.

   PCB footprints are created and stored in PCB library documents within a design database. The extensive footprint libraries supplied with Protel 99 SE can be found in the library design databases in the **\Design Explorer 99 SE\ Library\PCB** folder. For the purposes of this example we will create a new PCB footprint library based on the footprints on the LCD controller board.

2. Click the **LCD Controller.pcb** icon in the navigation tree to open the PCB layout document in the design window.

3. Select **Design » Make Library** from the menus. The Library Editor is opened and a new library **LCD Controller.lib** is created within the design containing all footprints on the board.

4. Click the **Browse PCBLib** tab at the top of the navigation pane to activate the browse panel and see a list of components in this library.

5. To create a new component, select **Tools » New Component** from the PCB Library menu. The **Component Wizard** will start. The Wizard steps you through the process of creating a new footprint, allowing you to choose from a variety of packaging technologies and set various options and dimensions. When the Wizard is finished, the new component is added to the current library ready for use.

You can also copy existing components within or between libraries to use as the basis of a new component. Simply select the component in the browse panel of the PCB Library Editor and then select **Edit » Copy Component** from the menus. Select **Edit » Paste Component** to paste a copy of the component into the same or a different library.

Teamwork with Protel 99 SE

How can multiple designers work on the same design?

1. Open any of the design examples found in the **\Design Explorer 99 SE\Examples** folder.

2. Use the **+** symbol next to the design name in the navigation tree to expand the tree and view the contents of the design.

3. Click the **+** symbol next to the **Design Team** icon to expand the folder and click on **Sessions**. This window shows a list of open folders and documents in the design along with information on who has them open.

4. Use the navigation tree to open some design documents then return to the **Sessions** tab. The documents you opened will appear in the list.

5. Right-click on any document name list and select **Lock** from the popup menu. Locking a document prevents other team members from editing it and inadvertently saving over your work. Right-click on a document you have locked to unlock it. All documents you have locked are automatically unlocked when you close the document or design, or when you quit Protel 99 SE.

When you work with designs using the MS Access Database data storage type, Protel 99 SE's SmartTeam technology allows safe sharing of designs across a network, facilitating design team collaboration. Different designers can simultaneously work on different documents within a single design database. You can lock a document while you work on it to prevent other team members from accidentally overwriting your work. Also, Protel 99 SE will warn you if you are trying to save a file that is in use by another team member, whether they have locked it or not.

How do I restrict access to certain design documents?

1. Open any of the design examples found in the *****Design Explorer 99 SE* **Examples*** folder. Use the **+** symbol next to the design name in the navigation tree to expand the tree and view the contents of the design.

2. Click the **+** symbol next to the **Design Team** icon to expand the folder and click on **Members**. This window shows a list of design team members. Every design has two default members – **Admin** and **Guest**. The **Admin** member has unrestricted access to the entire design, and is used as the default login name. To activate the login screen for a design you must assign a password to **Admin**.

3. Right-click on **Admin** in the members list and select **Properties** from the popup menu. In the **User Properties** dialog, enter and confirm a password for Admin. Click **OK** to close the dialog and assign the password. Now, each time the design is opened a login screen will ask for a user name and password.

4. To create new team members, each with their own password, right-click anywhere in the members folder in the design window and select **New Member** from the popup menu and complete the details in the dialog.

5. By default, all new members you create have full access to all parts of the design, except the design team folders. To restrict access for a particular member or members you must set up a **Permission** rule.

6. In the navigation tree, click on the **Permissions** icon to open the folder in the design window. This shows the read (R), write (W), delete (D) and create (C) permissions set for the database.

7. To add a new permission, right-click anywhere in the **Permissions** folder in the design window and select **New Rule** from the popup menu. Configure the rule in the Permission Rule Properties dialog and click **OK** to save it.

When working with designs using the MS Access Database data storage type, creating team members and permissions for each design allows you to easily manage design access and team collaboration, all from within the Protel environment.

Further Exploration

How do I learn more about Protel 99 SE?

Would you like more information about the great features in Protel 99 SE?

Fax or mail a copy of this page to your nearest Protel Sales and Support Office to receive your Protel 99 SE Product Brochure.

Refer to the back cover for contact details.

☐   Please send me more information about Protel 99 SE

☐   Please have a sales representative call me

Company _____

Department_____

Contact Name _____

Title _____

Address_____

_____

State _____Postcode _____

Email_____

Phone _____

Fax _____

How to purchase Protel 99 SE

For more information about Protel's range of products and technical resources for designers, visit: www.protel.com

NORTH AMERICA
Protel Technology Inc.
5252 N. Edgewood Dr.
Suite 175, Provo, UT 84604
Phone: (801) 224-0333
Fax: (801) 224-0558
Toll Free: 1 800 544 4186
E-mail sales: salesusa@protel.com
E-mail support: helpme@protel.com

EUROPE
Protel Europe AG
Hinterdorfstrasse 33
CH-4334 Sisseln
Phone: +41 62 866 41 11
Fax: +41 62 866 41 10
Freecall sales:
In deutscher Sprache: 00800 776 776 77
In Dutch or English: 00800 776 776 44
En Français: 00800 776 776 55
Support phone:
In deutscher Sprache: +41 62 866 41 31
In Dutch or English: +41 62 866 41 32
En Français: +41 62 866 41 30
E-mail sales: eda@protel.ch
E-mail support: support@protel.ch

JAPAN
Protel Japan KK.
351-1 Sunayama-cho
Hamamatsu, SHIZ 430 0926, Japan
Sales phone: +81 53 453 6705
Support phone: +81 53 453 8058
Fax: +81 53 453 6707
E-mail sales: sales@protel.co.jp
E-mail support: support@protel.co.jp

AUSTRALASIA
Protel International Limited
PO Box 1876 Dee Why
NSW 2099 Australia
Phone: +61 2 9984 0016
Fax: +61 2 9984 0017
Freecall sales: 1 800 030 949
Freecall support: 1 800 676 684
E-mail sales: sales@protel.com.au
E-mail support: support@protel.com.au