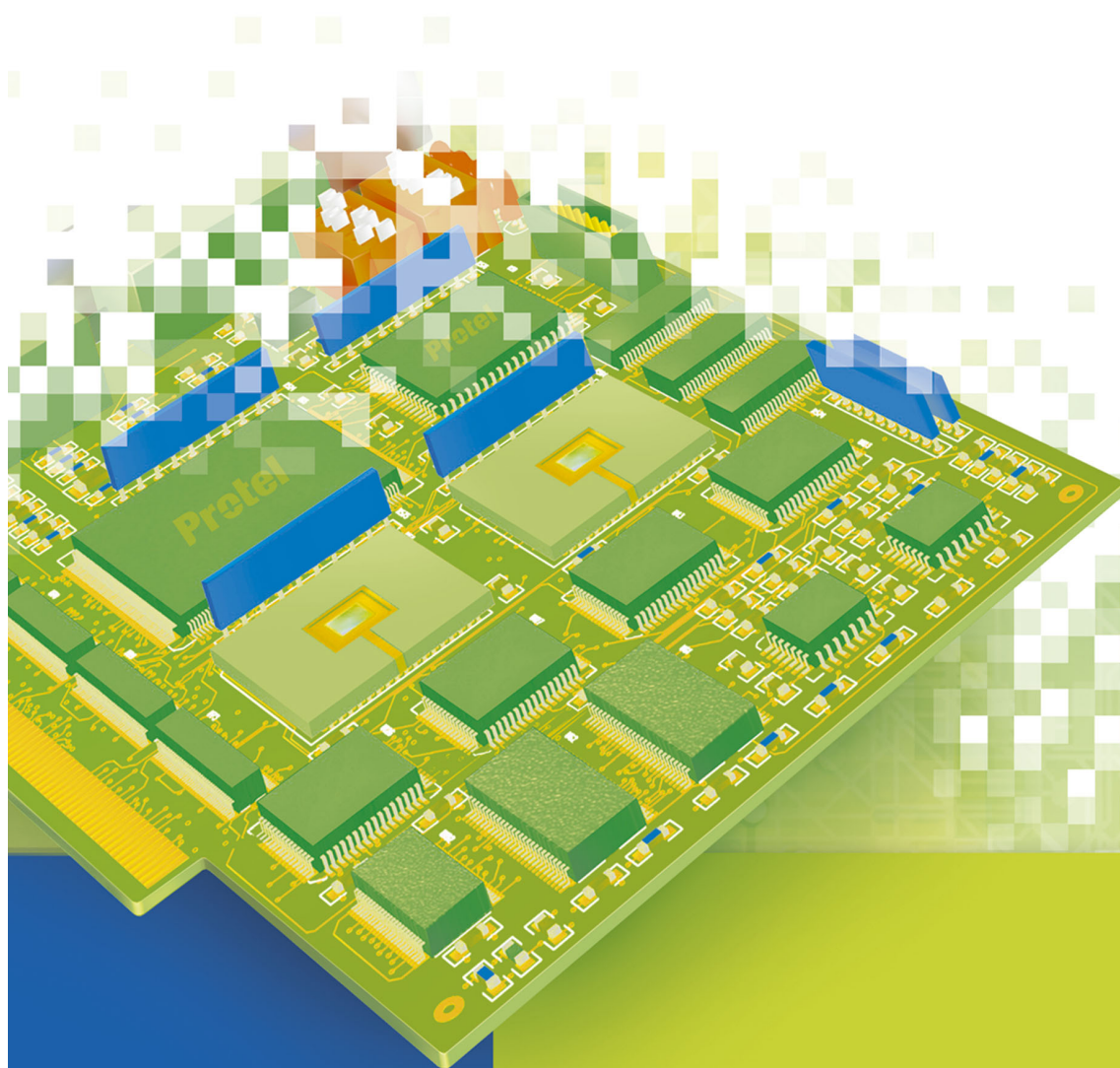


What's New in

***Protel**DXP™*



Protel®
Board-level design system from *Altium*.

An overview of the new features in Protel DXP

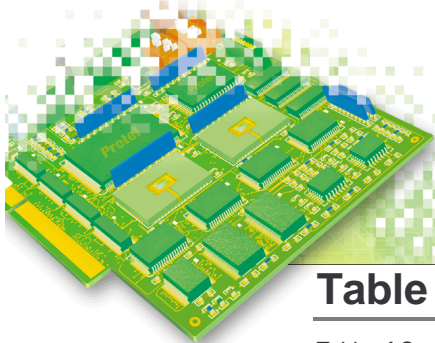
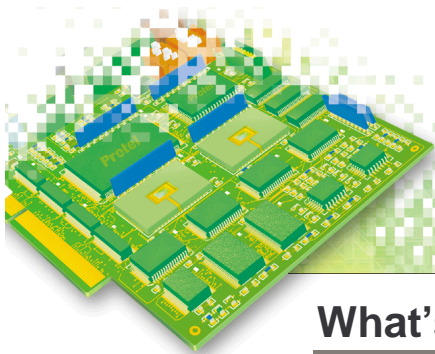


Table of Contents

Table of Contents	1
Table of Contents	2
What's New in Protel DXP.....	3
A New Approach to Design	3
The Protel DXP Project.....	3
The Design Process in Protel DXP	4
Design Synchronization – Matching the Schematic to the PCB	5
How the Schematic Components Link to their PCB Footprint	5
Passing Design Changes Between the Schematic and PCB	5
Designator management	6
New Workspace Concepts.....	7
Filtering the Objects in the Workspace	7
The Object Inspector.....	8
Querying for Objects.....	8
Filtering with Queries	8
Scoping Design Rules with Queries.....	9
The List Panel	9
Navigating the Design	10
Schematic Navigator Panel	10
Defining Design Rules in the Schematic.....	10
True Multi-channel Design	11
Creating a Multi-Channel Design	11
Designing Assembly Variants	12
Interfacing to a Version Control System	12
Integrated Component Libraries	13
Structuring Your Libraries.....	14
Signal Integrity Models	14
Accessing the Libraries	14
Situs Topological Autorouter	14
Design rule compliance	14
BGA Packaging Support	14
User definable Routing Strategies	14
PCB Layout Enhancements	15
Board Shape	15
Sheet Templates.....	15
New Rule Scoping System.....	16
Component Placement Rooms	16
Dimensioning tools	17
Enhanced Splitting of Power Planes.....	17
Fully Definable Pad Shapes	17
Summary of other Enhancements	18
Schematic Capture	18
Circuit Simulation Enhancements	18
Import and export waveform data	18
Enhanced plot graphics.....	18
Overlay different waveforms	18
Multiple scaled plots	18
Signal Integrity Enhancements.....	19
Signal integrity analysis with the DXP environment.....	19
Signal integrity analysis from the schematic.....	19
Enhanced model integration	19
Project Outputs	19
Dual monitor support.....	19
Glossary	20



What's New in Protel DXP

This document gives an overview of the new features in Protel DXP. It is intended to be read while you explore the new features in the software.

To help you get started with Protel DXP it is recommended that you do the Introductory Tutorial. The tutorial is a simple design for a multivibrator, it will take you through the entire design process – from creating a project, to capturing the schematic, transferring the design to the PCB Editor, generating output, and also simulating the schematic. Doing this tutorial will give you a good overview of designing in Protel DXP.

Protel DXP is a project-centric PCB design platform, with powerful new navigation and object management features, new high-level design capabilities such as design variants and multi-channel design, and numerous enhancements in the design tools.

The Protel DXP user interface has been extensively redesigned, bringing superior functionality to the design space. Features such as panels that can be docked, float, or set to Pop Out mode, automatic fade for floating panels and toolbars whenever an edit is performed, support for dual-monitors, and click and drag customization of resources, all go to make the design process more efficient and enjoyable.

The documentation in Protel DXP has been restructured, the on-line help presents information in a concise reference format for commands, dialogs, and what's this help. To complement the reference-style help there are design articles which explain concepts, and are written from an applied perspective, and tutorials, which detail how to perform various tasks in Protel DXP. Think of the articles and tutorials as a library of information, a library that will expand over time as new material is made available on the Protel website.

A New Approach to Design

The Protel DXP Project

When you design in Protel DXP, you start by creating a project. The project links the elements of your design together, including the source schematics or netlist, and the PCB file. The project also stores the output setups; including schematic and PCB print setups, Gerber and NC drill, and the BOM configuration.

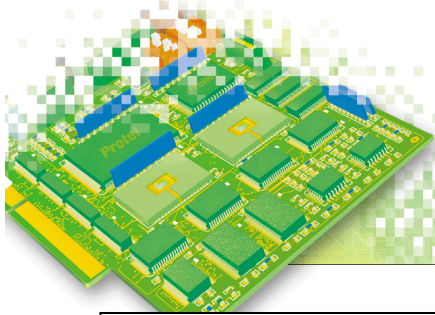
To create a new project select **New Project** from the **File** menu. As you add documents to the project, such as schematic sheets or a PCB, a link to each document is entered into the project file. The schematic and PCB documents can be stored anywhere on your network, they do not need to be in the same folder as the project file.

To edit a document that is part of a project, first open the project to display a list of the documents in the project, then double click on the document you wish to edit in the Protel DXP *Projects* panel.

To add a new document to the active project right-click on any document in the project and select **New** from the floating **Project** menu. Note that this action adds the document to the project – to get a schematic sheet to become part of a schematic hierarchy you place a sheet symbol on the parent schematic, and set the sheet symbol Filename field to that of the child schematic sheet.

Tips

- *As well as being able to open and edit documents by first opening their project, you can also directly open and edit any document.*
- *Project documents store on the same drive are referenced using relative linking in the project file, documents stored on another drive are referenced using absolute linking.*



Protel DXP™

Panels in Protel DXP

Protel DXP makes extensive use of panels – you use them to open files, access libraries, navigate your schematic or board, and edit objects.

Panels can be divided into 2 groups: panels you use from any editor, such as the **Libraries** or **Projects** panel; and panels you use with a specific editor, such as the PCB **Navigator** panel.

Panel display is controlled in the following ways:

Select **Workspace Panels** in the **View** menu to switch to another panel, or to turn a panel on or off.


Panels The **Panels** button located at the bottom right of the workspace can also be used to display or hide any panel.


Click the small down arrow at the top of the panel area to switch to another active panel.

Click on a Tab at the bottom of the panel area to switch panels, right-click on a panel Tab to close a panel.

Positioning Panels

Positioning options include:

Docked – positioned on an edge of the application, click the stuck-pin icon  to switch to **Pop Out** mode.

Pop Out – the panel presents as a button on the edge of the workspace, click on the button to pop the panel out, click back in the workspace and it will hide. Click the stick-pin icon  to switch to **Docked** mode.

Float – float within the workspace – floating panels automatically fade during an edit when the cursor approaches.

Tips

- Click and drag on a Panel Tab, or the Panel name, to move only that Panel.
- Hold the CTRL key to prevent docking.
- Right-click on a panel title bar to control where the panel can dock

The Design Process in Protel DXP

To create you design in Protel DXP you *capture*, *compile*, *debug* and generate *output*.

Capture

You *capture* your design by placing components from the libraries, and wiring them. This can be on a single sheet, or across multiple sheets.

The first step of design capture is placing components. To access components in the libraries first add them to the Installed Library list. To do this make the **Libraries** panel active, then click the **Libraries** button at the top of the panel to display the *Add Remove Libraries* dialog. Supported library formats include the new Protel DXP integrated libraries (*.IntLib), the Protel DXP libraries (*.SchLib and *.PcbLib), and Protel 99 SE schematic and PCB libraries (*.Lib).

Libraries included in your project are automatically made available in the panel whenever you are working on a document from that project.

Validate

Once the design has been captured the next step is to check the connectivity in the design – checking for both drafting type errors, such as a wire not terminating on a pin, and also checking for electrical design type errors, such as a floating input pin. To check the connectivity you *compile* the project. Compiling creates a connective model of the design, and checks this connective model for drafting and electrical errors.

To compile a project select **Project » Compile PCB Project** from the menus, or right-click on a document in the **Projects** panel. When the project is compiled the *Compiled* panel will appear, you can use this to browse the design, or alternatively you can navigate using the new spatial navigation features (described later in this document).

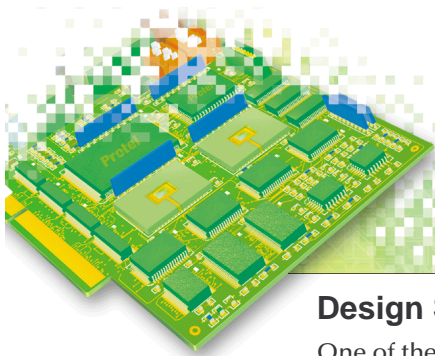
The error checks performed during compiling are configured in the **Error Reporting and Connection Matrix** Tabs of the *Options for Project* dialog (**Project » Project Options**).

Debug

Any warnings or errors detected in the compile process are displayed in the **Messages** panel at the bottom of the DXP interface. They are also listed in the schematic or PCB **Navigator** panel, from the panel you can quickly navigate to an error or warning. To *debug* the schematic project double set the **Navigator** panel to **Navigate Violations**, then single-click in the different fields to navigate to a specific violation. To debug the PCB set the **Navigator** panel to **Rules**, from there you can either examine the violations for a specific rule, or select **All Rules** in the **Rule Classes** and list all violations.

Output

Once you have a warning/error free compile you can generate *outputs* and transfer the design to PCB layout, or from the PCB you generate outputs for fabrication and assembly. As well as being able to configure and generate the outputs from the **File** menu, you can configure and generate all Project outputs by selecting **Output Jobs** the **Projects** menu. Setups configured here are stored in the Project file.



Design Synchronization – Matching the Schematic to the PCB

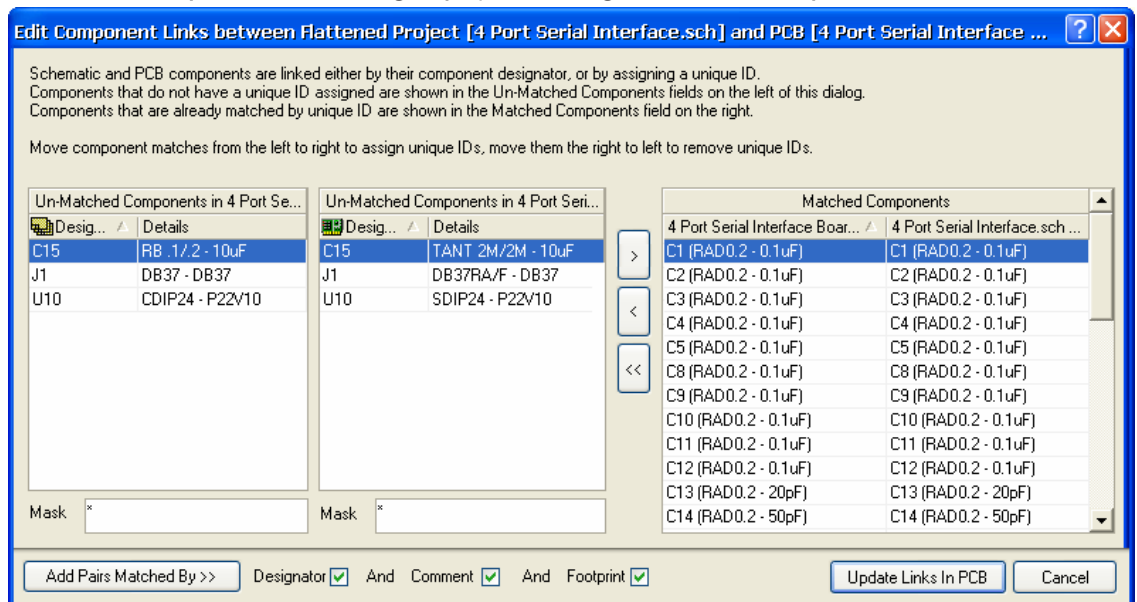
One of the challenges of PCB design is keeping the schematic and the PCB synchronized, or matched. Invariably design changes are made – to the schematic, the PCB, or both.

Underlying design synchronization in Protel DXP is a powerful comparison engine. This comparison engine is used to compare the source schematic project to the PCB. It can also be used to compare any other design documents, such as the PCB to a netlist, a netlist to a netlist, a schematic sheet (or project) to a schematic sheet (or project), in fact any appropriate comparison can be performed.

How the Schematic Components Link to their PCB Footprint

The key to synchronizing between the schematic and PCB is the linking from each schematic component to its corresponding PCB footprint. In Protel DXP this can be done in 2 ways, either by designator, or by a unique identifier.

To check the status of the component links select **Component Links** from the **Project** menu in the PCB editor. The Component Links dialog displays the linkage state of the components.



Component that are linked with a matching unique ID are listed in the **Matched Components** region on the right, components that are not matched by a unique ID are shown in the 2 fields on the left. At the bottom of the dialog are controls that can be used to quickly match components, or you can select unmatched schematic and PCB components and match them manually.

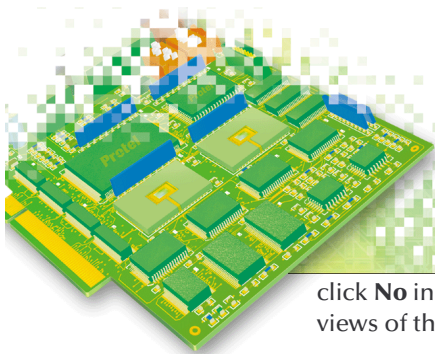
Generally it is better to match components using a unique ID, this allows you to re-annotate the designators on the schematic or the PCB, and always be able to resynchronize the design.

If you attempt to synchronize the design and unique IDs are not assigned for all the components, you will be prompted to attempt to match these components by designator. Note that this matching is by designator only, it does not check the footprint or comment. If you are unsure if matching by designator will be correct click **No** when prompted, then launch the Component Linking dialog to review the matching.

Passing Design Changes Between the Schematic and PCB

Select **Update** in the **Design** menu to transfer design changes from the schematic to the PCB, or from the PCB back to the schematic. The *Engineering Change Order* dialog will appear, listing the changes that need to be applied to the target document(s), to make them match.

Not all differences can be resolved when you perform an Update, for instance net connectivity changes can not be transferred from the PCB back to the schematic. If there are changes that can not be transferred a *Confirm* dialog will appear, detailing how many changes can not be performed. If you



ProtelDXP™

click **No** in this dialog the *Differences* dialog will appear, listing all differences found between the 2 views of the design. The role of this dialog is described below.

Resolving Differences - How Does it Work?

The comparison engine works by performing a detailed comparison of the connective model (netlists) compiled from each design view. The result of this comparison process is a set of differences. The process of detecting and resolving differences between design documents works in the following way:

Compare the design views: Typically this is the schematic project to the PCB. This is done either by selecting **Update** in a **Design** menu, or by selecting **Show Differences** in the **Project** menu.

Generate the list of differences: If you selected Update then Protel DXP knows which way you want the changes applied, so the *Differences* dialog is not displayed. If you selected **Show Differences** all differences are listed in the *Differences* dialog.

Set the change direction: If you have opted to examine the differences first, you must set the change direction in the Differences dialog. Right click and choose from the floating menu to set all updates in one action, or click in the **Update Decision** column and individually set the change direction.

Generate the Change list: After setting the direction you generate an ECO. The ECO is a list of actions that will be performed to resolve the differences.

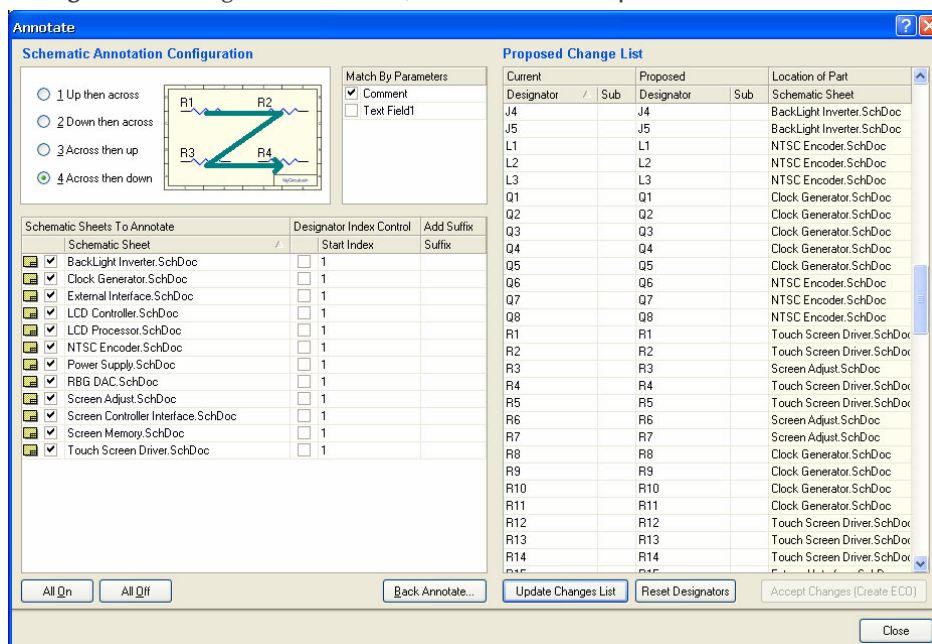
Apply the changes: Execute the ECO to perform the updates.

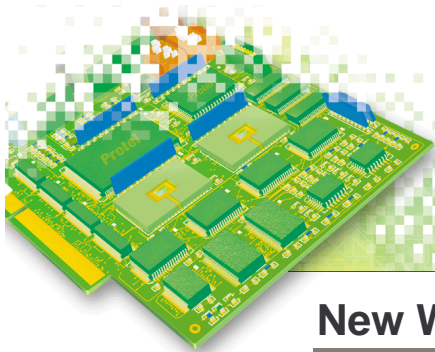
For a list of the object kinds supported in a schematic-to-PCB comparison, look in the **Comparator** Tab of the *Project Options* dialog. Not all object kinds are supported in each design view, for example a Protel Netlist does not support Rooms or Classes.

Designator management

There are tools in both the schematic editor and PCB editor for assigning designators on a positional basis. It is important to be aware if the schematic-to-PCB linking is being done via designator or unique ID. If it is done by designator, you should always perform an Update to resynchronize the schematic and PCB after changing the designators on either side. If linking is done by Unique ID then you are free to annotate in either editor as often as you like before performing an Update. To check how the linking is being done select **Component Links** from the **Project** menu in the PCB Editor.

Select **Tools » Annotate** in the schematic editor to assign the component designators. The Update Change list displays the designators as they are currently assigned, to change the annotation pattern select the direction in the top left of the dialog, click **Reset Designators**, then click the **Update Changes List** button. When the correct designation pattern has been established click the **Accept Changes** button to generate an ECO, which is used to update the schematics.



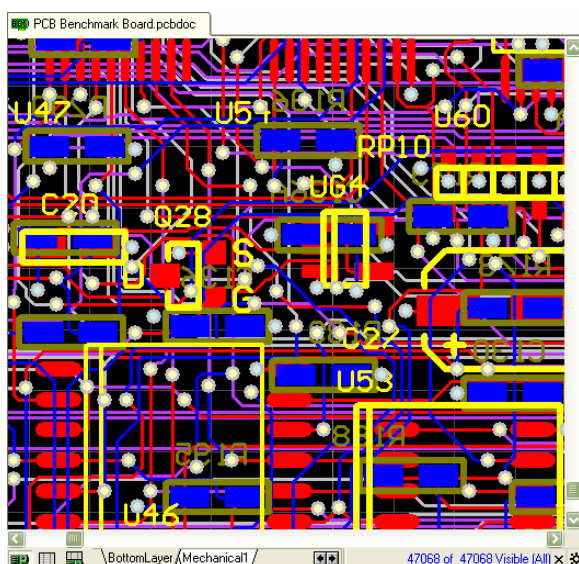


New Workspace Concepts

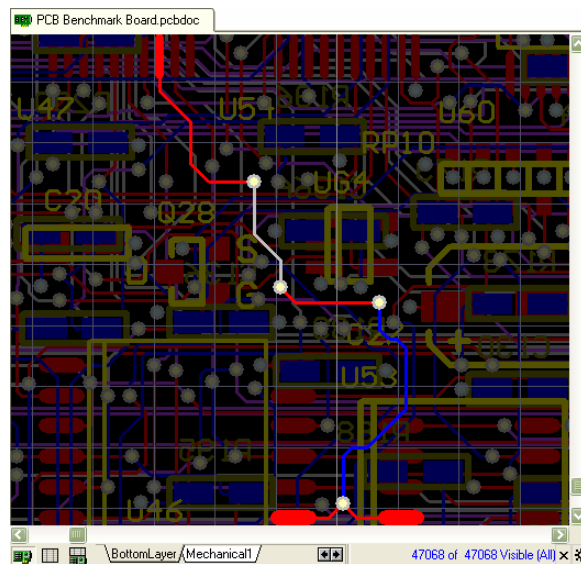
Protel DXP introduces 2 powerful new concepts to design editing – workspace filtering and alternate views of object data. Together these editing techniques give a high level of control over what objects are displayed in the workspace, and editing those objects.

Filtering the Objects in the Workspace

How often have you wanted to control exactly what is currently displayed? Perhaps you wanted to examine the routing a particular net and compare it to another net, or check the location of a set of components, without the visual clutter of seeing all the other components. Protel DXP's new workspace filtering system delivers precisely this – complete control over exactly what is visible in the workspace, with an option to *Mask* objects that have been filtered out. When you mask filtered objects only the objects targeted by your filter remain at normal brightness.



Looking for a net on the board?



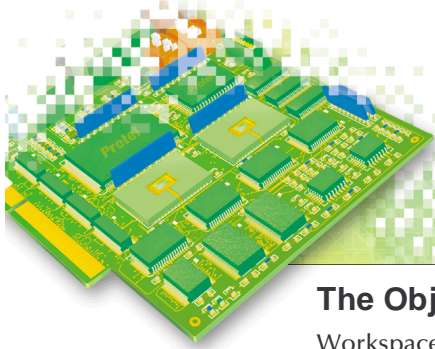
The same section of the board, after clicking on the net name in the PCB Navigator panel to filter for that net

Filtering can be invoked in a variety of ways, including:

1. Clicking on an entry in the schematic or PCB Navigator panel, all objects are faded, except the object(s) belonging to the entry you clicked on.
2. Typing a query in the *List* panel. Queries provide a powerful mechanism to target objects in the workspace.
3. Selecting a net, component, or typing a query in the PCB Filter toolbar.
4. Selecting an entry in the Filter popup menu, press the **Y** key to pop up the menu.

Tips

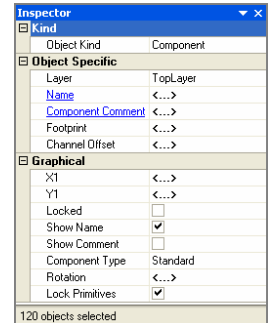
- To change the fade level of masked objects click on the **Mask Level** button on the bottom right of the workspace.
- Only objects displayed at normal brightness can be selected and edited.
- Browsing by Design Rules is an excellent way of verifying what each rule will apply to.
- The PCB Filter Toolbar is a convenient way of filtering by component, net, or query – right click on any toolbar to enable it.



The Object Inspector

Workspace filtering is not only useful for easing the task of controlling what you see, it is also an excellent method of controlling what can be edited. Once you have applied a filter only those objects displayed at normal brightness can be selected or edited.

An individual object can be editing in the traditional way, by double-clicking on it to display its dialog. Alternatively, you can use the new *Object Inspector* panel. The Object Inspector can be enabled by selecting **View » Workspace Panels » Inspector** from the menus, or by pressing the **F11** shortcut key. There are 2 advantages to using the Object Inspector, the first is that it can remain open, you simply click once on any object to display and edit its attributes in the inspector, the other advantage is that it can be used to edit multiple objects, including different kinds of objects. As you add objects to the current selection, only attributes that are common to the selected objects will be displayed. Any changes you make are applied to the object as soon as you click outside the attribute being modified.



Edit multiple objects in the Inspector

Querying for Objects

Protel DXP brings a powerful new approach to targeting objects in your design – queries. A query is a combination of symbols — keywords, object identifiers, operators, and values – that is analyzed, and then applied to every object in the workspace to see if the object complies with that query. The results of the query then become available for display in the workspace.

Queries can be simple, such as:

OnTop

Where the OnTop keyword targets all objects on the Top Layer of a PCB. Queries can also be very specific, such as:

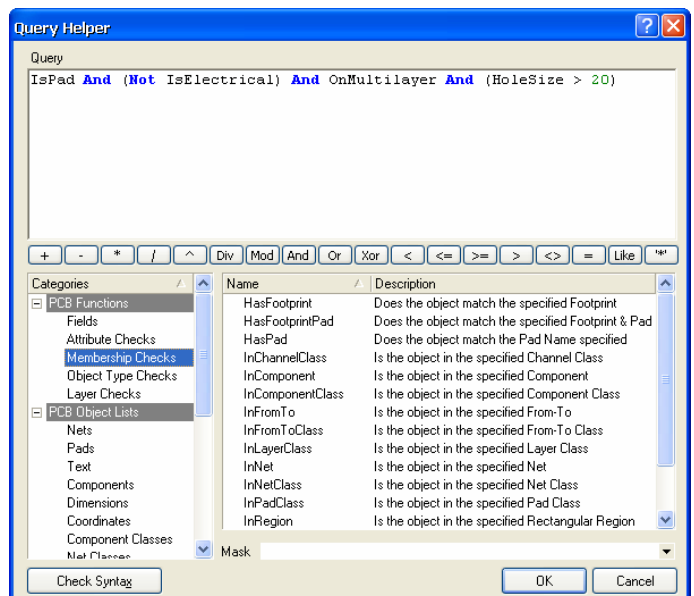
IsPad And (Not IsElectrical) And OnMultilayer And (HoleSize > 20)

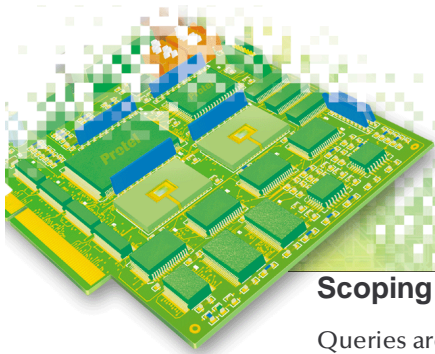
Which targets pads (**IsPad**), that do not have a net name assigned (**Not IsElectrical**), that are on the Multilayer (**OnMultilayer**) and have a hole size greater than 20mils (**HoleSize > 20**).

A complete list of keywords, object identifiers and operators are available in the *Query Helper* dialog. Use the Query Helper to build queries and check their syntax before applying them. To open the Query Helper click the **Helper** button in the *List* panel, or the **Query Builder** button in any of the design rule setups.

Filtering with Queries

Queries can be used for filtering, giving you the ability to precisely target a set of objects in the workspace. You can filter using queries in the *List* panel, or in the *Filter* Toolbar. Once you have applied a filter to target the objects, you can easily view and edit those objects. For example, an earlier image shows how the panel can be used to filter for a net, this filter could also have been applied by typing **InNet('NetName')** in the Filter Editor or the *Filter* Toolbar (where NetName is the name of the net you wish to filter for).





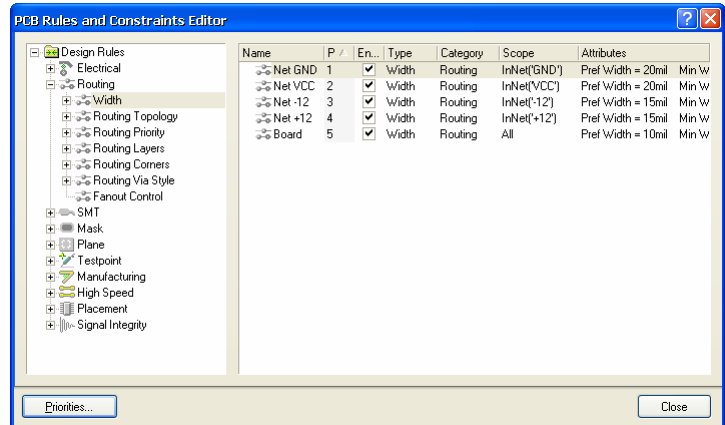
ProtelDXP™

Scoping Design Rules with Queries

Queries are also used to scope design rules, the query defines what the rule will target. To simplify the process of setting up rule scopes, each rule has a set of standard options (All, Net, Net Class, Layer, Net and Layer), selecting one of these will build a query automatically. There is also an Advanced option – if you use this option you write your own query.

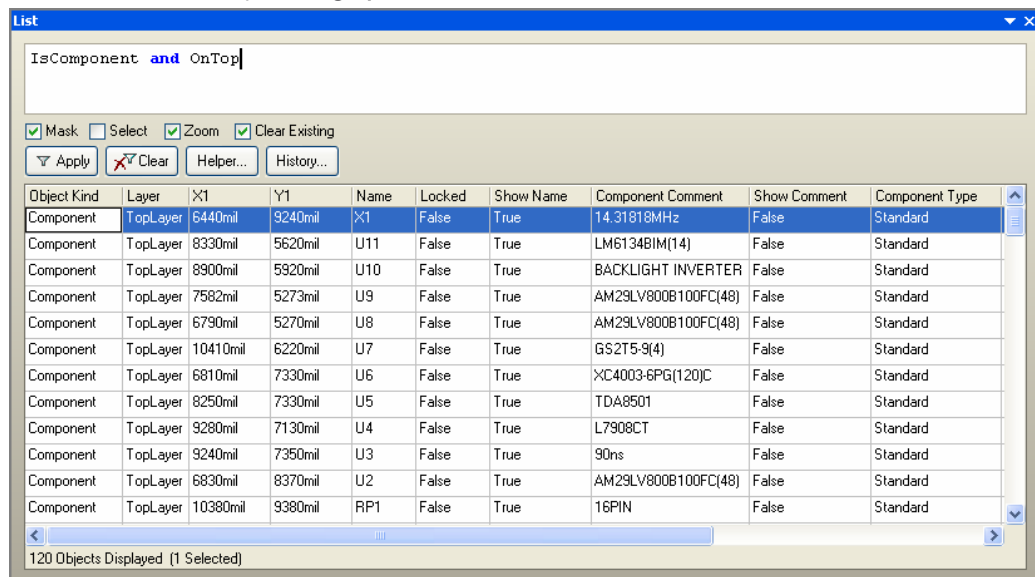
Rule Priority

The order that rules are applied is defined by their priority. The priority is displayed in the main window of the Design Rules dialog, if you navigate down using the tree on the left, when you select a rule Type (for example Width) you can click the Priorities button at the bottom of the dialog and change the priority. The highest priority rule is priority 1.



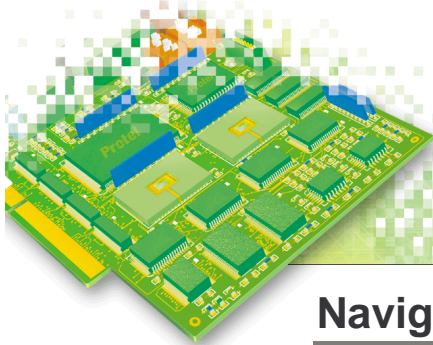
The List Panel

As well as the traditional graphical view, where the objects are arranged in the workspace as you placed them, you can also view and edit objects in the new *List* panel. The *List* panel is a combination of a query editor, highlight controls, and a spreadsheet. When you type in a query at the top of the panel and enable the **Mask** highlighting option, the contents of the spreadsheet are automatically updated to remove all objects filtered out by the query. The spreadsheet is a genuine alternate view – any action you perform in the spreadsheet, such as selecting a group of objects, or editing an object, occurs simultaneously in the graphical view.



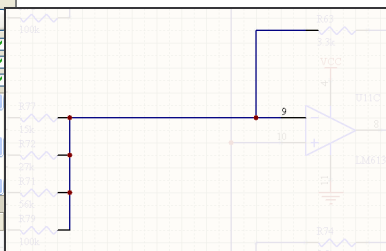
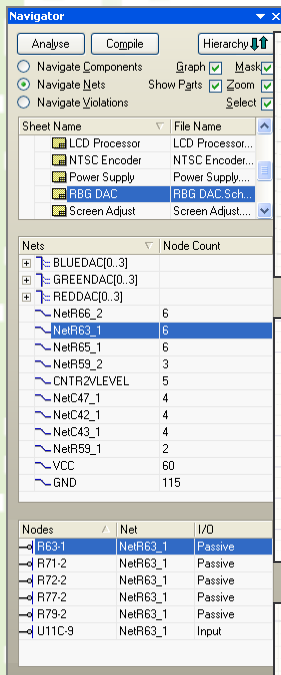
Tips

- Queries typed in the List panel apply in both the graphical view and the spreadsheet.
- Build a selection in the spreadsheet using the standard SHIFT+click and CTRL+click key combinations, then right-click on a selected attribute and Remove Non-Selected to reduce the list to only selected objects. You can also chose Edit Selected to edit this attribute in all selected items.
- Right click to create a report from the spreadsheet, from the Report Preview window you can export to a text, CSV, or Excel spreadsheet file.

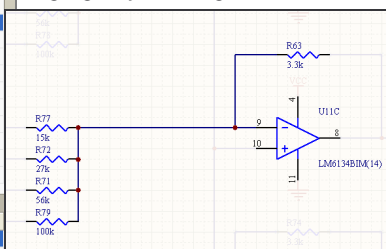


Navigating the Design

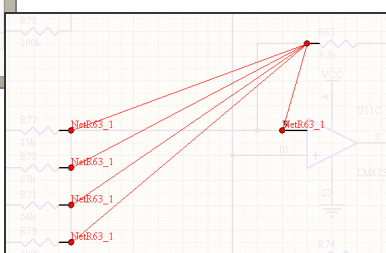
Both the schematic and PCB editors have new *Navigator* panels. These panels are used to browse your design, by components, nets, or errors (and rules as well in PCB). The Navigator panels use the new workspace masking feature to fade away all objects, except those that you have clicked on in the panel.



Highlight by Masking



with Show Parts enabled



graph the net, with parts not shown

Schematic Navigator Panel

Prior to compiling the design, the schematic is a collection of objects – components, wires, buses, ports, and so on. The net related objects define how the connections in the design must be created, but until the design is compiled there is no connective view, or model, of the design.

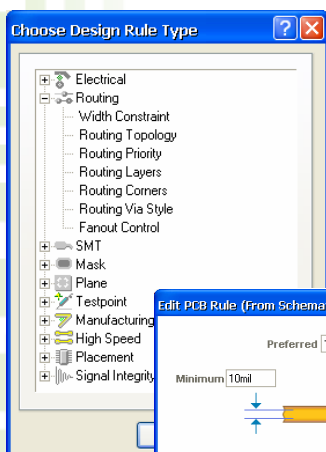
When the design is compiled a connective model is created – and the schematic editor Navigator panel can then be used to navigate the design, using this connective model.

There are a number of highlight options at the top of the Navigator panel. The *Mask* option fades unrelated objects as you browse, and the *Graph* option displays the connective relationship between objects – red when navigating by nets, green when navigating by components.

As well as navigating in a panel centric mode, where you click on the object of interest in the panel, if you click the **Hierarchy** button on the panel you can then navigate directly in the sheet. As you click on an object in the sheet it will be highlighted according to the options at the top of the panel. For example, enable the *Graph* option, click **Hierarchy**, then click on a wire to examine the connectivity created by that wire. You can trace a net off sheet by clicking the small red diamond that appears on a highlighted port.

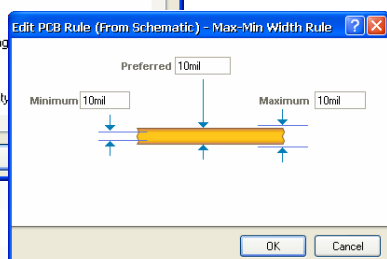
For a more comprehensive view of the compiled design, use the *Compiled* panel. As well as navigating by net or component, you can also examine the elements that make up each compiled entity.

Defining Design Rules in the Schematic

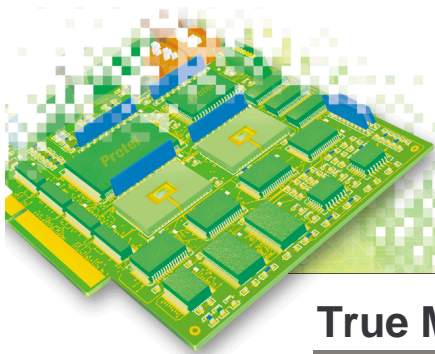


PCB specific design rules can be defined in the schematic, they are added as parameters. Whenever you add a parameter to an object there is an **Add as Rule...** button. If this is clicked the resulting *Parameter Properties* dialog will include an **Edit Rule Values...** button, click this to display the *Choose Design Rule* dialog, where the rule type can be chosen, and then configured in an edit rule dialog.

In the PCB Editor the scope of a rule – the set of objects that the rule will target – is defined in the rule. In the schematic editor the scope of the rule is defined by where the parameter is added. The following schematic parameter-to-PCB rule scope options are supported:



- Pin parameter – pad scope
- Port parameter – net scope
- Parameter Set object on a wire – net scope
- Parameter Set object on a bus – net class scope
- Component parameter – component scope
- Sheet symbol parameter – component class scope
- Sheet parameter – All scope



True Multi-channel Design

How often has your design includes a section of the circuit that is repeated – repeated twice, 4 times, or perhaps 32 times. Protel DXP includes a number of features that bring full support for multi-channel design – including sheet symbol instantiation, multi-channel designator management, automatic component class and room creation, and channel placement and routing replication.

True multi-channel design means that you only draw the channel schematic once – there is no need to create multiple copies of the sheet. The schematic project remains in this state, even after you transfer the design to PCB layout. The channel can be referenced by multiple sheet symbols, or you can use the new sheet symbol instantiation syntax to reference all channels from a single sheet symbol.

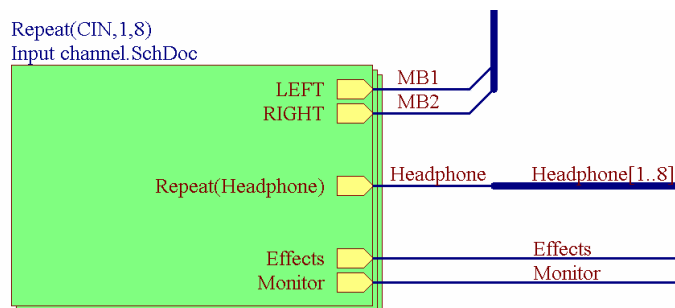
The Repeat keyword is used to define the number of times the channel is to be instantiated. When the project is compiled the compiler instantiates the channel the required number of times as it builds the internal compiled model, using a simple annotation scheme to uniquely identify each component in each channel. It does not duplicate the channel sub-sheet, instead it creates a table of designators. Once the project is compiled a Tab appears at the bottom of the Schematic Editor window, with one Tab for each channel of that sub-sheet.

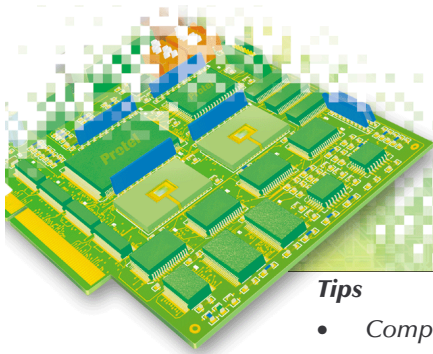
Multi-channel support flows though to the PCB Editor. The components are placed in their channel groups (component classes), with each component class on the board being placed in a placement room.

After you place the components in one channel you can use the **Autoroute » Room** command to route the channel, then select the **Tools » Rooms » Copy Room Formats** command to step and repeat the placement and routing across all the channels.

Creating a Multi-Channel Design

1. Draw the channel on a sub-sheet.
2. Place a sheet symbol on the parent sheet to represent this sub-sheet.
3. Define the number of channels – this is done by including the Repeat keyword in the Sheet Symbol Name field, as shown in the adjacent figure. In this example the Name of the sub-sheet is **CIN**, and the channels are to be numbered **1 to 8**.
4. Nets that are common to all sub-sheet are connected in the normal way, for example the Effects net.
5. Nets that connect individually to each sub-sheet are bought in as a bus, with one bus element connecting to each sub-sheet. You will note that the net label on the wire does not include a bus element number, and the sheet symbol includes the Repeat keyword. When the design is compiled this bus is resolved into the individual nets Headphone1, Headphone2, thru to Headphone8, with one assigned to each channel.
6. Once the design has been compiled it is transferred to the PCB Editor in the normal way. The transfer process will automatically create a component class for each schematic sheet in the design, a room for each component class, and group the components in each class in their room, ready for placement.
7. After placing and routing one channel select **Tools » Copy Room Formats** in the PCB Editor to copy the placement and routing of that channel to the other channels.
8. Select **View » Channels** from the **Project** menu to examine the designators assignments for the project.





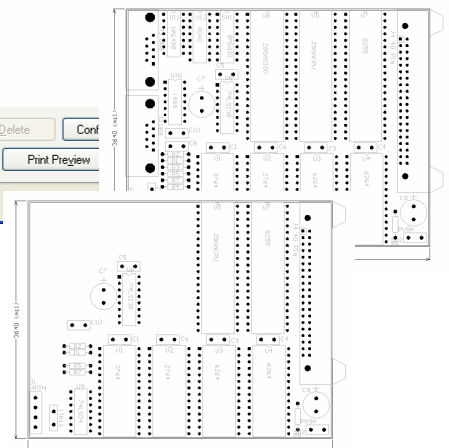
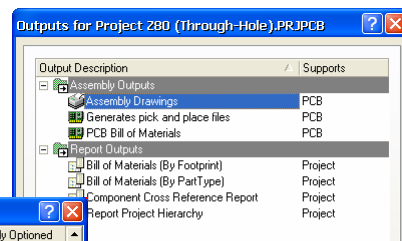
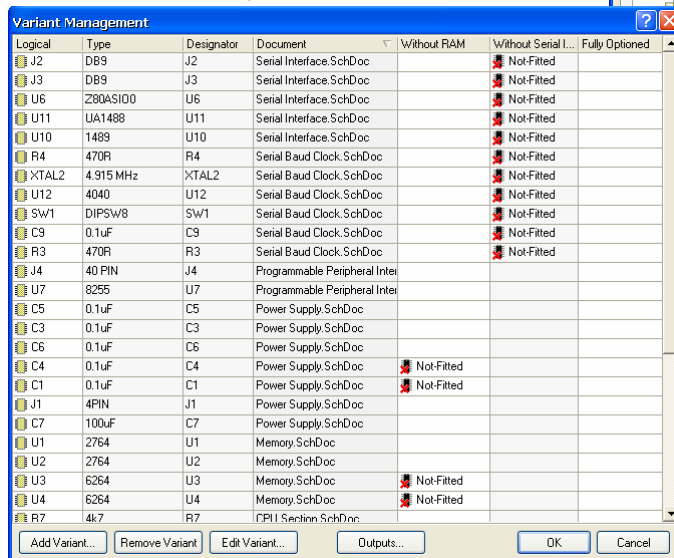
Tips

- Compile the Multi-channel Mixer example to see how a multi-channel design works.
- The name of the sheet symbol is used to uniquely identify each component in each channel, for example R1 in channel 8 would become R1_CIN8. Use short names for the sheet symbols to keep the designators short.
- Remember that there is always only one schematic sheet of the channel, the designator assignments for each channel are stored in a table (**Project » Channels**). When the multi-channel design is compiled there is still only one sheet shown, the difference now is that there are Tabs along the bottom of the sheet in the Schematic Editor, one for each channel.

Designing Assembly Variants

Protel DXP supports assembly variants. An assembly variant is created when you wish to design one PCB, but load it in different configurations – each configuration being an assembly variant.

The typical process is to first complete the full design and lay out the PCB. Once it is complete, and the schematic and PCB synchronized, you are ready to define the assembly variants.



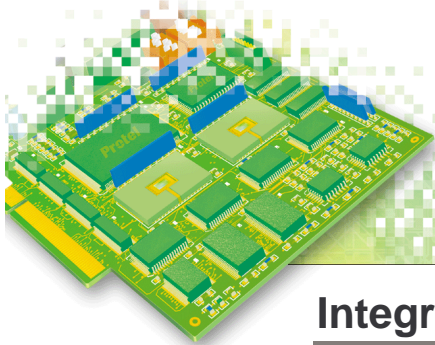
Variants are created and managed in the Variant Management dialog, from there you generate variant specific output, such as assembly drawings

Select **Project » Variants** to create an assembly variant. The Variant Management dialog shows the set of components for the entire design. Once you have added in the required variants and set the **Not Fitted** option for those components that are to be left off in that variant, you can generate the variant specific outputs, including the Bill of Materials, the pick and place file, and the assembly drawing.

Interfacing to a Version Control System

Version Control – interface directly from the Protel DXP interface to popular 3rd party version control systems, including Visual Source Safe®. Right-click in the *Project* panel to add a project to the version control system, or select Version Control in the project menu for a complete set of features.

The VCS interface in Protel DXP complies with the Microsoft SCC interface standard, giving direct access to an SCC compliant VCS from within the DXP environment.

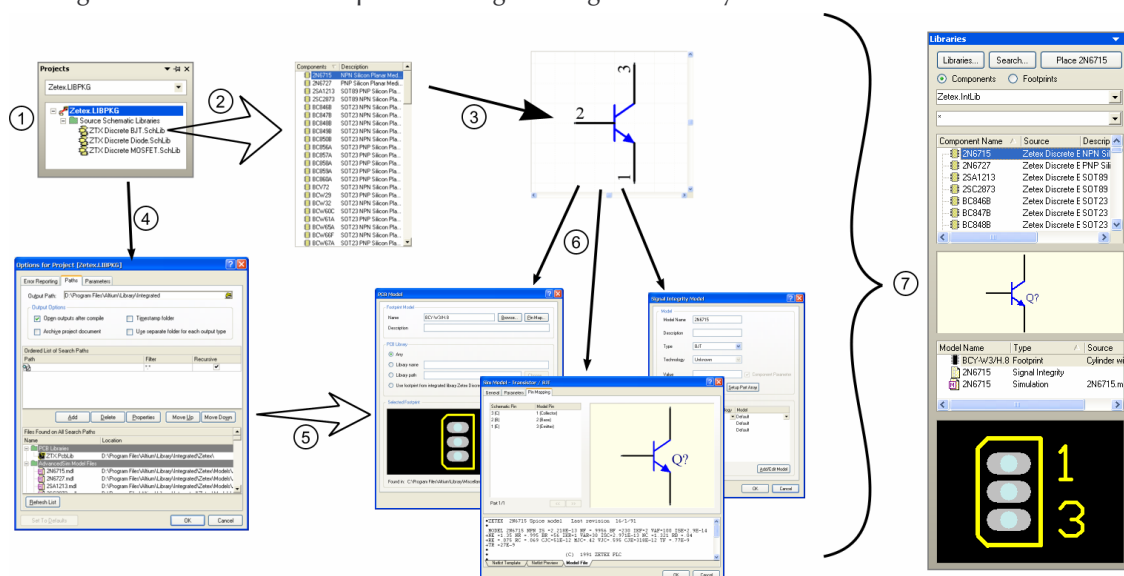


Integrated Component Libraries

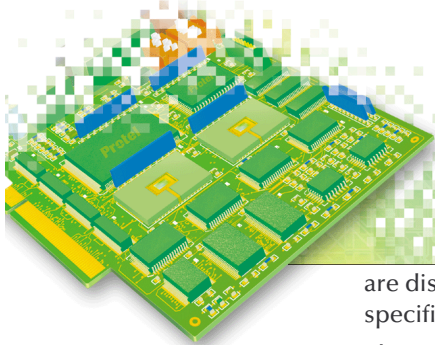
An integrated component library is a complete and portable package, it can include the schematic symbols, the PCB footprints, the SPICE models and signal integrity models, with the ability for further model types to be added in the future.

To build an integrated component library you start by creating a *library package*. The library package is a project file that defines what components are in the integrated library. Once the library package is complete, you compile it to produce the integrated library.

The Figure below shows the steps to creating an integrated library.



1. Create the source Library Package.
2. Once the library package has been created add in the schematic symbol libraries (**Projects » Add to Project**). There is no limit on the number of schematic libraries that can be added in. While Protel 99 SE schematic libraries can be included in DXP's Installed libraries list, they can not be compiled into an integrated library – open and save them in Protel DXP first.
3. Create/edit the component symbols in each schematic library. Each schematic component is created in its entirety – draw the graphics and place the pins for each part in the component. If there are pins that are on the component that you do not want displayed, place them in the library editor and set their display state to hidden.
4. Configure the search paths to the models in the **Options for Project** dialog. Note that the current location of the Library Package file is automatically included as a search path, add in others to point to the location of your footprints and other models. After adding in a new path click the Refresh list to confirm that the models are being located correctly. You can also add model files into the library package.
5. For each component, select **Tools » Edit Component** in the library editor menus to link in the appropriate models. This can include the PCB footprint, a SPICE model, or a signal integrity model. Note that each model is normally referenced by name only, when you compile the library the compiler looks down the search path for each model, and compiles it into the library. You can specify the exact location of each model if required. When you compile the library package the search order for models is – exact location; models included in the library package; models in the installed library list; then models down the search path.
6. When the library is complete and ready to compile, select **Project » Compile Integrated Library** from the menus. The integrated library is compiled from the various source libraries and model files. The compiler checks are configured in the **Options for Project** dialog, and warnings/errors



ProtelDXP™

are displayed in the *Messages* panel. The compiled integrated library is written to the location specified in the Output Path field in the **Options for Project** dialog.

7. The integrated library is now ready for use. If the **Open outputs after compile** option is enabled it will automatically be added to the installed library list.

Structuring Your Libraries

The Library Package/Integrated Library structure is flexible, you can add in as few or as many schematic libraries as you like. The integrated libraries supplied with Protel DXP are organized with each being like a manufacture's databook – a schematic library has been created for a series of components, then this library is added into the library package.

Signal Integrity Models

The signal integrity simulator uses pin models rather than component models, to configure a component for signal integrity simulation you either set the Type and Technology options, which will use default pin models, or import and IBIS model.

Accessing the Libraries

In Protel DXP all library types, including schematic PCB and integrated libraries are accessed through the *Libraries* panel.

Tips

- *You do not have to use integrated libraries, Protel DXP supports independent schematic and PCB libraries, Both Protel 99 SE libraries and Protel DXP libraries can be used, add these into the Library List, then access the symbols/footprints via the Libraries panel.*

Situs Topological Autorouter

Design rule compliance

Situs offer complete design rule compliance for all electrical and routing design rules, including via style and blind and buried vias.

BGA Packaging Support

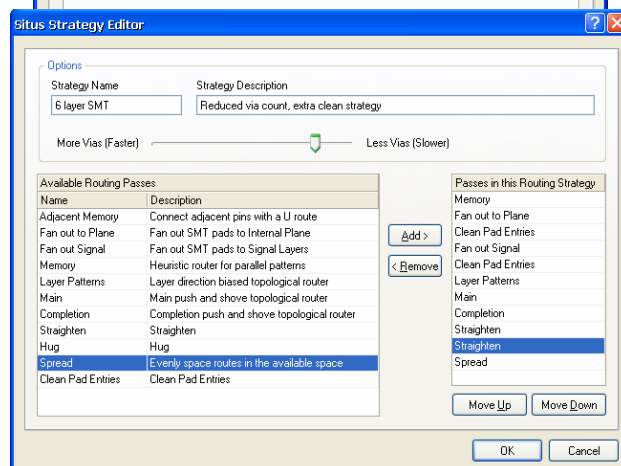
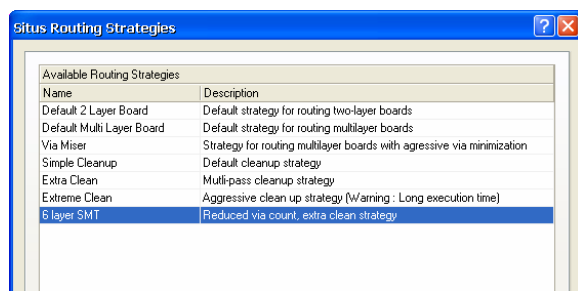
It includes built in strategies for BGA component routing, supporting an unlimited number of BGA pads.

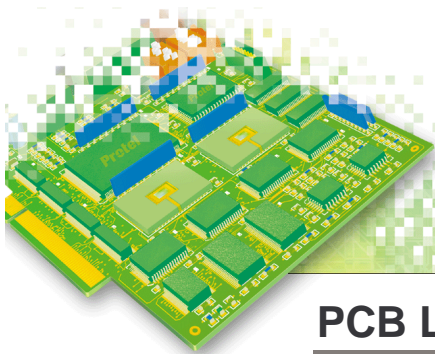
There is also a new Fanout Control design rule, use this to control the fan out requirements of the multi-pin surface mount components in your design.

User definable Routing Strategies

Situs comes with a number of default routing strategies, each focused to route well in a specific situation. Select the preferred routing strategy before launching the router.

The default strategies cannot be edited. They can be duplicated, and any number of user defined strategies can be created.





PCB Layout Enhancements

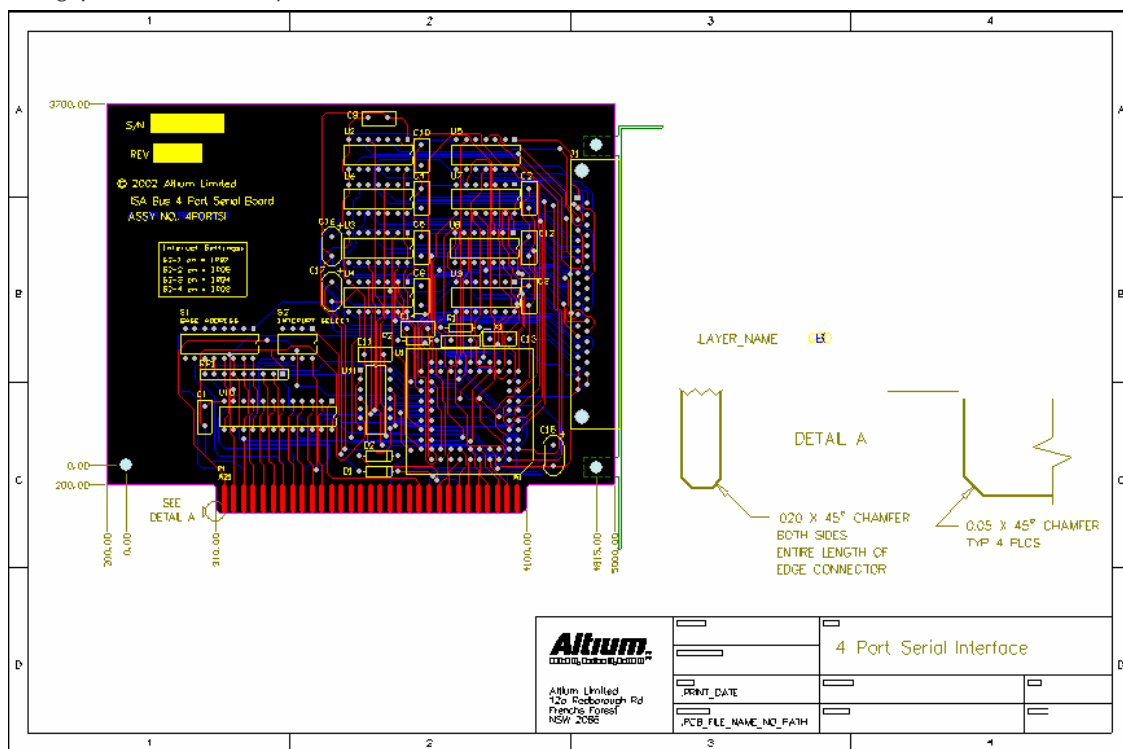
Board Shape

The board shape defines the boundary, or extents of the board. It is used by Protel DXP to determine the extents of the power planes for power plane edge pullback, and for calculating the board edge when outputting design data to other tools, such as the 3D viewer.

When a new board file is created a default board shape is also created. This can be resized, or redefined using the commands in the **Design » Board Shape** sub-menu.

Tips

- Press the Spacebar to change the corner style while you are defining the board shape.
- If your design already includes a board boundary defined on a mechanical layer, use the Define from Selected Objects command to automatically match the board shape to your shape. Make sure that the shape defined by the selected objects on the mechanical layer is closed – there is no gaps in the boundary.



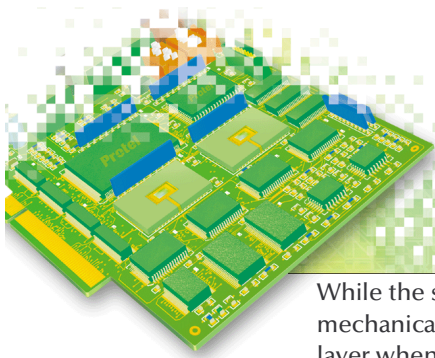
The board shape defines the edge of the board, and is used for automated functions such as the power plane pullback.

Sheet Templates

If you open the PCB example files supplied with Protel DXP you will notice that most of them display the board on a sheet, and that the sheet includes a border, grid reference, and title block.

Rather than use special objects to implement the template; the border, grid references and title block are drawn on one of the mechanical layers (Mechanical16 in the examples).

The sheet itself – the white region – is a special drawing feature. It is controlled using the options in the *Board Options* dialog. The sheet can be associated with mechanical layers – by placing objects on mechanical layers you can then create any sort of drawing template you require. Once the template elements are placed on a mechanical layer, the layer can be Linked to the Sheet in the *Board Layers* dialog.



Protel DXP™

While the sheet size and location can be defined manually, if you have linked the sheet to a mechanical layer the white sheet background is resized automatically to fit the objects on the linked layer when you select **View » Fit Sheet** from the menus.

Tips

- *Protel DXP includes a set of pre-defined PCB templates (Altium\Templates), open the required template and copy the contents from it and paste them into your board file.*
- *The sheet can hidden at any time, disable the **Display Sheet** option in the Board Options dialog. All linked mechanical layers will also be hidden.*
- *If you set the workspace start and end colors to black (Board Layers dialog) you can easily switch from the new sheet mode to the tradition full black mode by switching the sheet display on and off in the Board Options dialog.*

New Rule Scoping System

You create design rules to define the requirements of your design. Each rule has a rule scope, the scope defines exactly what the rule is to apply to, such as all objects on the board, or all objects in a particular net. Rather than using a fixed, predefined set of rule scopes, Protel DXP uses queries to define what objects a rule is to apply to. To make the scoping system more flexible, the order that rules are applied (the rule precedence), is user-definable.

For information on using queries to scope rules, refer to the *Querying for Objects* section earlier in this document.

Component Placement Rooms

Protel DXP has enhanced component placement room functionality, including: polygonal rooms, copy room formatting, creating a room to fit a component class, and autorouting and unrouting in a room. The set of tools for working with rooms are detailed below. Watch the status bar when using each of the tools:

Place Rectangular Room – place a rectangular room on the top or bottom side of the board (**Tools » Rooms** menu).

Place Polygonal Room – place a polygonal room on the top or bottom side of the board (**Tools » Rooms** menu).

Copy Room Formats – copy the placement and routing in the selected room, to other rooms that contain an identical set of components. Use this feature in a multi-channel design, after placing and routing one channel Protel DXP will automatically place and route the other channels (**Tools » Rooms** menu).

Wrap Room Around Components – if you have already placed the components in a component class use this command to automatically wrap the room around the components.

Create Non-Orthogonal Room from selected components – create a component class from the selected components, then create a non-orthogonal room around them (**Tools » Rooms** menu).

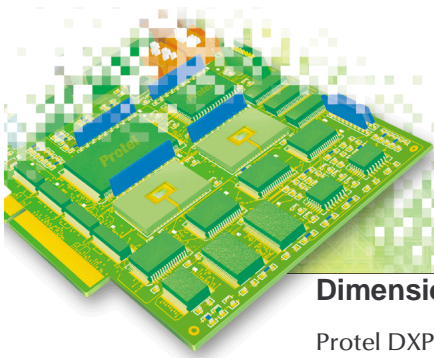
Create Orthogonal Room from selected components – create a component class from the selected components, then create an orthogonal room around them (**Tools » Rooms** menu).

Create Rectangle Room from selected components – create a component class from the selected components, then create a rectangular room around them (**Tools » Rooms** menu).

Autoroute room – autoroute all connections that start and end within a room (**Autoroute** menu).

Un-route room – un-route all connections that have at least one node in a room (**Tools » Un-Route** menu).

Slice a room – use this to slice one room into 2 rooms (**Edit » Slice** menu).



ProtelDXP™

Dimensioning tools

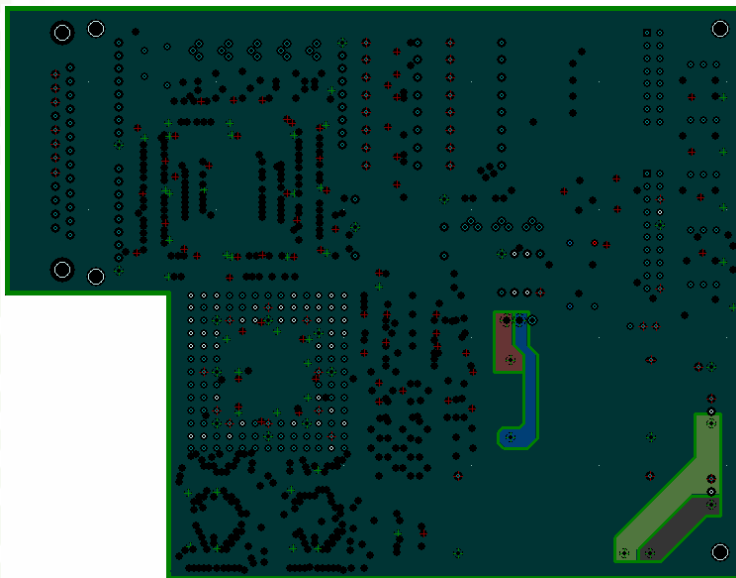
Protel DXP includes a comprehensive set of dimensioning tools, including: Linear, Datum, Baseline, Leader, Angular, Center, Radial, Linear Diameter, Radial Diameter. Dimensions can be placed from the **Dimensions** sub menu, or the *Dimensions* toolbar.

Dimensions are associative, once they are connected to an object they will remain connected to the object as it is moved. To associate a dimension to an object, such as a track end, look for the small hollow circle drawn in the same color as the layer being placed on, this circle indicates that the dimension will associate with this point. Each dimension type also offers complete control over the units, the precision, the text alignment, and the arrow characteristics – double click on a dimension to edit any of these parameters.

Keep an eye on the Status bar when you are placing a dimension – each type has a different placement sequence, the Status bar will prompt you what to do next as you place the dimension. For information on a particular dimensioning tool press F1 when the mouse is over a dimension menu entry or toolbar button.



Enhanced Splitting of Power Planes



DXP's new power plane splitting feature works like cutting with a knife. To split a plane make the plane the current layer, and switch to single layer mode (**Shift+S** shortcut toggles single layer mode).

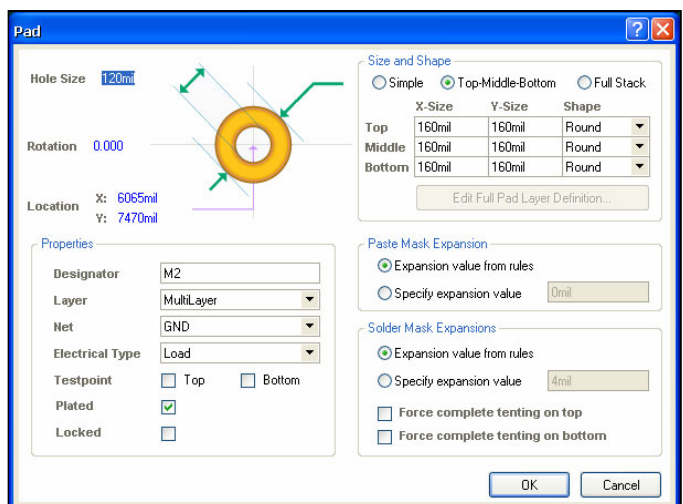
Select **Place » Line** from the menus. Starting from just outside the board shape, place lines to define the cut, or separation, between the 2 regions you are splitting apart. The width of the line defines the no-copper separation, press the Tab key during line placement to change the width.

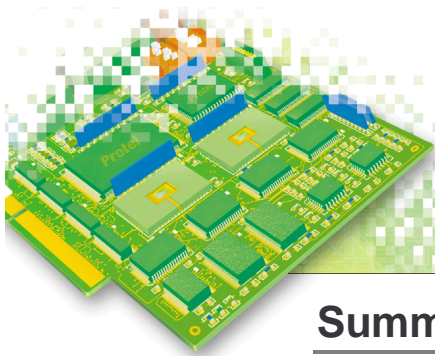
When you split a plane you must start from outside the board shape and finish outside the board shape. You can also cut within the plane, as long as you always start and finish on the edge or an existing cut.

Fully Definable Pad Shapes

Protel DXP supports 3 types of pad definition:

- Simple - where the pad properties are the same on all layers
- Top-Mid-Bottom – where the top layer and bottom layer are defined individually, and all mid-layers are defined as a set
- Full Stack – where the pad properties can be defined uniquely on every layer.





Summary of other Enhancements

Schematic Capture

User-definable component and pin parameters

Unlimited user-definable component parameters and pin parameters, editable in the library and on the schematic sheet.

On-sheet graphical component editing

Edit and move components pins on the schematic sheet. Double click on a component to unlock the pins and move them.

Circuit Simulation Enhancements

Import and export waveform data

Import and export plot points to a spreadsheet, and Copy the chart to the clipboard and paste into other Windows applications.

Enhanced plot graphics

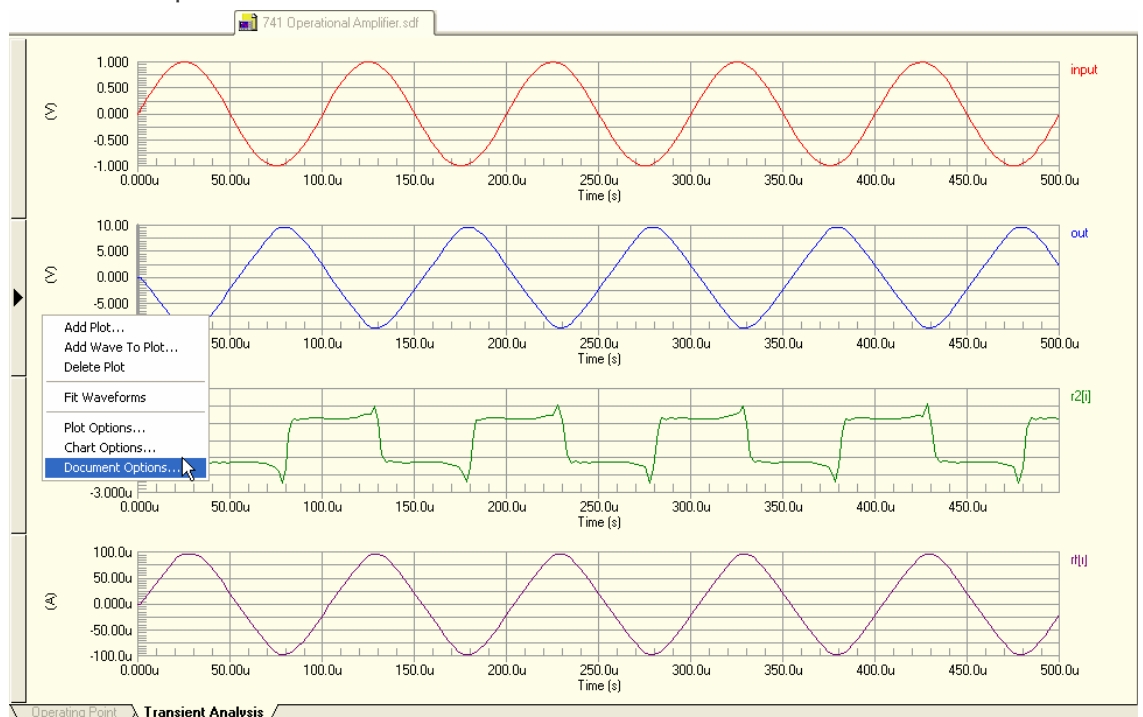
Improved access to plot elements such as titles, labels, lines, and so on.

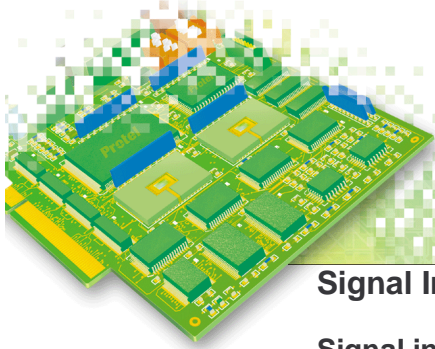
Overlay different waveforms

Different waveform types can share the same plot, with support for multiple Y axes.

Multiple scaled plots

Display from 1 to 4 scaled plots simultaneously, right-click and select Document Options to configure the number of plots.





ProtelDXP™

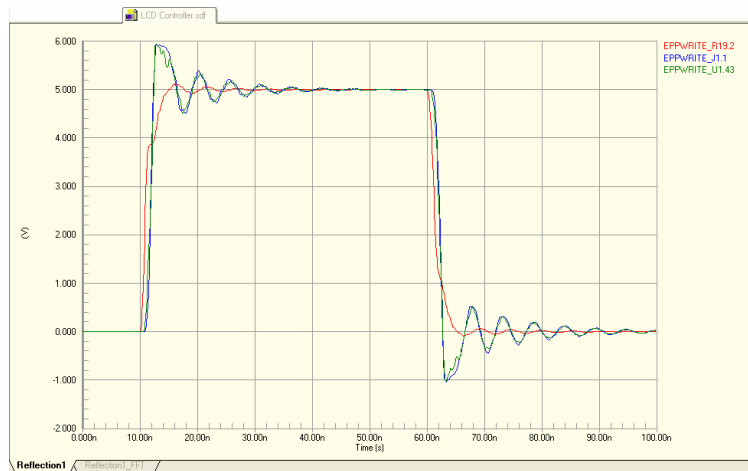
Signal Integrity Enhancements

Signal integrity analysis with the DXP environment

Analyze the SI performance of the PCB from either the schematic or the PCB editors, perform reflection and crosstalk analysis on selecting nets, and display the waveforms within the Design Explorer.

Signal integrity analysis from the schematic

You can also perform an SI analysis on the design from the schematic without having a PCB as part of the project. In this mode a default average route length can be defined, as well as the PCB stackup.



Enhanced model integration

SI models can be included in the new integrated component libraries.

Project Outputs

Generate ODB++ output directly from the PCB

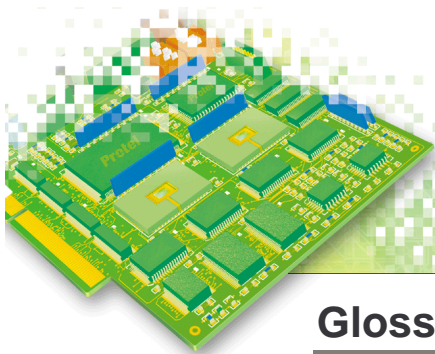
Automatically load outputs into CAMtastic.

Dual monitor support

Protel DXP takes full advantage of a dual monitor configuration. You can arrange the documents across both monitors, or you can use the second monitor for secondary views, such as the object *Inspector* Panel or the *Filter* Panel.

To position a document on the second monitor right-click on a document Tab and select Open in New Window to create a second DXP window, ready to position on your second monitor. Alternatively, you can drag a document Tab out when DXP is not maximized. Note that this second window is not a second instance of Protel DXP, it runs in the same MS Windows process.

To position an individual panel on the second monitor simply click and hold on the actual Panel name in the caption bar of the panel, and drag and position it. Hold the CTRL key as you drag it to prevent it from docking to the edge of application.



Glossary

Analyze document – compiles (netlists and validates) the selected document.

Compile Integrated Library – creates an integrated library that includes the schematic symbols, plus any models that have been linked to the symbols. Note that an integrated library can not be edited.

Compile Project – create a connective model (or netlist) of the design, and check the validity of this connective model. When a PCB project is compiled the connective model only exists in memory. The connective model can be examined in the *Compiled* Panel. If the project has already been compiled only those documents that have been changed since the last compile are recompiled.

Connective model – general term to describe the generic internal netlist that is created when a compile is performed.

Design View – general term used to describe any of the different views of the design, for example the set of sheets that make up the schematic project is one view of the design, the PCB is another view, and a netlist is another.

Filtering – a technique used to control what is available in the workspace views. Filtering can be done via the Navigator panel, via the Find Similar Objects dialog, or via a query. The display of the filtered result set depends on the highlight controls

Highlighting – wherever a filter can be applied you also have options to control how to highlight the result set. Highlighting options include: masking, selecting and zooming.

Inspector – the Inspector panel presents the properties of the current selection for editing. It can be used to edit an individual object, or multiple objects.

List panel – alternate view into the set of design objects in the current document. The List panel has 3 distinct areas; the query editor at the top, a set of highlighting controls, and the spreadsheet-like list view at the bottom.

Masking – a filter highlighting mode that effectively removes objects from the editable workspace. In the graphical view this 'removal' is done via fading, or dimming, all the objects that have been filtered out, leaving only the objects targeted by your filter shown at normal brightness. The brightness of filtered objects is controlled by adjusting the Mask Level (click the button at the bottom of the workspace). Masking in the *List* view is done by removing masked objects from the list.

Multi-channel design – technique of instantiating a section (sub-sheet) in a design multiple times.

Panel – a window that provides another view into a design, or the design environment. There are 2 types of panels, workspace panels that are available at any time, such as the *Files* panel or the *Projects* panel; and editor panels, which are only available when a document of that kind is visible, such as the schematic or PCB Navigator.

Query – method of targeting objects in the workspace. Uses a query engine to first parse the query string, then check each object to see if it complies with query or does not comply with the query.

Variant – variants are different assembly configurations of the one design. A variant of a design specifies the fitted and not fitted components in that variant. Applicable output documents, including the BOM, pick and place files, and assembly drawings are created for each variant. Note that the project documents always define the complete (or fully loaded) design, so there is only one set of fabrication documents for the PCB.

Version Control System (VCS) – document management system that stores electronic documents, providing access to them through a formal interface that tracks when documents are taken out of the VCS (checked-out), and when they are returned to the VCS (checked-in). The VCS tracks the changes each time a document is checked back in, either by saving a new copy, or saving the differences between the versions. The VCS allows you to restore (or rollback) a design back to any earlier version. Document access permissions can also be controlled in the VCS. The VCS interface in Protel DXP complies with the Microsoft SCC interface standard, giving direct access to a SCC compliant VCS from within the DXP environment.

Workspace Views – a workspace view is an interface where you can edit design objects. Protel DXP includes 3 workspace views, the main graphical view, the *List* view, and the *Inspector* panel.