



Welcome to the Altium Designer Environment

Summary

Guide

GU0112 (v1.0) April 29, 2005

Altium Designer brings a complete electronic product development environment to your PC's Desktop, providing multi-document editing and full customization of the design workspace. This guide provides an introduction to Altium Designer and an overview of its unique, singular design environment.

Altium Designer provides a unified electronic product development environment, catering for all aspects of the electronic development process, including:

- System Design and Capture
- Physical PCB Design
- FPGA Hardware Design
- Embedded Software Development
- Mixed-Signal Circuit Simulation
- Signal Integrity Analysis
- PCB Manufacturing
- FPGA system implementation and debugging (when working with a suitable FPGA development board, such as an Altium NanoBoard).

All of these design areas are intrinsic parts of a single, cohesive system, built on Altium Designer's Design Explorer (DXP) integration platform. The extent of this unified system, in terms of the features and functionality available, will depend on the specific licensing purchased.

Underlying Altium Designer is the DXP integration platform which brings together Altium Designer's various editors and software engines, and provides a consistent user-interface across all the tools and editors.

The Altium Designer environment is fully customizable, allowing you to set up the workspace to suit the way you work. A consistent selection and editing paradigm across different editors allows you to easily and smoothly switch between various design tasks within the Altium Designer environment.

The Altium Designer Environment

Altium Designer offers a unique design environment, with all areas of design – from capture to generation of PCB manufacturing output; from embedded software development to processing and download of an FPGA design into a physical FPGA device – brought together in the one environment through the underlying support of the DXP integration platform. Supporting the display and editing of multiple design documents, of differing type, this environment is fully customizable, allowing you to tailor the workspace resources - including menus, toolbars and shortcuts - in accordance with preferred design habits.

Figure 1 summarizes some of the key elements of the Altium Designer environment.

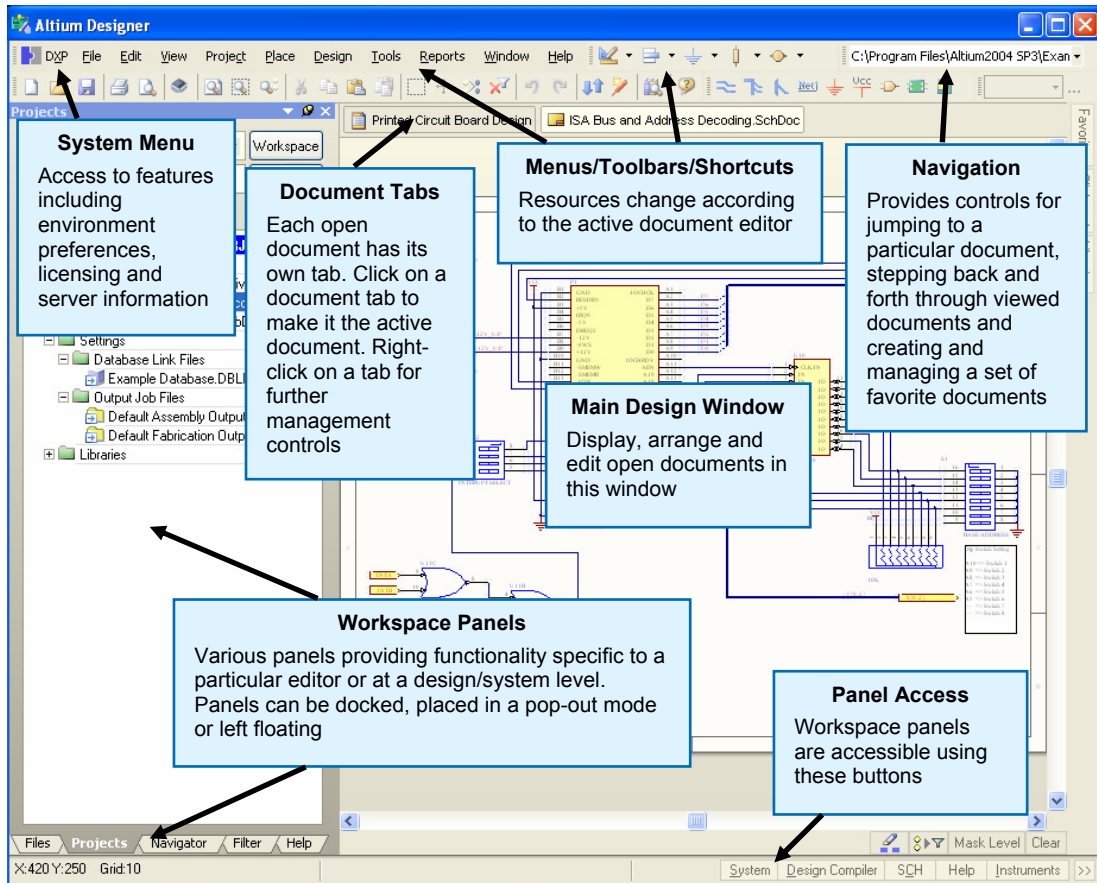


Figure 1. Altium Designer

The following sections take a closer look at these and some of the other key features and functionality that make up this environment.

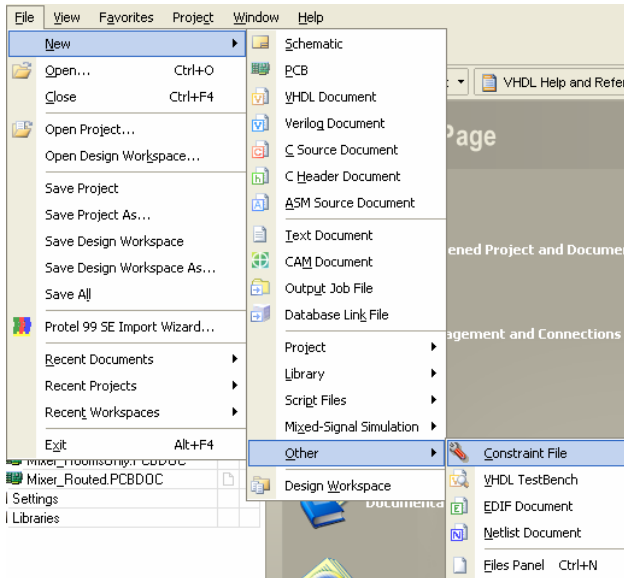
Working with Documents

In Altium Designer, each type of document is opened and edited in an associated editor. For example, a schematic document is opened and edited in the Schematic Editor, a PCB library document in the PCB Library Editor, and so on. As you create a new document or open an existing document, the associated editor for that document type will automatically become the active editor.

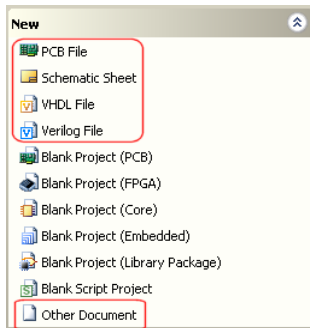
Creating new Documents

New documents can be created using one of the following three methods:

- select the required document type from the **File » New** sub-menu



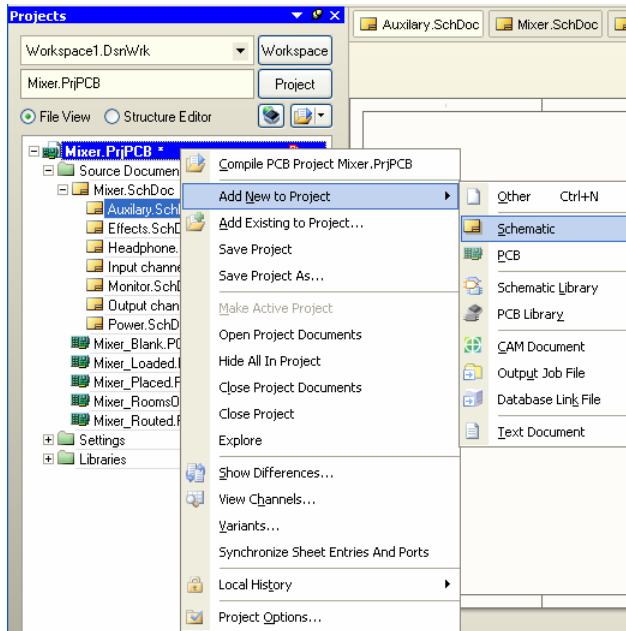
- click on the required entry in the New section of the **Files** panel. If this panel is not open, click on the **System** button at the bottom right of the main application window and choose **Files** from the pop-up menu that appears



If one or more projects are currently open in the **Projects** panel, the new document will be added to the active project. Alternatively, to add a new document directly to an open project, irrespective of whether

Welcome to the Altium Designer Environment

it is the active project or not, right-click on the project's entry in the **Projects** panel and choose the document type from the **Add New to Project** sub-menu. The menu will list the source documents typically added for that project type.



Opening and Displaying Documents

When you open a document, it becomes the active document in the application's main design window. Multiple documents may be opened simultaneously. Each open document has its own tab at the top of the design window, but only one document may be active within that window. Figure 2 shows three documents open – one PCB and two schematic – with the PCB document currently active.

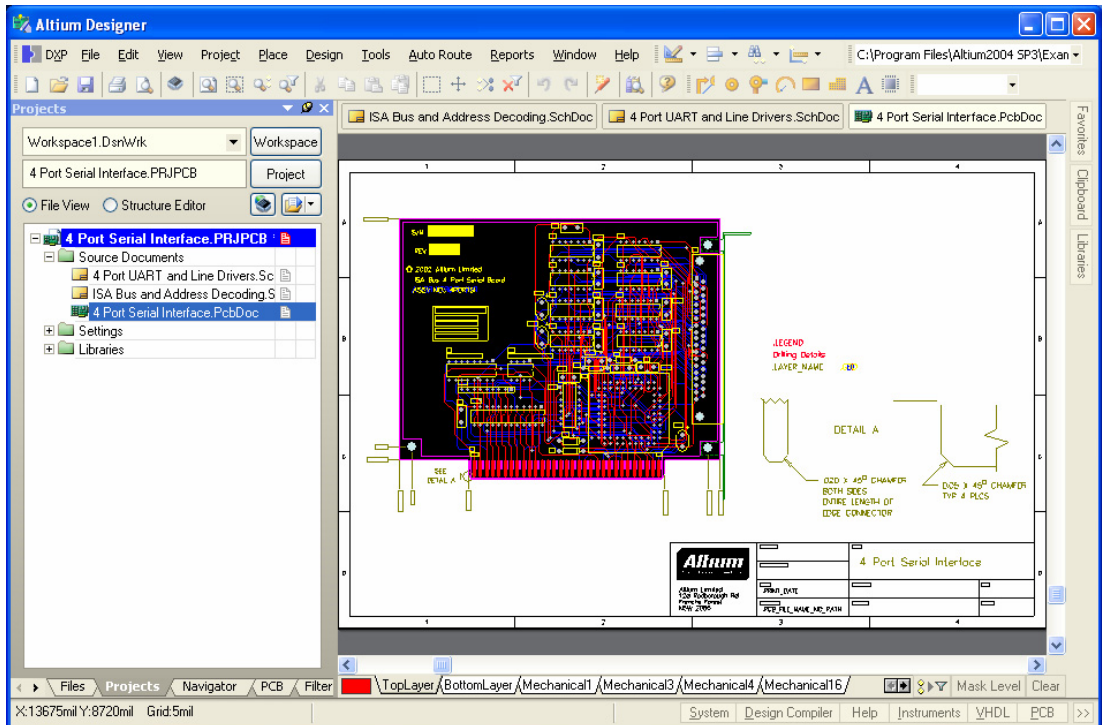


Figure 2. Multiple tabbed documents open in the main design window

The active document is distinguished by a highlighted appearance to its tab. To make another open document the active document, simply click on its tab. Alternatively, use the **Ctrl+Tab** and **Ctrl+Shift+Tab** keyboard shortcuts to cycle forward or backward through all open documents respectively.

Hiding Documents

Various features require that all source documents be compiled and, where necessary, this compilation is performed automatically. For this to happen, all such documents must be opened. Depending on the project, there may be a sizable number of source documents involved in the compilation. To have all documents open as tabbed documents in the main design window can create a frustratingly cluttered workspace. To this end, Altium Designer provides the ability to 'hide' documents. They still meet the requirement of being open, with respect to features such as compilation, cross probing and annotation, but are hidden with respect to display in the main design window.

Any open document can be placed in this hidden mode by:

- right-clicking on its tab and choosing the **Hide** command
- right-clicking on its entry in the **Projects** panel and choosing the **Hide** command
- using the **Hide** command available from the main **Window** menu, when the document is active in the main design window.

Welcome to the Altium Designer Environment

Hidden documents become listed in a drop-down at the far right of the Document bar which contains the document tabs (Figure 3).

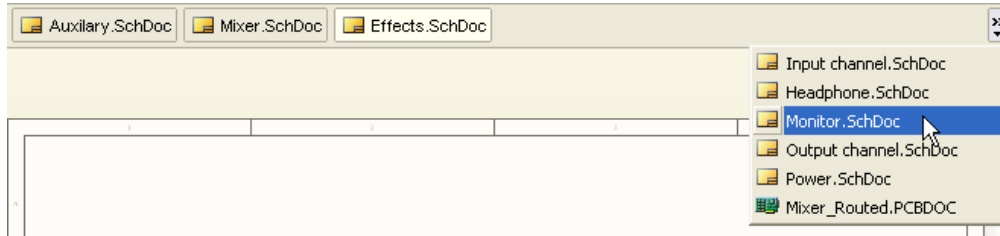


Figure 3. Accessing hidden documents

Clicking on an entry in the list will unhide the document and it will reappear as a tabbed document in the main design window. The main **Window** menu and the document's right-click menu in the **Projects** panel also provide commands to unhide the document.

Indicating Document Status

As part of the **General** options for the **Projects** panel, you can enable the iconic display of document open/modified status. This provides a quick visual summary, allowing you to see which documents are open, hidden or modified (Figure 4).

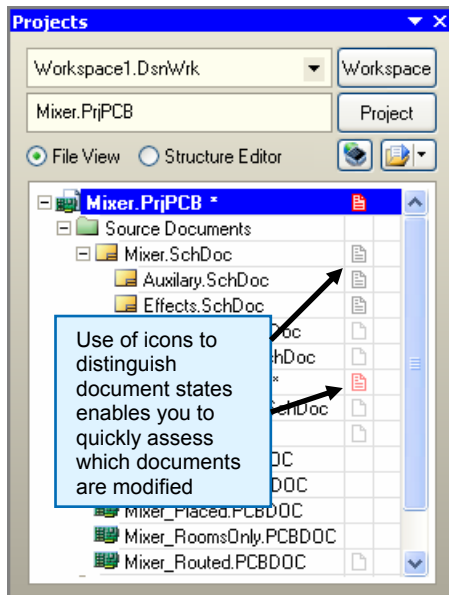






Figure 4. Iconic display of document status

The icons used are listed for ease of reference below. The entry in brackets shows the textual confirmation that appears when hovering the cursor over the icon.

-  (Open) - the document is open as a tabbed document in the main design window
-  (Hidden) - the document is hidden
-  (Open/Modified) - the document is open and has been modified (yet to be saved)
-  (Modified) - this icon appears next to the main project document only, to show that the project has been modified (yet to be saved)

A modified document, project or design workspace that has yet to be saved is also distinguished by an asterisk next to its entry in the panel. A modified document is also indicated by an asterisk inside its associated tab in the main design window.

Document Window Management

With Altium Designer, you are not limited to viewing and working with a single design document at a time. Various commands are provided that allow you to effectively manage the open documents and tailor their display to your preferred working habits.

Right-click on a document tab to access various commands for managing the document. These commands including closing, saving and hiding the document, as well as commands affecting the display of all open documents within the main design window (Figure 5).

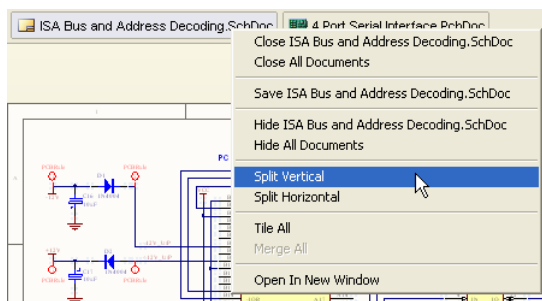


Figure 5. Accessing document management commands

Commands for splitting the main design window horizontally or vertically can prove invaluable when wishing to use cross probing with, for example, the schematic and PCB documents open side by side.

Welcome to the Altium Designer Environment

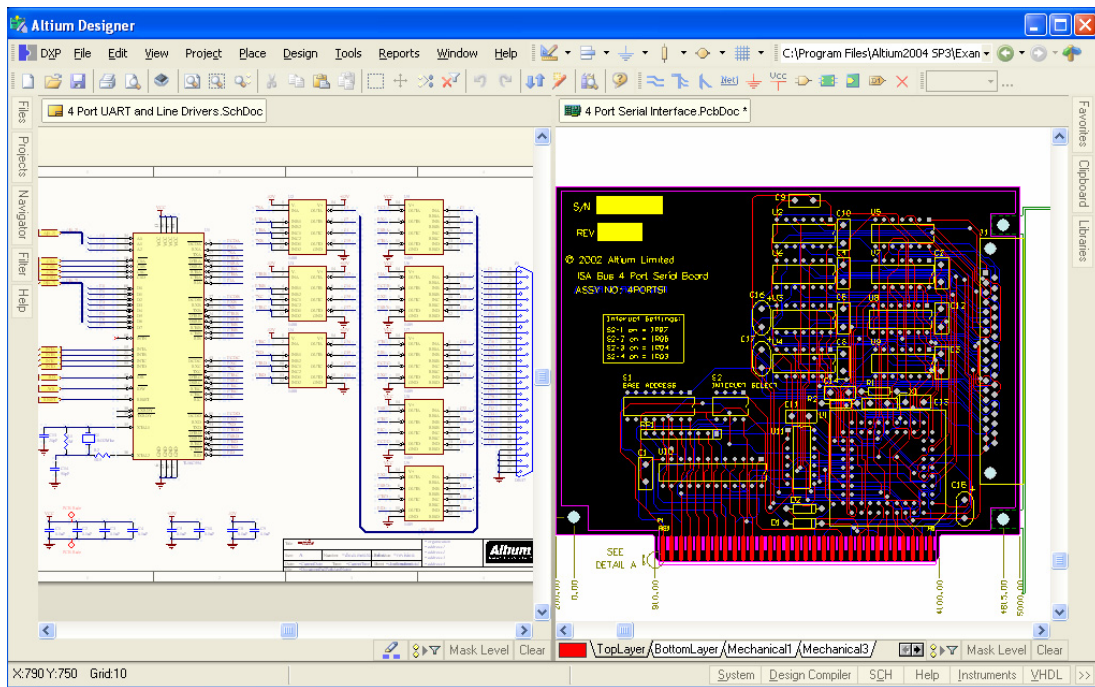


Figure 6. Splitting the main design window

When split, the regions are treated as though they were individual windows. One document may be active in each region, but only one document, across all regions, can have the focus at any given time. When a new document is added, or an existing document opened, it will be opened as part of the region in which the currently focused document resides.

Commands also allow you to tile the documents or merge them back into the single, default design window.

Should you wish, you can open a document in its own, separate design window. Simply right-click on its tab and choose the **Open In New Window** command. Alternatively, click on the document's tab and drag it onto the desktop area, outside of the main application window. Commands are then available from the main Windows menu to arrange the separate windows horizontally (Figure 7) or vertically.

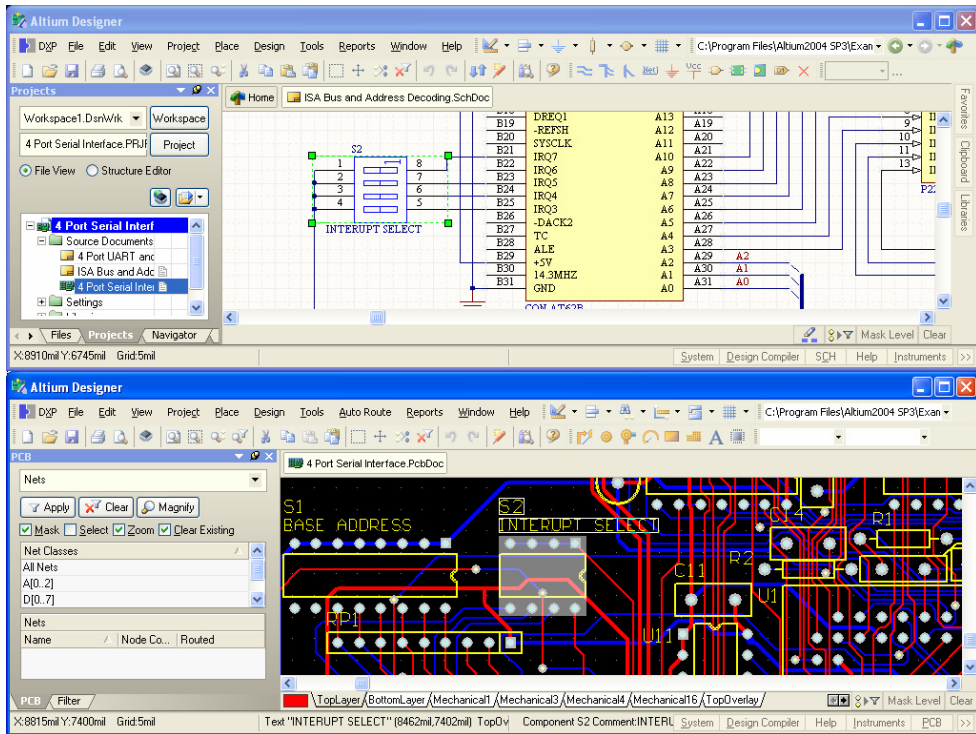
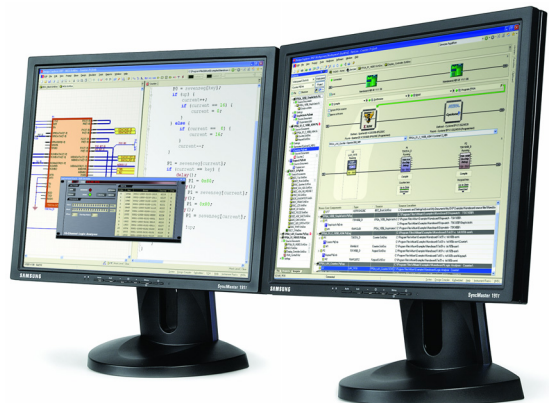


Figure 7. Horizontally arranging Altium Designer windows on the Desktop

Enlarging the Design Canvas – Dual Monitor Support

Altium Designer supports the use of dual monitors, with a recommended resolution of 1280x1024. With many documents open concurrently and a plethora of workspace panels, toolbars and support documents, this feature affords you space enough to define a more comfortable working environment. You might, for example, dedicate the use of one monitor solely for design, while arranging various workspace panels and additional documents on the other.



Organizing the Environment - Desktop Layouts

As a further aid in setting up the workspace environment, Altium Designer supports the notion of Desktop Layouts. This feature allows you to arrange the application's document windows, workspace panels and toolbars around the desktop as required and then save that layout to file. In this way, multiple users can quickly arrange the workspace to suit their design habits by loading in their preferred layout.

Commands for saving and loading layouts, including the default layout shipped with the software, are available from the **View » Desktop Layouts** sub-menu.

Document Storage

Altium Designer stores all design documents and generated output files on your hard disk as individual files. You can use the Windows Explorer to search for them. Design documents will typically be encapsulated into projects, both for management and to access certain design features, such as design verification, comparison and synchronization. A project file contains links to design documents, as well as any other project-level definitions.

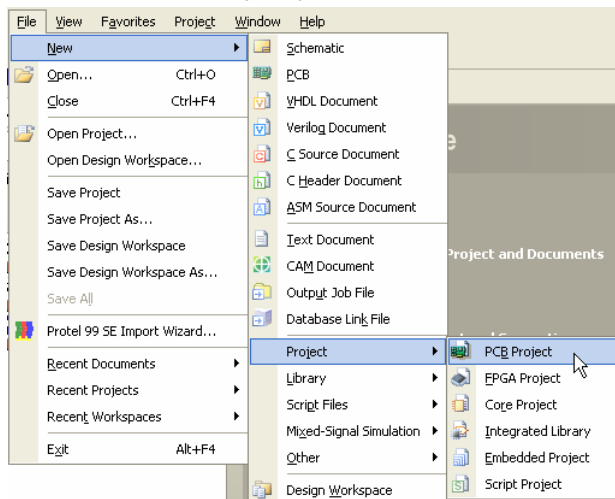
Projects

Altium Designer allows you to create and maintain the following project kinds:

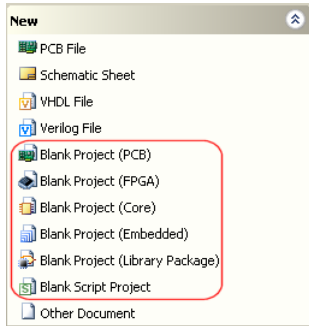
- PCB project (*.PrjPcb)
- FPGA project (*.PrjFpg)
- Core project (*.PrjCor)
- Embedded project (*.PrjEmb)
- Library package (*.LibPkg)
- Script project (*.PrjScr)

A new project of any of the above types can be created using one of the following three methods:

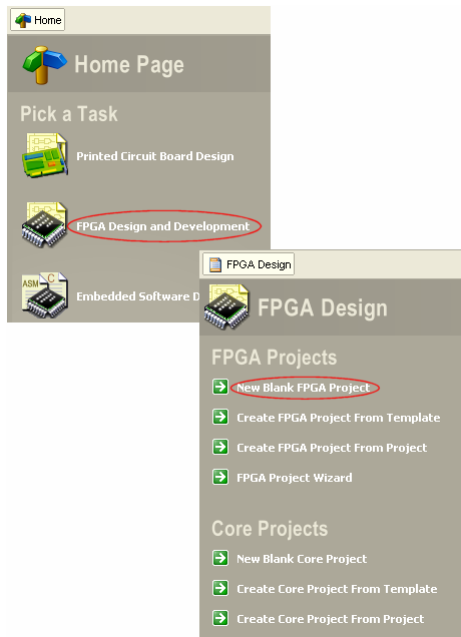
- select the required project type from the **File » New » Project** sub-menu



- click on the required entry in the New section of the **Files** panel. If this panel is not open, click on the **System** button at the bottom right of the main application window and choose **Files** from the pop-up menu that appears



- click on the associated design area link in the Pick A Task section of the Home Page (**View » Home Page**) and then click on the available entry to create a new blank project



As you open an existing project, or create a new one, an entry for it will appear in the **Projects** panel. Any existing documents that are part of a project and any new ones that may be added, will appear under sub-folders, according to their purpose and/or type.

Welcome to the Altium Designer Environment

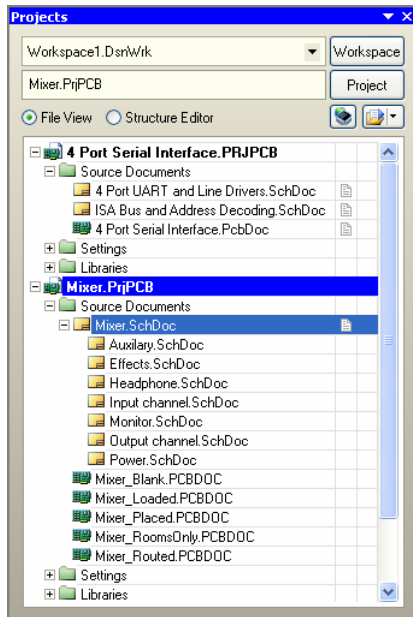


Figure 8. The Projects panel



For further information about projects, see the [Project Essentials](#) application note. This application note includes information on the role of projects in the design process, the different kinds of projects available, techniques for working on projects and how to use the Projects panel effectively.



For further information on the Projects panel, press **F1** when the cursor is over the panel.

Project File Management – The Storage Manager

To allow greater control over the management of files in projects, Altium Designer features a dedicated Storage Manager. Implemented as a workspace panel (see [Workspace Panels](#)) the Storage Manager can be accessed by clicking on the **System** button at the bottom of the application window and choosing **Storage Manager** from the subsequent pop-up menu that appears. Once opened, the Storage Manager presents a folder/file view of the active project's documents (Figure 9).

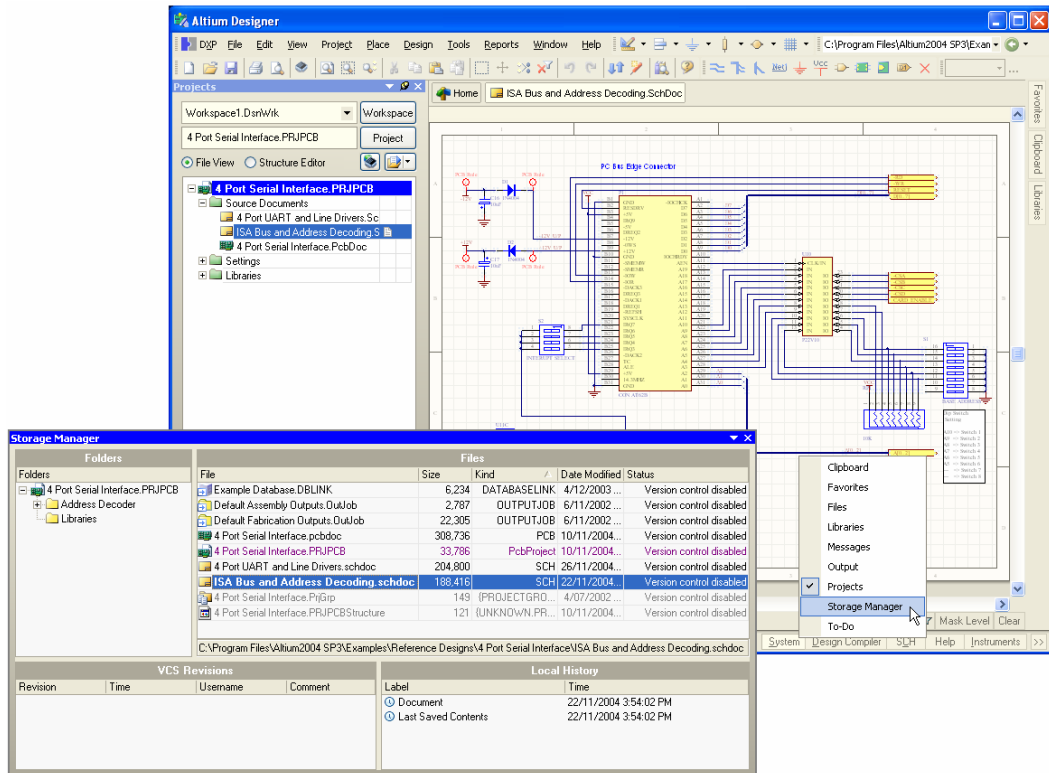


Figure 9. The Storage Manager panel

The Storage Manager can be used for:

- general everyday file management functions – for files in the project or within the active project's folder structure
- management of backups using the Local History feature (see [Document Management – Local History](#))
- as an SCC (Source Code Control) compliant version control interface for your projects
- as a CVS compliant (Concurrent Versions System) interface for your projects
- as an SVN (Subversion) compliant interface for your projects
- performing a physical and electrical comparison of any two versions in either the Local History or the VCS Revision list.



For further information on the Storage Manager panel, press **F1** when the cursor is over the panel.

Design Workspaces

As you can open and edit multiple design projects in Altium Designer, provision is made for you to be able to save a set of open projects as a Design Workspace (*.DsnWrk). This can prove to be a highly efficient option when working with multiple projects that are closely linked in some way. For example,

Welcome to the Altium Designer Environment

you may have a PCB design that incorporates one or more FPGA devices. The designs targeting those devices may include processor cores, within which, dedicated embedded software programs will run. Instead of having to remember and open the individual projects involved (PCB, FPGA and Embedded), all three projects can be saved, opened and manipulated as a single design workspace.

Document Management – Local History – Local History

Working in harmony with the Storage Manager, an internal version history management system allows you to track document changes without the use of an external version control system. Document history management includes the ability to view differences – both physical and logical – as well as revert to previously saved versions of the document.

Local document history management also works in harmony with an installed Version Control System. Individual designers can manage their own changes using the local history system, with the VCS providing a complete team-oriented document management system.

The approach of the local history management system is to take a copy of a file each time you perform a Save, keeping all the copies in a project History folder (the copy is the file before the save event). The project History folder is created in the same folder as the project file. If your project includes documents stored in sub-folders, then this sub-folder structure will be repeated within the History folder.

You can also use a central repository for the history of all projects. Configure this on the **Version Control – Local History** page (Figure 10) of the *Preferences* dialog (accessed by choosing **Preferences** from the main **DXP** menu). The number of days of file history is also configured here too. A history of file save events will continue to be maintained for this many days.

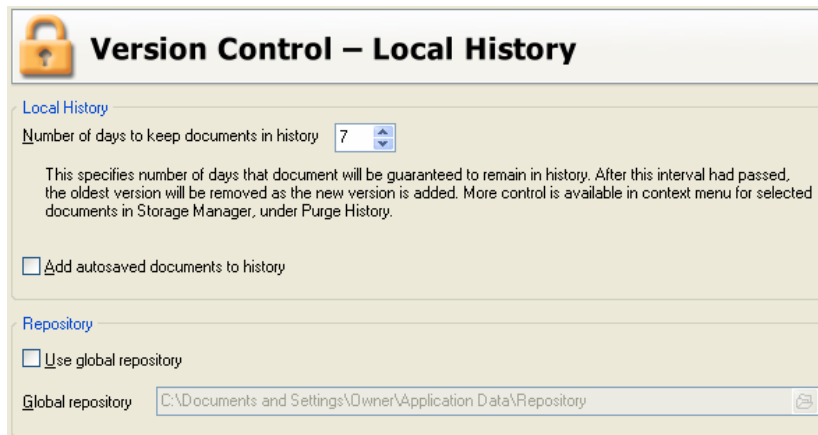


Figure 10. Options for the Local History feature

The lower region of the Storage Manager shows the local history for the selected file, with each history file being labeled **Version x**, where **x** increments with each save. Right-click on a saved version to **Apply a Label**; this allows you to ‘tag’ a particular version for later reference.

Right-click on a file in the history list to **Open** it, or to **Revert to** that version. **Ctrl**+click to select two files, then right-click to **Compare** them (Figure 11).

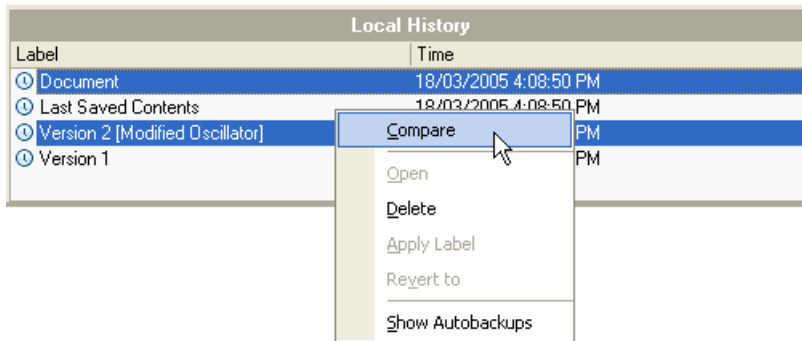


Figure 11. Comparing two versions of a document

Document Management – External Version Control

Version control is becoming the preferred method of electronic document management within many companies. Not only do version control systems provide safe and controlled storage and retrieval of a company's valuable documents, they also support easy retrieval of an older version of a document, and with suitable comparison tools, the ability to detect and examine document changes.

Use the options available in the Version Control section of the *Preferences* dialog to select either an SCCI compliant VCS, or to interface directly to either the CVS or SVN version control system.

If your project files are already stored in your version control system and checked out into a suitable working folder (or sandbox), you must also establish an initial connection between Altium Designer and the VCS-controlled project. To establish this connection, right click on the project file in the Storage Manager and select **Add Project To Version Control**. This will not create another copy of the project in the VCS, it will simply allow the appropriate VCS files and settings to be configured correctly, so that whenever you work on this project in the future, Altium Designer will recognize that it is under source control and correctly reflect the status of the documents. If the project was not already under version control, then use this process to add the project files (in the current folder) to the storage repository.

Once this step has been performed, you can then modify and **Check In** (or **Commit**) changes back to the repository directly from the Storage Manager. If you are working with CVS or SVN directly, the Revisions section of the Storage Manager will list the revision information for the active file (Figure 12).

VCS Revisions			
Revision	Time	Username	Comment
Document	31/01/2005 3:17:14 PM	jason.howie	
Last Saved Contents	31/01/2005 3:17:14 PM	jason.howie	
1.2	31/01/2005 3:17:34 PM	jason.howie	Interrupt rewired
1.1	31/01/2005 3:16:49 PM	jason.howie	*** empty log message ***

Figure 12. Accessing revision information in the Storage Manager panel



For further information on managing your design documents using an external VCS, refer to the [Working with a Version Control System](#) tutorial in the *System Design, Capture and Analysis* book.

Document Editor Resources

Each document editor has an associated set of default resources – menus, toolbars and shortcuts – the definitions of which are stored in corresponding resource files (*.rcs). These files can be found in the \System folder of the installation.

As you change back and forth through different document types in the main design window, the resources associated with that document's editor will automatically be loaded.

Behind each menu command, toolbar button and shortcut definition lies a pre-packed process launcher, which contains the process to run, along with any specific parameters, when the menu entry/toolbar button/shortcut key is invoked.

Any of the resources for an editor can be modified or added to as required, enabling you to fully customize your working environment to your own needs and preferences. Access to the relevant customization dialog for an editor can be made in one of the following ways:

- right-click on a menu bar or toolbar and choose **Customize** from the pop-up menu that appears
- double-click on a blank area of a menu bar or toolbar
- choose **Customize** from the main **DXP** menu.



For further information on customization of the design environment, refer to the [Customizing the Altium Designer Resources](#) tutorial in the *Inside the Altium Designer Environment* book. This tutorial includes rearranging the existing menus and toolbars, adding and deleting toolbar or menu commands, creating new drop-down menus and working with shortcut key tables.

Workspace Panels

Workspace panels are a staple ingredient of the Altium Designer environment. Whether specific to a particular document editor or used on a more global, system-wide level, they present information and controls that aid productivity and allow you to design in a more efficient capacity.

Accessing Panels

When the Software is first started, a number of panels will already be open. Some panels, including the **Files** and **Projects** panels, will appear grouped and docked to the left side of the application window. Others, including the **Libraries** panel will be in pop-out mode and appear as buttons on the right-hand border of the application window.

At the bottom of the application window there are a number of buttons that provide quick access to the available workspace panels, in context with the document Editor you are currently using. Each button is labeled with the name of the category of panels it gives access to. When a button is clicked, a pop-up menu of the constituent panels in that category will appear.

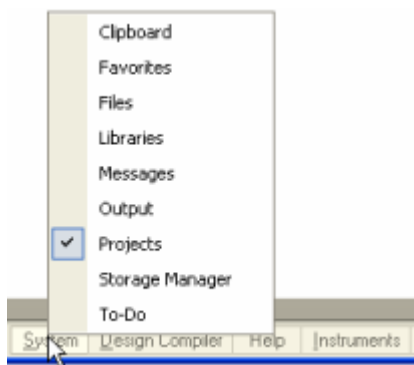


Figure 13. Accessing workspace panels

Click on an entry in a menu to open the corresponding panel. A tick symbol is used to indicate that the associated panel is open and visible in the workspace. If a panel is open but not visible, for example it is a non-active panel in a group of docked panels or it is currently in pop-out mode, clicking its entry in a menu will make it visible and it will become the active panel.

All currently available workspace panels can also be accessed from the **View » Workspace Panels** sub-menus.

Managing Panels

Depending on the particular document editor that is currently active, a large number of panels may be accessible or indeed open at any given time. To facilitate the layout and use of multiple panels in the workspace, various panel display modes and management features are provided.

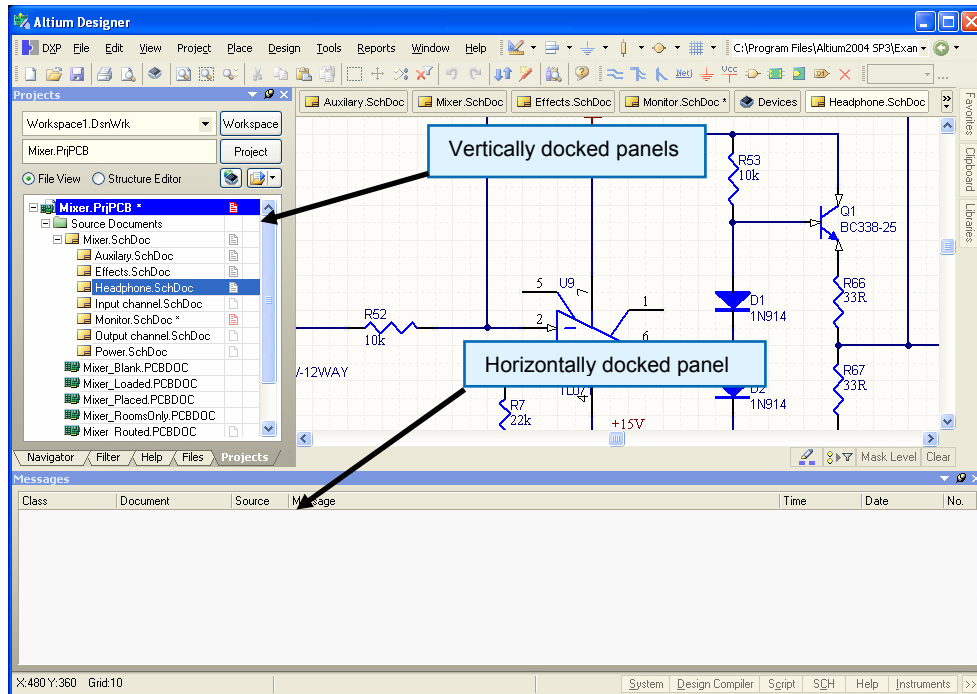
Panel Display Modes

Three different display modes are supported for panels:

- **Docked Mode** - In this mode a panel can be docked horizontally or vertically within the main application window. Right-click on a panel's title bar or tab and choose Allow Dock from the pop-up menu to configure the docking capability of the panel - horizontally and/or vertically

When docking vertically, a panel will dock to the right or left of the main design window. When docking horizontally, a panel will dock either above the main design window (and below docked toolbars) or below the main design window (and above the Status Bar)

Welcome to the Altium Designer Environment



- **Pop-out Mode** - This mode is essentially an extension of the standard docked mode. A docked panel can be put into this mode by clicking on the pin symbol next to the panel's close cross. The pin symbol will change to indicate the mode:

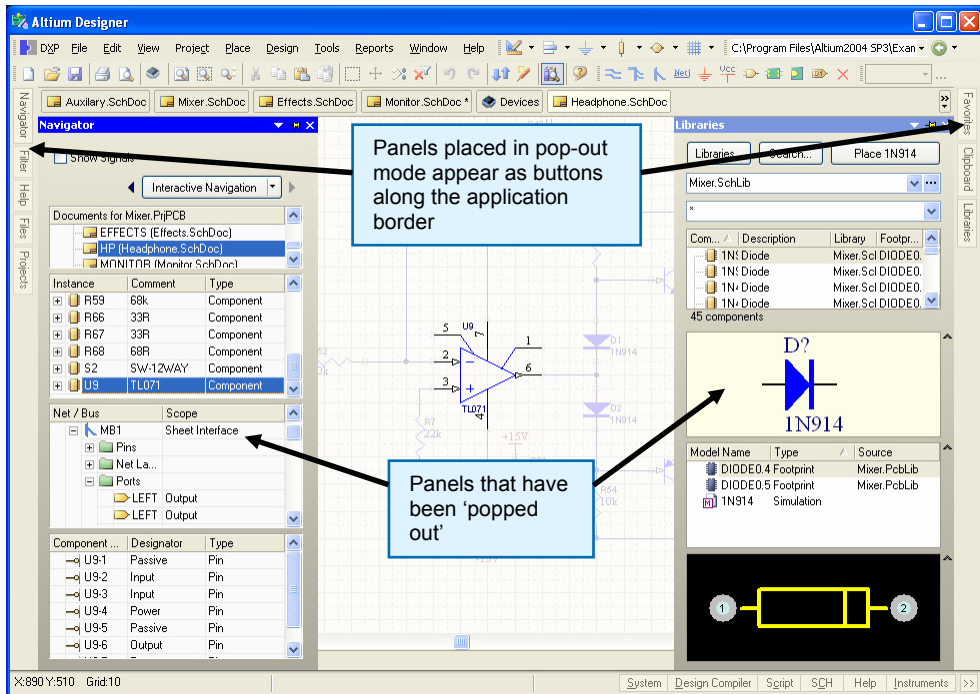


panel in original docked mode



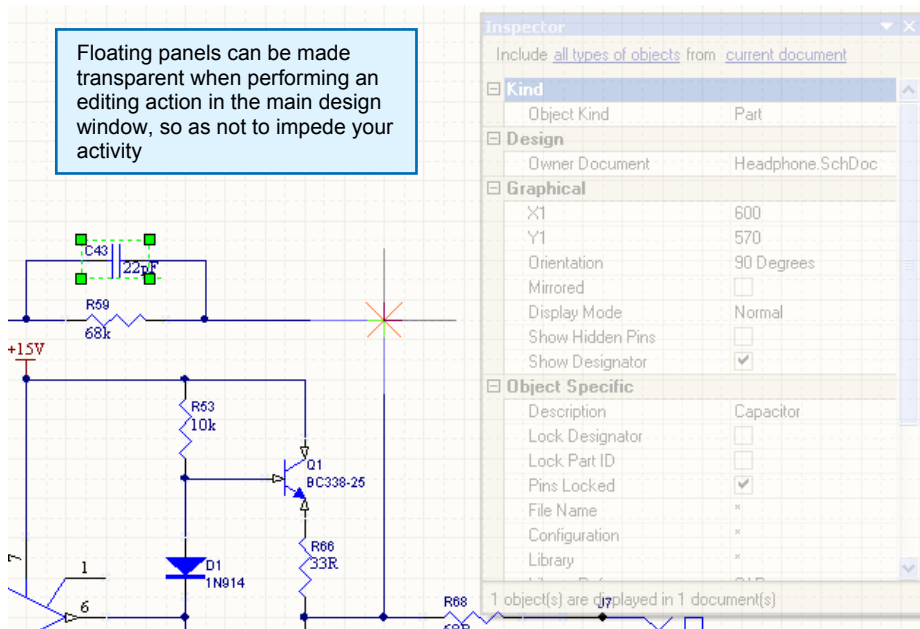
panel now in pop-out mode.

In this mode, the panel will appear as a button on the application border. Moving the cursor over the panel button will result in the panel sliding out from the border. Moving the cursor away from the panel will cause it to slide back again. Clicking on the panel button will expand the panel without sliding. In this case, click away from the panel to slide it back again



- **Floating Mode** - In this mode a panel can be placed anywhere within, or outside of, the Altium Designer environment. This is the standard opening mode for panels that have not been previously placed in docked or pop-out modes

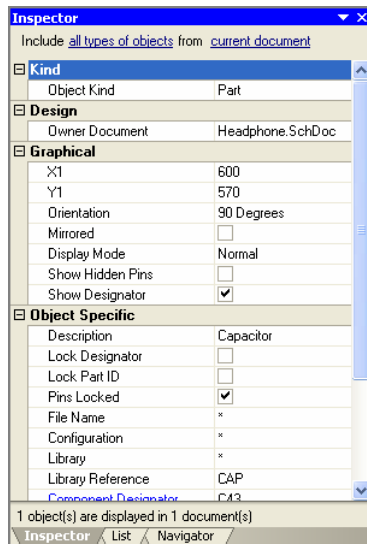
A floating panel positioned over the editing area of the main design window will be made transparent when carrying out an interactive operation in the main design window, in accordance with options defined on the System - Transparency page of the *Preferences* dialog (DXP » Preferences).



Grouping Panels

Panels may be grouped by simply dragging and dropping one panel on top of another. The resulting display of the group depends on where exactly you 'drop' the panel being added. Two modes of grouping panels are supported:

- **Standard tabbed grouping** - This mode displays a set of panels as a tabbed group, with only one panel in the group visible at any time.



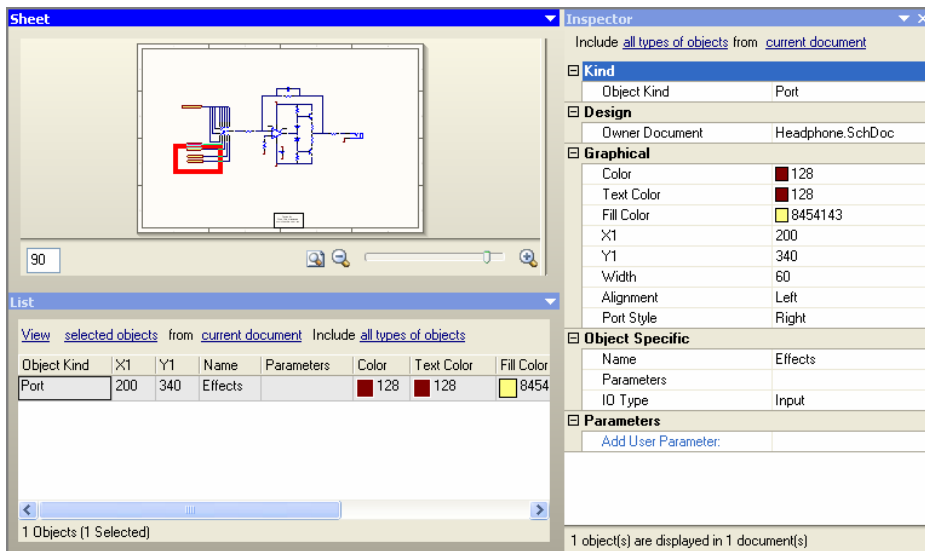
Use one of the following methods to group panels in this fashion:

- drag the panel you wish to add to the center of the target panel (or existing group) and drop. A yellow positional arrow will appear to the right of the tab(s) for the target, indicating that the panel will be added to the group as another tab
- drag the panel you wish to add to the region containing the tab(s) for the target panel (or existing group). A yellow positional arrow will appear, indicating that the panel will be added to the group as another tab. Simply move the mouse until the arrow indicates the specific position within the group where you wish to place the new panel and drop.

Use the small down-arrow at the top-right of the panel grouping to change the visible/active panel. Alternatively, make another panel in the group the visible/active panel by clicking directly on its tab.

The order of the panels in a tabbed group of panels can be changed at any time. Simply click on a panel's tab and drag to the left or right as required. A positional arrow will appear marking the position in the group order at which the panel will be placed if the mouse button is released

- **Fractal grouping** - This mode displays a set of panels as a fractal grouping, with multiple panels in the group visible at the same time.



This mode is analogous to horizontal/vertical tiling of open windows - you can simply drag a panel to dock it within another panel, effectively tiling them.

A fractal grouping can consist of individual panels and/or standard tabbed panels.

To group panels in this fashion, simply drag the panel you wish to add towards the top, left, right or bottom of the target panel (or existing group) and drop. The direction you choose determines where, in relation to the target, the new panel will appear. Ensure that no yellow positional arrow is displayed when adding the panel, otherwise it will be added to the target as an additional (and standard) tabbed panel.

Welcome to the Altium Designer Environment

To make a panel active in a fractal grouping, simply click on it. If the required panel is part of a tabbed grouping within the overall fractal structure, simply make it visible/active by clicking on its corresponding tab.

Moving Panels

A single floating or docked panel is moved by clicking within the panel's header and dragging it to a new position.

For a single panel that has been put in pop-out mode, the panel can be also moved by clicking on its corresponding button in the application border and dragging the panel to the intended position.

When panels have been grouped (either tabbed or fractal), clicking within a panel's header and dragging will move all panels in that group. To move a single panel elsewhere, thereby disassociating it from its group, click and drag the panel by its caption (or its tab if available).

To take a panel out of fractal grouping mode (where it has been tiled) and back into a standard tabbed grouping, simply drag the tiled panel, by its caption, to a target panel or existing tabbed group until a yellow positional arrow appears and then release - the panel will be added to the tabbed grouping.

Moving a panel to an application border that already contains one or more panels, will result in the panel being added in the same mode (docked or pop-out) as those already present.

As you move a panel close to another floating panel, the edges will snap together. Similarly, moving a panel towards the extremities of the desktop will snap the panel to the desktop's edge. This 'snap-attraction' feature allows for easier arrangement of floating panel sets within the environment.

When moving a panel, to prevent it from automatically docking, grouping or snapping, use the **Ctrl** key.

Closing Panels

A panel can be closed by right-clicking in its title bar (or on its tab, where available) and choosing the **Close** entry from the subsequent pop-up menu.

Use the 'close cross' at the far right of a panel's header to close an individual panel. However, if the panel is part of a group (fractal or standard tabbed), using this cross will close all panels in the group.

Maximizing/Restoring Panels

When in floating mode, a panel can be maximized by right-clicking in its title bar (or on its tab, where available) and choosing the **Maximize** entry from the subsequent pop-up menu. To restore a maximized panel back to its original size, simply right-click in its title bar or tab and choose the **Restore** command from the pop-up menu.

Alternatively, double-click in the title bar to toggle between maximized and restored state.



For further information on workspace panels, refer to the [Altium Designer Panels Reference](#) in the *Inside the Altium Designer Environment* book (online only). This reference provides complete and detailed information with respect to all panels available.

Environment Settings

Altium Designer streamlines the setting of environment options across all document editors and servers, by centralizing these options within a single, context-sensitive dialog – the *Preferences* dialog (Figure 14). Featuring a tree-like navigation structure, the dialog not only allows you to quickly and efficiently set system-wide preferences, but also load and save favorite preference settings, giving you still greater control over your working environment.

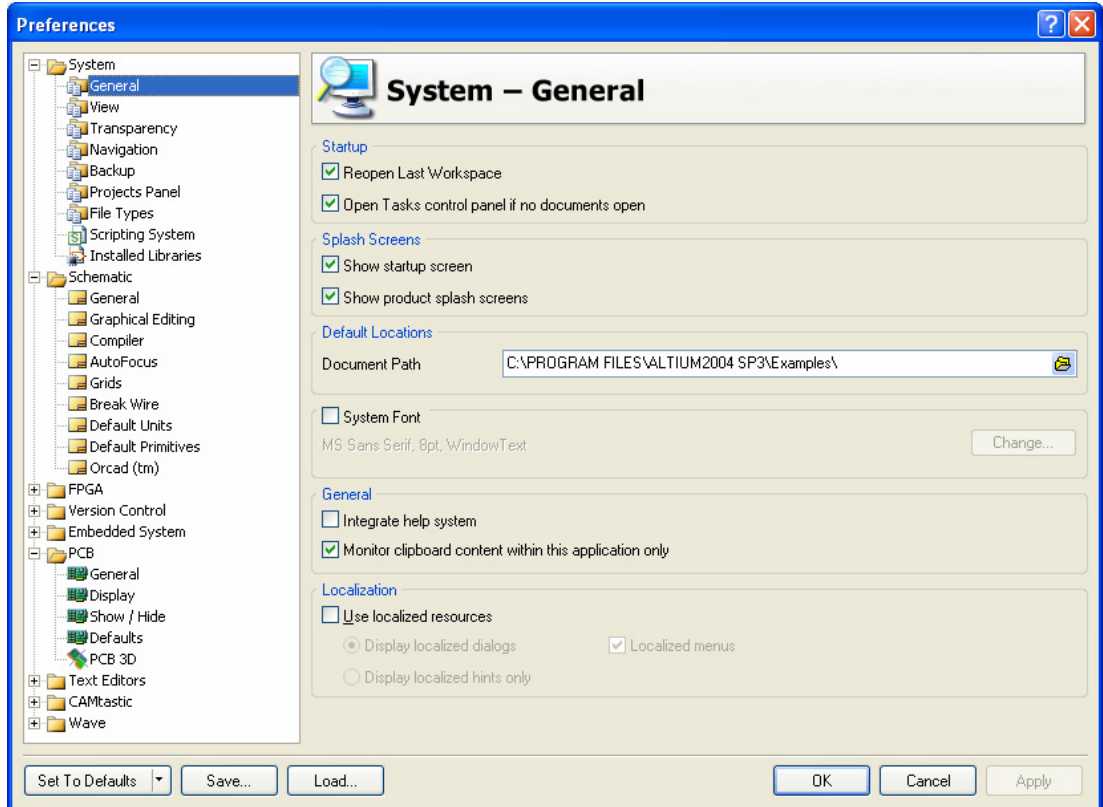


Figure 14. The Preferences dialog – command central for Altium Designer environment settings

Access to this dialog is made by choosing the **Preferences** command from the **DXP** main menu. From within an editor, you can also access the dialog using the relevant **Preferences** command (e.g. **Tools » Schematic Preferences** from the schematic main menus).

Navigation

To aid navigation of design documents, Altium Designer provides a dedicated **Navigation** toolbar (Figure 15), which can be accessed from within any of the application's document editors.

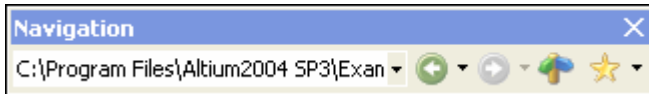


Figure 15. Dedicated document navigation tools

Direct Document Navigation

The field at the left of the bar enables you to navigate to any directory or document on any network or local storage medium, as well as any page on the internet. Simply type or paste the destination address into the field and press the **Enter** key. The destination directory or document will open as the active view in the main design window.

Destination addresses for directories and folders on local or network storage mediums take the form:

`file:///RootDirectory:/Path/Document`

Destination addresses for documents on the internet take the standard URL form:

`HTTP://WebAddress`

All Altium Designer integrated navigation support pages take the form:

`DXP://DocumentName`

The field in which you enter the destination address is actually a drop-down list which provides a history of all destination addresses (valid and invalid) which have been previously entered. This history list is not persistent across design sessions. It is cleared upon exiting the software.

Browsing Viewed Documents

The left and right arrow buttons allow you to effectively pass back and forth through documents that have previously been opened as active views in the main design window. The drop-down arrows located to the immediate right of these buttons provide a list of all documents that are part of the backward or forward browse sequence respectively. Click on an entry to view that document directly.

Integrated Navigation Pages


Click the  button to access the Home Page – part of the Altium Designer integrated navigation support pages.



Figure 16. Integrated navigation Home Page

This is the top-level page from where all predefined navigation support pages can be accessed. Each support page provides links to perform common tasks (e.g. creating a project or document file), as well as links to relevant documents contained within the Altium Designer Documentation Library (Figure 17).

Welcome to the Altium Designer Environment

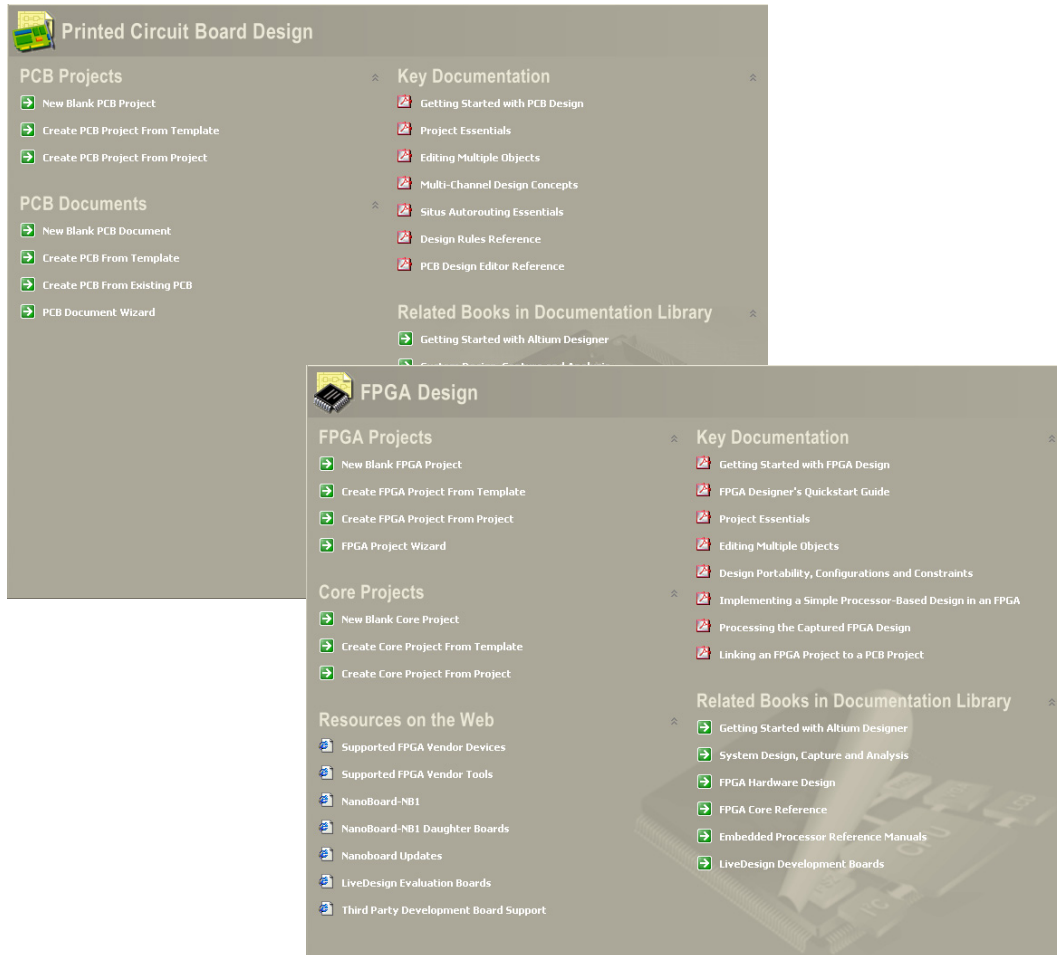



Figure 17. Support pages are available across the key areas of design

Favorites

This feature of the environment's navigational tools allows you to create, organize and maintain a Favorites list, which can include links to any directory or document on any network or local storage medium, as well as any page on the internet.

To add the active document in the main design window to the Favorites list, simply click on the  button of the **Navigation** toolbar and choose the **Add to Favorites** command from the associated drop-down menu. You can define a name for the link and also specify where in the Favorites folder structure the link is to be added.

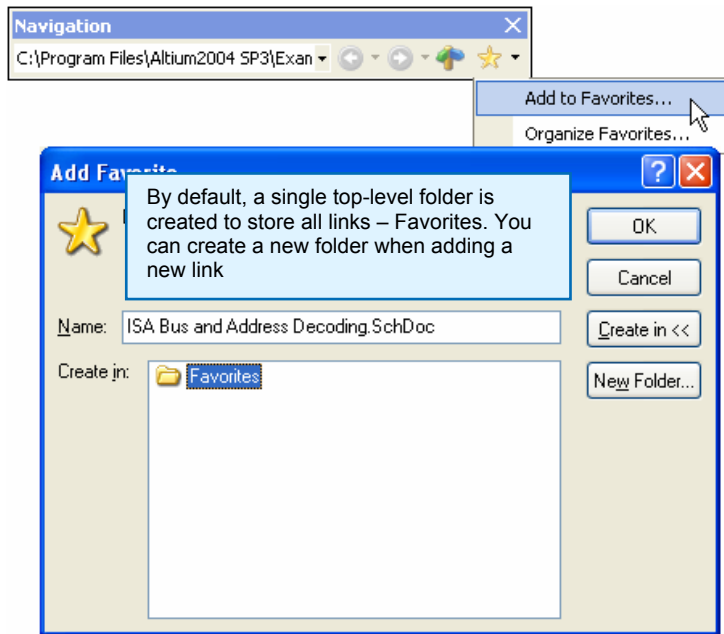


Figure 18. Adding a new document link to the Favorites list

Organization and modification of the Favorites list is carried out from the *Organize Favorites* dialog, accessed by choosing the **Organize Favorites** command from the same drop-down menu.

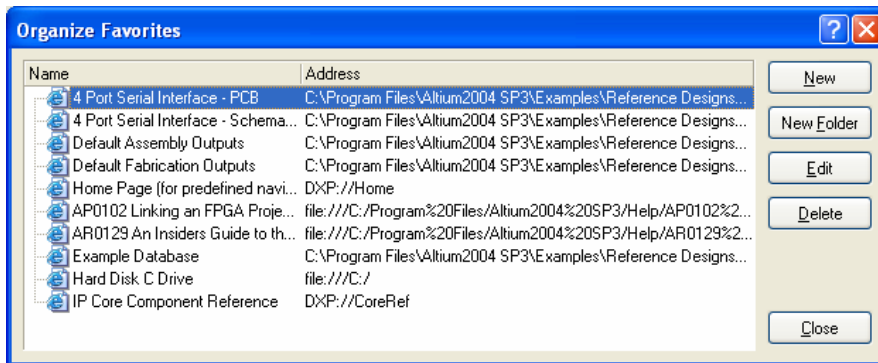


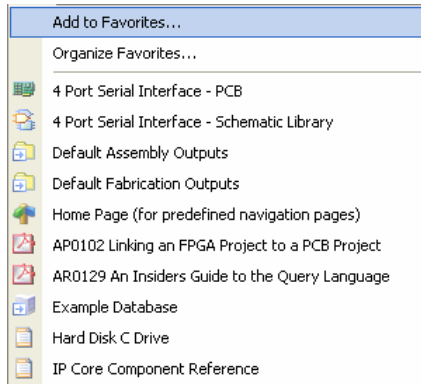
Figure 19. Organization of the Favorites list

The main window of this dialog lists each of the documents currently added to the Favorites list and their corresponding locations. Documents can be organized into specific folders. By default, a single top-level folder exists - Favorites. You can add additional folders in order to create a tree-like structure.

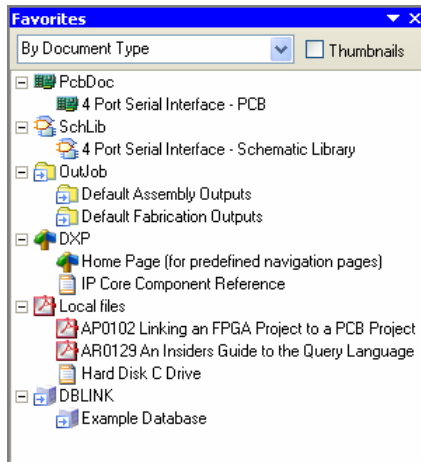
The list of documents added to the Favorites list can be accessed in two ways:

Welcome to the Altium Designer Environment

- from the drop-down menu associated with the  button, on the **Navigation** toolbar.



- from the **Favorites** panel. The panel can be configured to display the list in various ways.



For further information about the Favorites panel, press **F1** when the cursor is over the panel.

Localization for Foreign Languages

Altium Designer provides support for Chinese (Simplified), Japanese, German and French language installations. All menu items and most dialog text can optionally reflect the language chosen from those installed on the PC. The localized language is available as either translated hints over English text, or in translations of the forms and menus themselves.

Localization is configured on the **System – General** page of the *Preferences* dialog (**DXP » Preferences**).

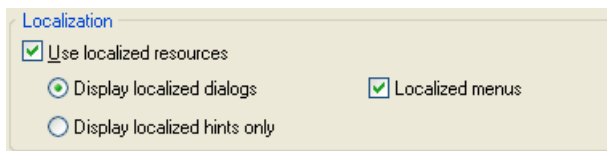


Figure 20. Setting localization options

Importing Design Files

Altium Designer incorporates a wide variety of importer and translator technologies, allowing you to effortlessly import designs originating from previous versions of the software or alternative software packages. Import of such files is performed using the *Choose Document to Open* dialog (Figure 21), accessed by choosing the **Open** command from the main **File** menu.

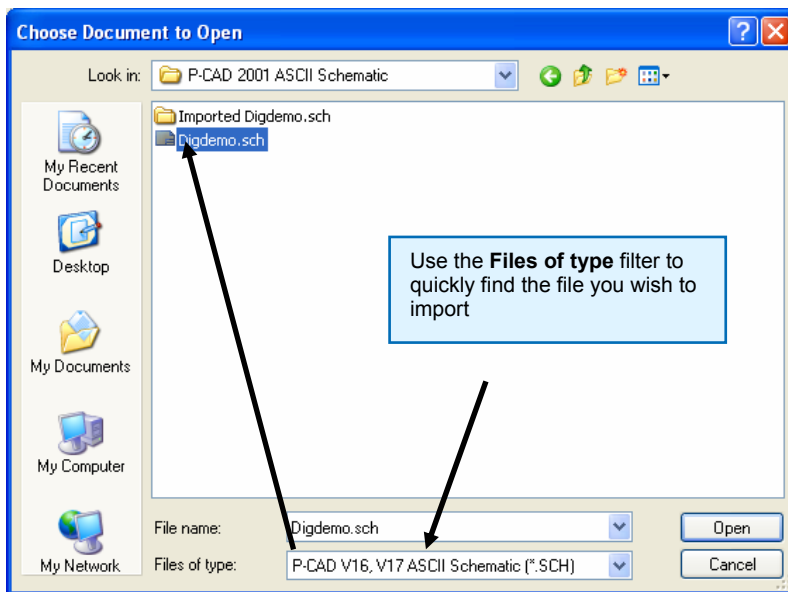


Figure 21. Importing documents using the Open command

The following non-Altium Designer documents can be imported:

- Protel 99SE Design Database (*.DDB)
- P-CAD V16 or V17 ASCII Schematic (*.sch)
- P-CAD V16 or V17 ASCII Schematic Library (*.lia)
- P-CAD V15, V16 or V17 ASCII PCB (*.pcb)
- P-CAD PDF file (*.pdf)
- CircuitMaker 2000 Design (*.ckt)
- CircuitMaker 2000 Binary USER Library (*.lib)
- OrCAD Layout file (*.max)
- OrCAD Max Library file (*.llb)

Welcome to the Altium Designer Environment

- OrCAD Capture Design (*.dsn)
- OrCAD Capture Library (*.olb)
- PADS PCB file (*.asc)
- SPECCTRA Design file (*.dsn)

Protel 99SE Import Wizard

To provide greater support and control over the import of Protel 99SE Design Databases (*.DDB), Altium Designer provides a dedicated import wizard (Figure 22) for this type of design file. Accessed from the main **File** menu, features include:

- the ability to run the wizard over multiple DDBs or folders. If the wizard is run over folders, the folders are recursively searched and all DDBs found will be imported
- file names changed automatically to ensure file associations are correct – for example schematic documents with no extensions will have `.SchDoc` appended to their file name. Sheet symbol linkage will remain functional, ensuring that the design compiles correctly
- an option to translate schematic documents to the latest format during the import process
- full control over how projects are created, with document-level options to define how each document should be handled.

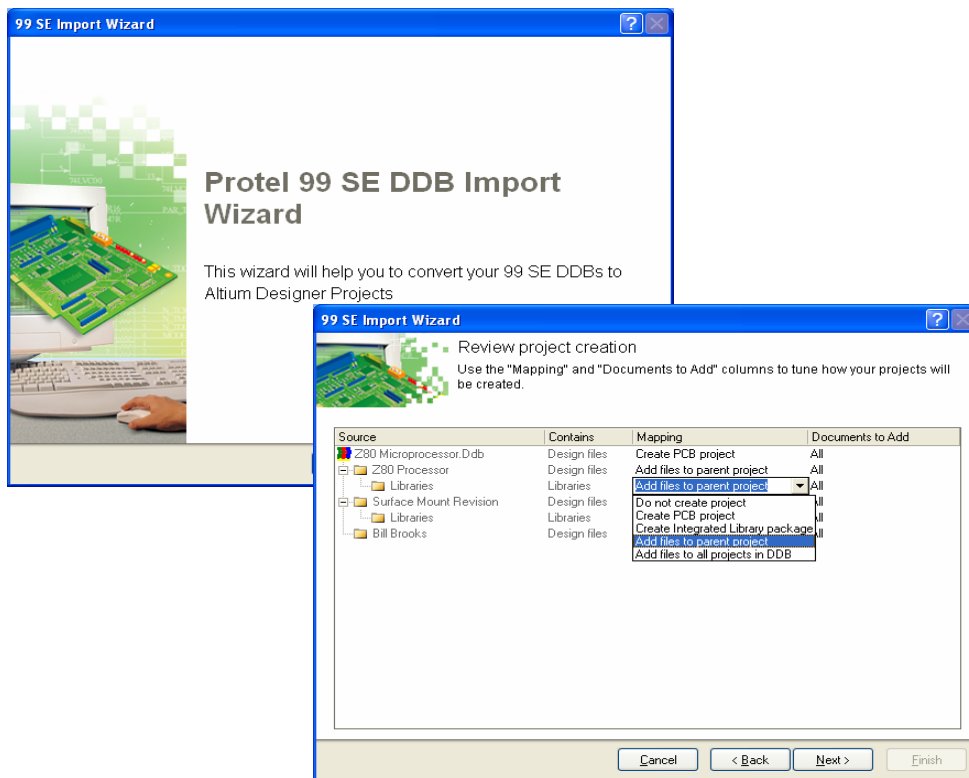


Figure 22. Importing a 99SE design using a dedicated import Wizard



For further information on importing a Protel 99SE design, see the [Transferring a Design from Protel 99SE](#) article.

Exporting Design Files

Design files (e.g. schematic, PCB) or projects themselves may be exported into various formats – including previous versions of the software or alternate software packages. Export of such files is performed using the **Save As** dialog (Figure 23), accessed by choosing the relevant **Save** command from the main **File** menu.

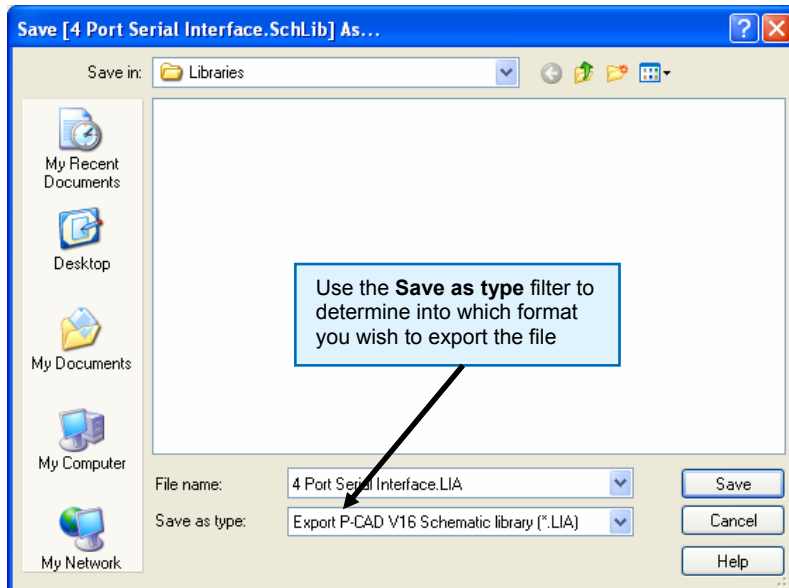


Figure 23. Exporting design files using the **Save As** command

Help at Hand

Altium Designer provides a range of documentation gathered together under a single 'umbrella' entity – the Altium Designer Documentation Library. The library not only contains information to get you started with designing in Altium Designer, but also detailed literature spanning the various, key areas of design. It also includes a dedicated Command Reference, where you can find help about each of the available, predefined commands, across the entire application.

Configuring and Accessing the Library

The Altium Designer Documentation Library can be launched in one of two modes:

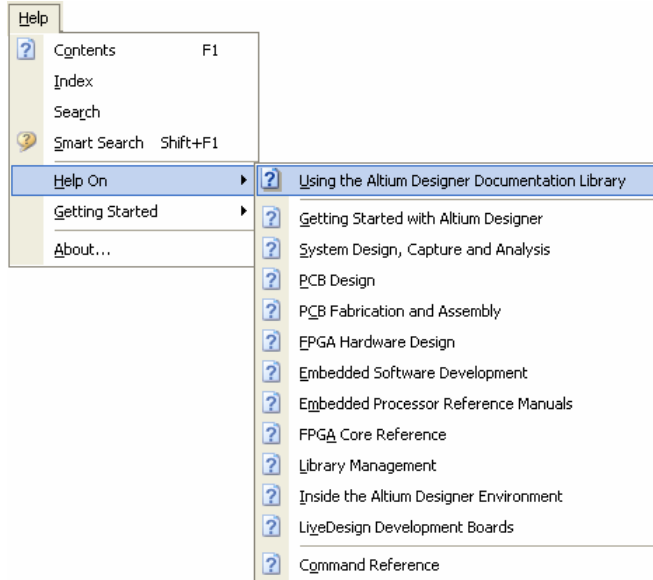
- **External Viewer Mode** - the documentation library appears in a separate tri-pane HTML Help Viewer and is very much suited when working with dual monitors
- **Integrated Mode** - the documentation library appears in a Task view within Altium Designer itself and navigation is controlled through the use of a dedicated workspace panel.

Welcome to the Altium Designer Environment

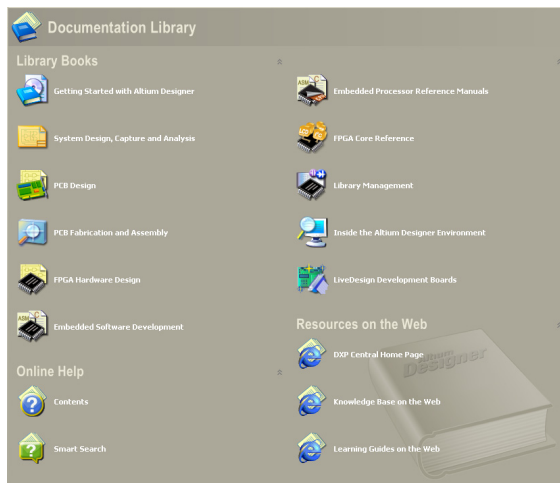
The mode is determined by the Integrate help system option, which can be found on the **System - General** page of the *Preferences* dialog (accessed by choosing the **Preferences** command from the **DXP** menu). By default, the documentation library will appear in the External Viewer Mode.

The Altium Designer Documentation Library can be accessed in a variety of ways, including:

- directly using commands available from the main **Help** menu, in any of the application's document editors










- pressing **F1** over any menu entry, toolbar button, workspace panel, or editor workspace
- opening any of the HTML Help files (*.chm) contained in the \Help folder of the installation
- Using the pages and links presented under the Documentation Library section of the Altium Designer integrated navigation pages



Once the Altium Designer Documentation Library has been launched, you can browse through the information it contains using its available navigation tools - the Contents, Index, Search and Smart Search tabs.

Library Structure

Documentation is essentially gathered together under top-level books. The type of documentation contained within each book varies and can appear in a number of different forms, ranging from high-level articles, to detailed references. Each document type is distinguished in the content tree by the use of a unique icon. The following is a summary of the document types available:

-  Articles - providing higher level, conceptual information, outlining how the technology provided by the application can be used to aid you in your designs
-  Tutorials - providing hands-on, "how to" information, helping you to learn the features of the software by example
-  Application Notes - providing information and tips about various specific features of the application
-  Guides - providing a mixture of detailed technical reference information and step-by-step, tutorial-like assistance
-  White Papers - providing detailed information relating to the introduction of a new technical feature or concept
-  References - providing detailed reference information with respect to a variety of features used in the software (e.g. design and scripting languages, workspace panels, PCB design rules, etc)
-  Core References - providing detailed reference information specifically related to FPGA-based core devices, including Processors, Peripherals and Instruments

Note: All documentation is supplied in PDF format, with the exception of References (supplied in both PDF and HTML format) and the Command Reference (supplied in HTML format only).

Natural Language Search Facility

The Smart Search feature provides a Natural Language Search Facility. Use the **Smart Search** tab available from the **Help** panel when in Integrated Mode or the tri-pane viewer when in External Viewer Mode to interrogate all areas of the documentation library simply by typing a question, structured as though you were asking a colleague. Double-clicking on a search result will take you directly to the topic in the associated area of the library.

Context-Sensitive Help – F1

Underlying context-sensitive help technology enables you to access information directly and in context with the action you are currently performing - providing you with help, right when and where you need it. The key to this technology - literally - is **F1**. Pressing **F1** in the following situations will access, where available, information relating to that specific area of design:

Welcome to the Altium Designer Environment

- press **F1** over a menu entry or toolbar button, to directly open the help topic in the Command Reference area of the system pertaining to that command
- press **F1** over a dialog to obtain detailed information about that dialog, in terms of its overall use
- press **F1** over a panel to obtain detailed help specific to that panel
- press **F1** while hovering the cursor over a design object in one of the following editors, to access help specific to that object:
 - Schematic Editor
 - Schematic Library Editor
 - PCB Editor
 - PCB Library Editor
- press **F1** inside an editor - away from design objects - to access information specific to that editor
- position the text cursor inside a key word within a script, embedded code or VHDL document and press **F1**, to access information specific to that keyword.

What's This? Help

Use the 'What's This Help' facility to gain detailed information about each of the individual options available in a dialog. Simply click on the question mark button at the top right of a dialog and then click over a field or option to pop-up information specific to that field or option.

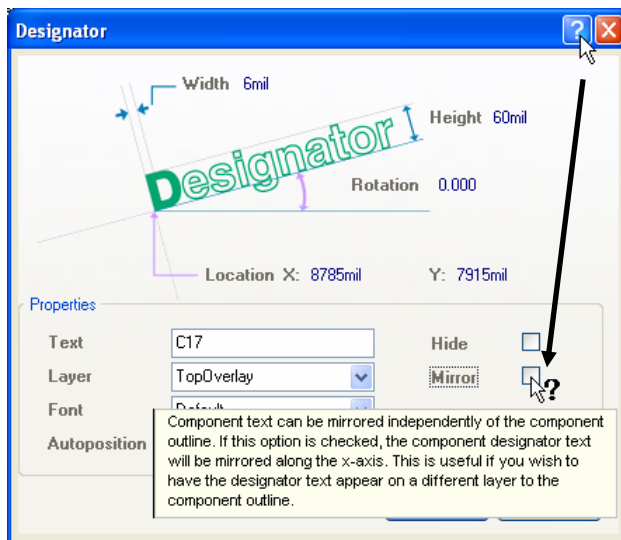


Figure 24. Using the What's This Help feature to access dialog option information



For further information, refer to the topic 'Understanding and Using the Documentation Library', accessible once the Altium Design Documentation Library is opened.

The Design Explorer (DXP) Integration Platform

The DXP platform provides the key to integration of the various design tools into the single Altium Designer environment. As part of this integration, its job is to provide consistent user interfaces and enhanced tool-interoperability.

Beyond this integration, it is also responsible for providing a wide range of features that are common across tools, saving multiple implementations of the same features and therefore speeding up the development process. As a result, DXP-based applications are inherently more feature rich than other applications.

Externally, the platform presents all the features that the user interacts with – menus, toolbars, panels and shortcut keys. Internally, it is the platform that each server plugs into. When a server is plugged in, it tells the DXP platform what functions (or processes) it includes and passes over a definition of all its menus, toolbars and shortcuts (resources). When a user selects a menu item, DXP passes a message to the appropriate server, telling that server what process to run.

What is a Server?

In everyday computer-user language, a server is a module that plugs into the DXP platform, to add new functionality to the environment. It can range from a complete document editor, such as the Schematic Editor, to a complex analysis engine, such as the Mixed-Signal Circuit Simulator, through to a simple utility that counts all the holes on a printed circuit board.

In software jargon, each Altium Designer server is a DLL (Dynamic Link Library). In Microsoft Windows a DLL is a library of functions and procedures, which can be used by any application and other DLLs. Microsoft developed the EXE/DLL model to allow software to be reusable. Software functions that are used by more than one application are stored in these libraries, which can then be called when the application needs that function. Windows is structured so that using a function from a library (DLL) is as quick and easy as using a function that is internal to the application.

Altium Designer – through its DXP platform – extends this model by making the functions and procedures inside each server DLL directly available to the user, through menus, toolbars and shortcuts.

As well as exposing the functionality of a server to the user through the menus, toolbars and shortcuts, each server exposes its functionality to other servers through an open Application Programming Interface (API). The API is the definition of how all the functions in a DLL are used. An API is called “open” when this definition is published, so that the functions in this DLL can be accessed by other EXEs and DLLs.

As well as allowing programmatic access to the same functions that the user can access through the resources, the API also includes more powerful functions that support direct manipulation of information in the design document that the editor currently has open. An example of this is the Schematic server – when it has a Schematic document open for editing, the Mixed-Signal Circuit Simulator can examine the contents of this document, directly through the API. Using this mechanism, it can extract information about the objects on the Schematic, required to perform the simulation and ultimately generate a waveform analysis.

Viewing Installed Servers

The list of currently installed servers (i.e. plugged into the DXP integration platform) can be viewed from the *EDA Servers* dialog (Figure 25). Access to this dialog is made by choosing the **System Info** command from the main **DXP** menu.

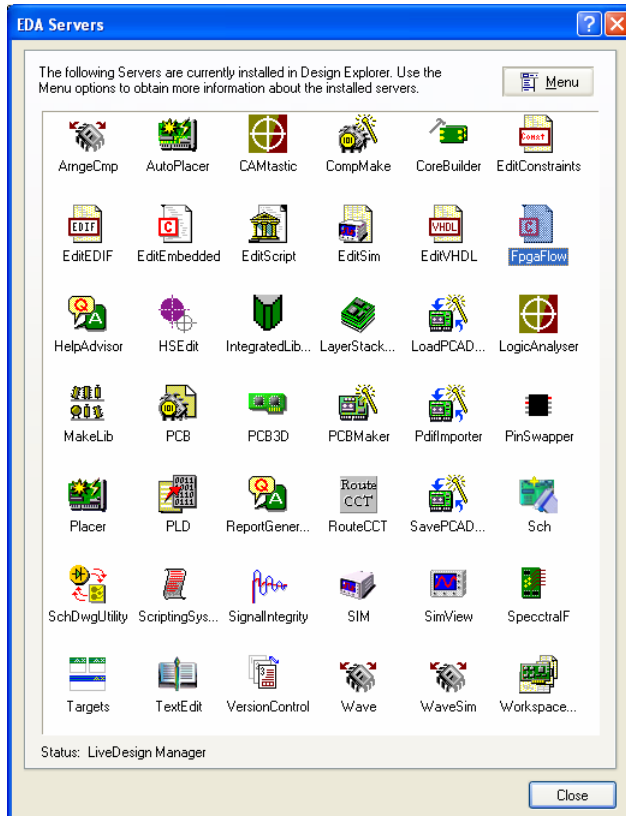


Figure 25. Displaying the list of installed servers

Servers can be grouped into three distinct categories:

- **Document Editor/Viewers** – these servers present a document editing (or viewing) window. Examples include the Schematic and PCB Editors
- **Wizards** – these servers pop up as a Wizard, where you step through a number of pages and answer questions. Examples include the Printed Circuit Board Wizard (PCBMaker) and the Component Maker Wizard (CompMake)
- **Utility Servers** – these servers work with one of the Document Editor servers. Typically they add items to the Document Editor's menus to allow access to their features. Examples are the Mixed-Mode Simulation Server (SIM), and the Hole Size Editor (HSEdit).

Use the commands available from the dialog's menu (button and right-click) to view information about a selected server in the list. Alternatively, double-click on an entry directly. The *Server* dialog will appear,

from where you can gain information on any document editors that the server provides and a list of processes that it provides (Figure 26).

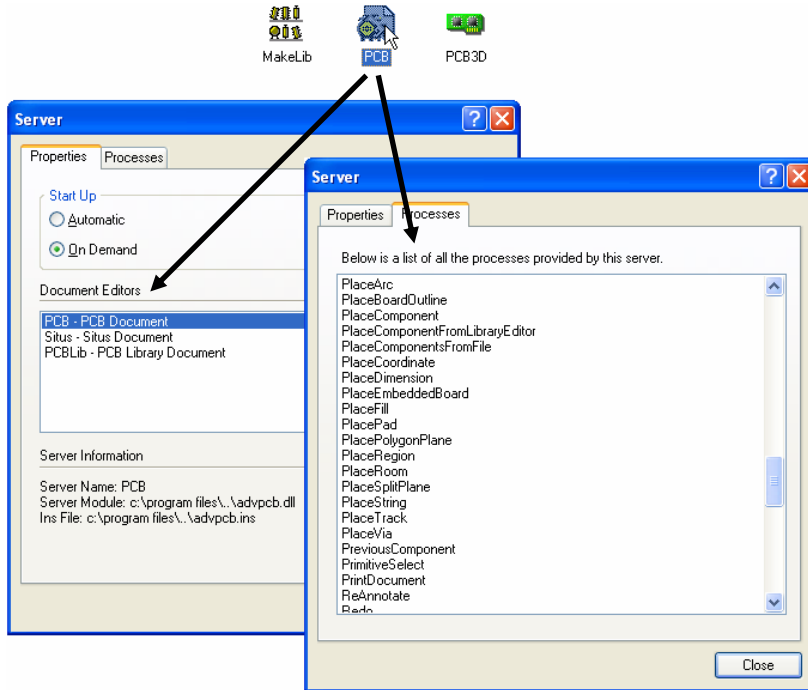


Figure 26. Interrogating server properties

Getting Started with Altium Designer

The following list of introductory-level documents provide further exploration of the environment and information that will aid you in getting started with designing using the Altium Designer software:

- Making Electronics Design Easier* - this article describes an entire design from the Engineer's perspective, dividing the job between PCB and FPGA design, then using Altium Designer's integrated tools to complete both
- Project Essentials* - this application note outlines the different kinds of projects, techniques for working on projects and how to use the Projects panel effectively
- Schematic Editing Essentials* - this application note looks at the placement and editing of schematic objects in Altium Designer
- Getting Started with PCB Design* - this introductory tutorial is designed to give you an overview of how to create a schematic, update the design information to a PCB and generate manufacturing output files. It also investigates the concept of projects and integrated libraries
- Getting Started with FPGA Design* - this tutorial gives an overview of how to create an FPGA design. It outlines how to create a schematic and then compile, synthesize, build and program a physical FPGA device on one of the Altium NanoBoard's satellite daughterboards. It also looks briefly at the use of sub-sheets and VHDL files in an FPGA design
- Editing Multiple Objects* - this tutorial describes various techniques for applying edits globally to multiple objects in your design. It covers using the Find Similar Objects dialog and Inspector panel combination, as well as the Parameter Manager and the Model Manager. Finally it introduces queries and the use of the Filter and List panels, which together provide a powerful technique for finding and editing design objects
- Transferring a Design from Protel 99 SE* - this article outlines the process you go through to transfer a Protel 99 SE design into the Altium Designer environment
- Creating a Multi-channel Design* - this tutorial shows how to create a multi-channel design in the Schematic Editor, including the use of sub-sheets, sheet symbols and the Repeat keyword. Setting room and designator formats and viewing the channel designator assignments are also covered

[Introduction to the Query Language](#) - this article introduces the query language and points readers to the resources within Altium Designer for understanding it. These resources include example queries in the History and Filter pop-up lists, and generation tools such as the Query Builder and Query Helper

[Shortcut Keys](#) this guide provides a list of shortcut keys available for use within Altium Designer's various Document Editors

Revision History

Date	Version No.	Revision
29-Apr-2005	1.0	Initial Release

Software, hardware, documentation and related materials:

Copyright © 2005 Altium Limited.

All rights reserved. You are permitted to print this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document is made. Unauthorized duplication, in whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium Limited. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties, including fines and/or imprisonment. Altium, Altium Designer, CAMtastic, Design Explorer, DXP, LiveDesign, NanoBoard, NanoTalk, CircuitStudio, Nexar, nVisage, P-CAD, Protel, Situs, TASKING, and Topological Autorouting and their respective logos are trademarks or registered trademarks of Altium Limited or its subsidiaries. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.