

Summary

Guide GU0112 (v2.0) June 19, 2006 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.

S Altium Designer DXP File Edit View Project Place Design Tools Reports Window Help 🛛 📈 🗸 🚔 🗸 🚽 🕇 🍦 🗸 C:\Program Files\Altium Designer 6\Ex. -😿 🤊 (* 🔰 🎾 🔯 🦃 🔁 🍾 📐 🔤 🚽 Vec De 📑 ▼ 🖉 × Printed Circuit Board D 📕 ISA Bus and Add ss Decoding.SchDoc System Menu Workspace Menus/Toolbars/Shortcuts Navigation Access to features **Document Tabs** Resources change according Provides controls for including environment Each open to the active document editor jumping to a preferences, document has its particular document, licensing and own tab. Click on a stepping back and server information document tab to forth through viewed make it the active documents and - Settings document. Rightaccessing the 🖃 🚞 Database Link Files click on a tab for integrated navigation Example Database.DBL ****** further Home Page E Dutput Job Files Main Design Window 💼 Default Assembly Outpu management 🔂 Default Fabrication Outp Display, arrange and controls + 🔲 Libraries edit open documents in this window Workspace Panels Various panels providing functionality specific to a particular editor or at a design/system level. Panel Access Panels can be docked, placed in a pop-out mode Workspace panels or left floating are accessible using these buttons < 🖉 🖇 🛛 Mask Level Clear Files Projects Navigator / SCH Filter X:420 Y:250 Grid:10 System Design Compiler SCH Help Instruments

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.

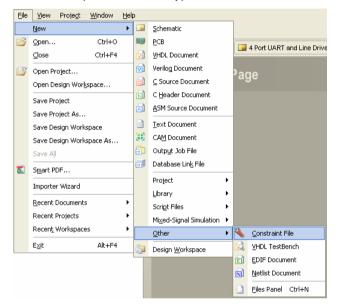
Working with Documents

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 methods:

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

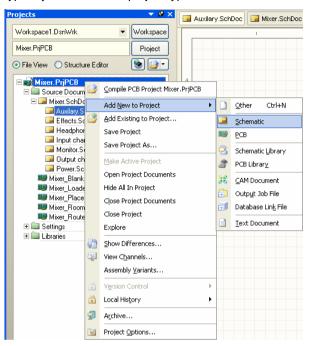


 click on the required entry in the New section of the Files panel. If this panel is not open, click on the System button at the bottom right of the main application window and choose Files from the pop-up menu that appears

New	۲				
PCB File					
🗔 Schematic Sheet					
🗑 VHDL File					
🗑 Verilog File					
👜 Blank Project (PCE	3)				
🔊 Blank Project (FPC	5A)				
📋 Blank Project (Cor	e)				
📄 Blank Project (Eml	bedded)				
🚔 Blank Project (Library Package)					
🛐 Blank Script Proje	t				
Dther Document					

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

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.

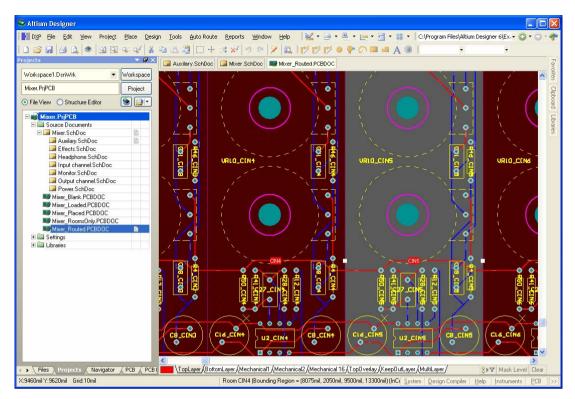


Figure 2. Multiple tabbed documents open in the main design window

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.

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

	📕 Auxilary.SchDoc	r.SchDoc) T	-
				🗔 Input channel.SchDoc	J
				📃 Headphone.SchDoc	I
Г	1	1	1	📕 Monitor.SchDoc	
				🗔 Output channel.Schooc	l
				📴 Power.SchDoc	l
Λ.				Mixer_Routed.PCBDOC	
					"

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).

Projects			• X
Workspace1.DsnWrk	•	Work:	space
Mixer.PrjPCB		Pro	ject
⊙ File View ○ Structure Ed	litor	۱	
🗆 🗐 Mixer.PrjPCB *		B	~
Source Documents Mixer.SchDoc Auxilary.SchDo Effects.SchDo			
Use of icons to distinguish document states enables you to quickly assess which documents are modified	Cc hDoc * CDC DC TOC TOC		
Mixer_RoomsOnly		:	
Wixer_Routed.PC	BDOC	ß	~

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).

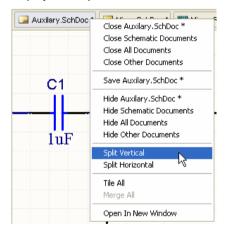


Figure 5. Accessing document management commands

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.

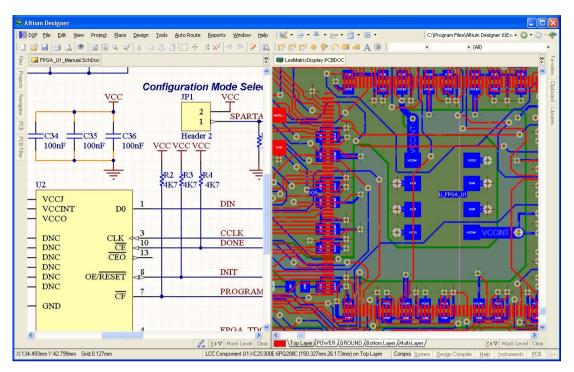


Figure 6. Splitting the main design window

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.

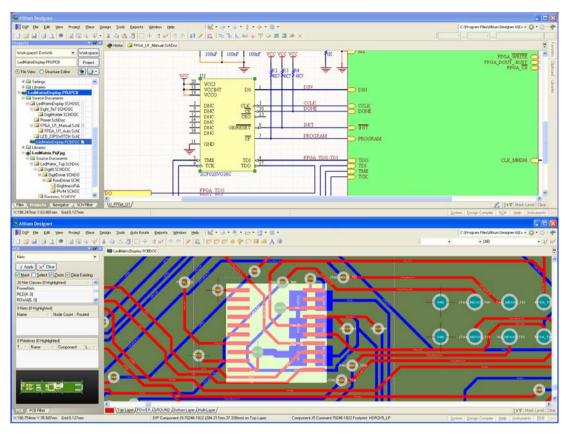


Figure 7. Horizontally arranging Altium Designer windows on the Desktop

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.

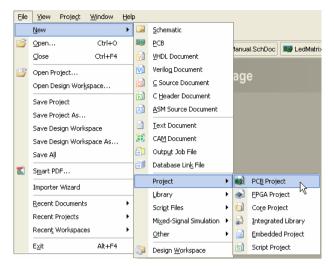
Projects

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

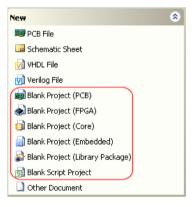
- PCB project (*.PrjPcb)
- FPGA project (*.PrjFpg)
- Core project (*.PrjCor)
- Embedded project (*.PrjEmb)
- Library package (*.LibPkg)
- Script project (*.PrjScr)

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

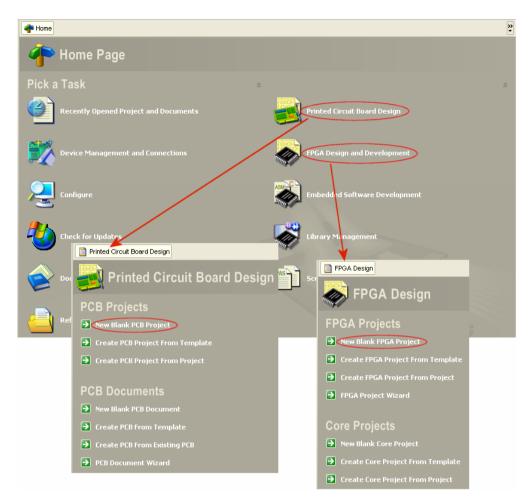
• select the required project type from the File » New » Project sub-menu



 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) 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.

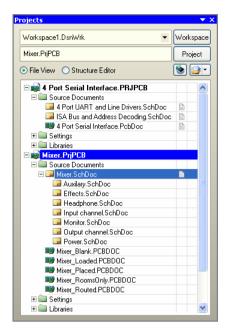


Figure 8. The Projects panel

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.

III 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).

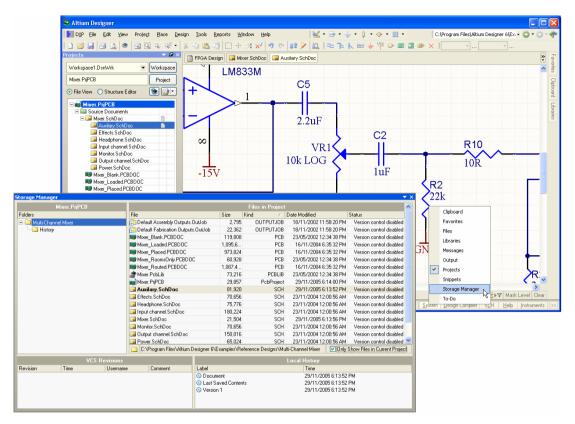


Figure 9. The Storage Manager panel

The Storage Manager panel can be used for:

- general everyday file management functions for files in the project or within the active project's folder structure
- management of symbol and footprint libraries stored in an SVN repository and associated with an SVN Database Library. The panel provides access to the local working directory into which a copy of the libraries stored in the repository are checked out. Committal of changes to the repository and reception of updates from the repository can only be performed through the panel.
- 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.

IF 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, 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

Working in harmony with the **Storage Manager** panel, 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.

🔒 Versio	n Control – Local History
Local History	
<u>N</u> umber of days to keep	documents in history 🛛 7 😌
the oldest version will	r of days that document will be guaranteed to remain in history. After this interval had passed, be removed as the new version is added. More control is available in context menu for selected a Manager, under Purge History.
Add autosaved docu	ments to history
Repository	
Use global repository	
Global repository	Documents and Settings\Dwner\Application Data\Repository

Figure 10. Options for the Local History feature

The lower region of the **Storage Manager** panel 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).

	Local History	
Label	Time	
O Document	18/03/2005 4:08:50 PM	
🕔 Last Saved Contents	18/03/2005 A-08-50 PM	
O Version 2 [Modified Oscillator]	Compare PM	
Version 1	Open VS PM	
	Delete	
	Apply Label	
	Revert to	
	Show Autobackups	

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 on the Version Control – General page 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** panel 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** panel. If you are working with CVS or SVN directly, the **VCS Revisions** section of the **Storage Manager** panel 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						
P Last Saved Contents	31/01/2005 3:17:14 PM	jason.howie						
P1.2	31/01/2005 3:17:34 PM	jason.howie	Interrupt rewired					
∰ 1.1	31/01/2005 3:16:49 PM	jason.howie	xxx empty log message xxx					

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.

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. 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

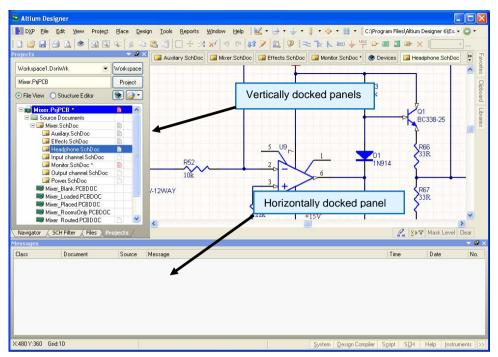
Depending on the particular document editor that is currently active, a large number of panels may be accessible or indeed open at any given time. To facilitate the layout and use of multiple panels in the workspace, various panel display modes and management features are provided.

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

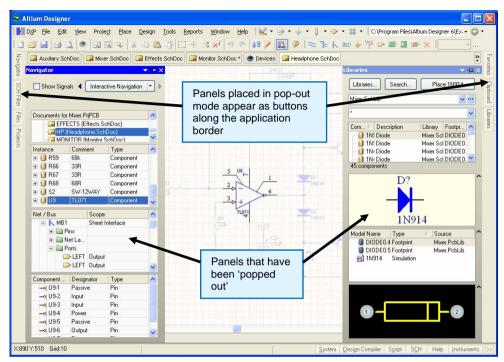
When docking vertically, a panel will dock to the right or left of the main design window. When docking horizontally, a panel will dock either above the main design window (and below docked toolbars) or below the main design window (and above the Status Bar)



- **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

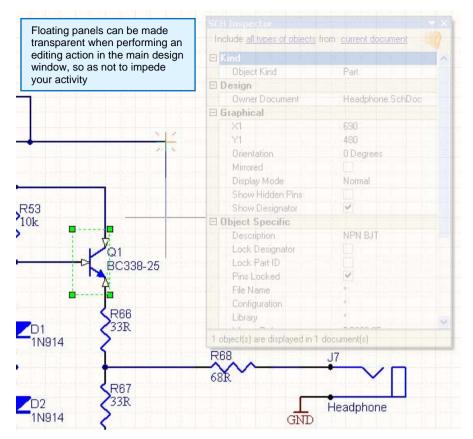
→ panel now in pop-out mode.

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.

Ξ	Kind		
	Object Kind	Part	
Ξ[Design		
	Owner Document	4 Port UART and Line Drivers.SchDoc	
30	Graphical		
	X1	200	
	Y1	630	
	Orientation	0 Degrees	
	Mirrored		
	Display Mode	Normal	
	Show Hidden Pins		
	Show Designator		
3 (Object Specific		
	Description		
	Lock Designator		
	Lock Part ID		
	Pins Locked		
	File Name		
	Configuration	*	

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. Shading (blue) and icons are used to also indicate where in relation to the target, the panel you are moving will be added. Dragging the panel to the center will cause the entire target panel/group to become shaded.

SCH Inspector	▼ X	SCH Li	t				. ×	SC	H Inspector		• ×
Include all types of objects	; from <u>current document</u>	View	elected obj	ects fror	n <u>curre</u> 🛒 r	<u>iment</u> Include <u>a</u>	all types of objects	In	clude <u>all types of object</u>	s from current document	
⊟ Kind	~	Object	ind X1	Y1	Orientation	Description	Mirrored Loc		Kind		~
Object Kind	Part	Part	200	630	0 Degrees				Object Kind	Part	
🗆 Design									Design		
Owner Document	4 Port UART and Line Drivers.SchDoc								Owner Document	4 Port UART and Line Drivers.SchDoc	
🖃 Graphical									Graphical		
X1	200								X1	200	
Y1	630								Y1	630	
Orientation	0 Degrees								Orientation	0 Degrees	
Mirrored			1						Mirrored		
Display Mode	Normal				_				 Display Mode 	Normal	
Show Hidden Pins			J						Show Hidden Pins		
Show Designator	 Image: A state of the state of								Show Designator	v	_
Object Specific									Object Specific		
Description									Description		
Lock Designator									Lock Designator		
Lock Part ID									Lock Part ID		
Pins Locked	✓								Pins Locked	✓	
File Name									File Name		
Configuration	н								Configuration	×	24
Library	н 🗸	<					>		1.9 1.5 ar 5 - 1.5 1.5		
	TI 400754		ts (1 Selecti						bject(s) are displayed in		
1 object(s) are displayed in	1 document(s)	SCH	ist / SCH	Filter				13	SCH List 🔬 SCH Filter !	SCH Inspector	

Drag the panel you wish to add to the region containing the tab(s) for the target panel (or existing group). The entire target panel/group will be shaded and 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.

5CH List									-	SCH Inspector		•
<u>View</u> <u>selecte</u>	<u>ed objects</u> fr	om <u>cur</u>	rent docum	i <u>ent</u> Include <u>al</u>	types of obje	<u>ects</u>			2	Include <u>all types of objec</u>	ts from current document	R
Object Kind	X1 Y1	Ori	entation	Description	Mirrored	Lock Designator	Lock Part ID	Pins Locked	F	🗆 Kind		
Part	280 61	0 0)egrees					~		Object Kind	Part	
									-	🗆 Design		
										Owner Document	4 Port UART and Line Drivers.Sch)oc
										🗆 Graphical		
										×1	280	
										Y1	610	
٢										Orientation	0 Degrees	
·).)					>	Mirrored		
1 Objects (1 S	electedj								_	Display Mode	Normal	
5CH Filter									-	Show Hidden Pins		
Limit search t	:0		Find items	matching these	criteria:		Objects passir	ng the filter 🔊		Show Designator	 	
 All Object 							Select			Object Specific		
							Zoom			Description		
 Selected 	Objects						_			Lock Designator		
O Non Sele	cted Objects						Objects not pa	assing the filter		Lock Part ID		
							Deselect			Pins Locked	✓	
Consider obje							Mask out			File Name		
Current Doc	ument	~					Mask out			Configuration	×	
			💦 Help	or 📥 Envori	tes 🙆 Hi	atoru				Library	×	
			A Helh			story		Apply		1 object(s) are displayed in	1 desument(s)	_

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. As you move the panel over the target, shading and icons will be evident – use these to help locate the panel at the required position. 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.

SCH Filter		▼ ×		
Limit search to	Find items matching these criteria:	Objects passing the filter 🔊		
 All Objects 		Select		
Selected Objects		Zoom		
		Objects not passing the filter		
Non Selected Objects				
Consider objects in		Deselect		
Current Document		Mask out		
	矝 Helper 👷 Favorites 🙆 History	Apply		
SCH List			SCH Inspector	▼ X
	<u>sjects</u> from <u>current document</u> Include <u>all ty reference</u>			cts from current document
Object Kind X		Lock Designator Lock Part ID Pins Locked	F 🗆 Kind	
Part 25			Object Kind	Part
	io oto obegiees			1 dit
			Owner Document	4 Port UART and Line Drivers.SchDoc
			🗆 Graphical	
			×1	280
			Y1	610
			Orientation	0 Degrees
			Mirrored	
			Display Mode	Normal
			Show Hidden Pins	
			Show Designator	v
			Object Specific	
			Description	
			Lock Designator	
			Lock Part ID	
			Pins Locked	v
			File Name	
	×.		Configuration	×
<			Library	
SCH List			SCH Inspector /	

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 (target becomes fully shaded) 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*. 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.

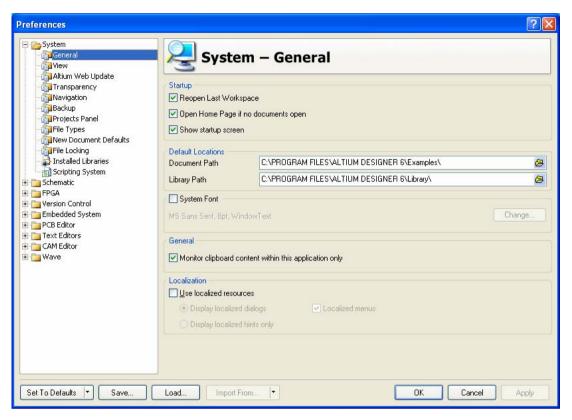


Figure 14. The Preferences dialog - command central for Altium Designer environment settings

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 **P** button to access the **Home** page – part of the Altium Designer integrated navigation support pages.

🕂 Home Page	
Pick a Task	
Recently Opened Project and Documents	Printed Circuit Board Design
Device Management and Connections	FPGA Design and Development
Configure	Embedded Software Development
Check for Updates	Library Management
Documentation Library	Script Development
Reference Designs and Examples	
or Open a Project or Document	*
Most Recent Project - LedMatrixDisplay.PRJPCB	
Most Recent Document - 4 Port UART and Line Drivers.5chDoc	
Open any Project or Document	
Project Name Project Type IndMatrix.PrjFpg FPGA Project IndedMatrix.Display.PRJPCB PCB Project Mixer.PrjPCB PCB Project	Location Last Saved C1Program Files\Altium Designer 6\Examples\Reference 28/07/2004 3:14:30 PM C1Program Files\Altium Designer 6\Examples\Reference 16/06/2004 12:46:40 PM C1\Program Files\Altium Designer 6\Examples\Reference 29/11/2005 6:14:00 PM

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).



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

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 18. 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 can be performed using the *Choose Document to Open* dialog (Figure 19), accessed by choosing the **Open** command from the main **File** menu.

Choose Docume	ent to Open	? 🗙
Look in:	🗁 P-CAD 2001 ASCII Schematic 🛛 🗸 🧭 📴	
My Recent Documents	Imported Digdemo.sch	
Desktop	Use the Files of type filter to quickly find the file you wish to import	
My Documents		_
My Computer		
	File name: Digdemo.sch	Open
My Network	Files of type: P-CAD V16, V17 ASCII Schematic (*.SCH)	Cancel

Figure 19. 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 PDIF file (*.pdf)
- CircuitMaker 2000 Design (*.ckt)
- CircuitMaker 2000 Binary USER Library (*.lib)
- OrCAD Layout file (*.max)
- OrCAD Max Library file (*.11b)
- OrCAD Capture Design (*.dsn)
- OrCAD Capture Library (*.olb)
- OrCAD CIS Configuration file (*.dbc)
- PADS PCB file (*.asc)
- SPECCTRA Design file (*.dsn)

Importer Wizard

To make the import process as smooth and efficient as possible, Altium Designer provides a unified means of importing designs from a variety of different design tools – the Importer Wizard. The Wizard walks you through the import process, handling both the Schematic and PCB parts of the project, as well as managing the relationship between them.

The Wizard can be used to import the following types of design projects:

- Protel 99 SE design databases
- CircuitMaker 2000 schematics and libraries
- P-CAD designs and libraries
- OrCAD designs and libraries
- OrCAD CIS Configuration files and libraries
- PADS ASCII PCB designs and libraries
- OrCAD and PADS designs and libraries.

Import Wizard			
		Iport Wizard mport Wizard Select Type of Files to Import Select the type of files you wish to import from the li	
		File Types	Description
		Pile Types 99SE DDB Files	99SE DDB (*,DDB)
	Cir Or Dr P4 Or	Conculta Alex 2000 Schematics and Libraries Files Orcad Designs and Libraries Files Orcad CIS Configuration Files and Libraries PADS ASCII PEO Besign And Library Files Orcad and PADS Designs and Libraries Files P-CAD Designs and Libraries Files	SSEE DOL (DDB) CircuitMaker Schematics (*:CKT), CircuitMaker User Libraries (*:LIB), CircuitMaker Device Libraries (*:LIB) Orcad Designs (*:DSN), Orcad PCB (*:MAX), Orcad Design Libraries (*:OLB), Orcad PCB Libraries (*:LLB) Orcad IOS Config File (*:DBC), Orcad Library Files (*:OLB), *:LLB) PADS ASCI IPC (*:ASC), PADS ASCII IPCB Library (*:D) Orcad Designs (*:DSN), Orcad Design Libraries (*:OLB), PADS ASCII PCB (*:ASC), PADS ASCII PCB Library (*:D) P-CAD ASCII Schematics (*:SCH), P-CAD ASCII PCB (*:PCB), P-CAD ASCII Libraries (*:LIA)
			Cancel (Back Next> Errish

Figure 20. Importing from a variety of design tools using the Import Wizard.

- For further information on importing a Protel 99SE design, refer to the *Moving to Altium Designer from Protel 99 SE* article.
- For further information on importing a design from P-CAD, refer to the *Moving to Altium Designer from P-CAD* application note.
- For further information on importing a design from OrCAD, refer to the *Moving to Altium Designer from OrCAD* application note.

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 21), accessed by choosing the relevant **Save** command from the main **File** menu.

Save [4 Port Se	rial Interface.S	chLib] As	? 🔀
Save in:	🗀 Libraries	🕑 🧿 🖻 📰	
My Recent Documents Desktop My Documents		Use the Save as type filter to determine into which format you wish to export the file	
My Computer	File name:	4 Port Serial Interface.LIA	Save
	Save as type:	Export P-CAD V16 Schematic library (*.LIA)	Cancel
My Network			Help

Figure 21. Exporting design files using the Save As command

Documentation and Help

The Altium Designer documentation is accessed through the **Knowledge Center** panel.

The **Knowledge Center** panel presents help information while you work. It tracks the command, dialog, object, or panel that is currently under the cursor and loads help about it – hover for a second or so for the content to appear. If you would prefer to 'freeze' the content of the panel at the currently loaded topic, simply click the Autoupdate button to disable auto-loading. You can still use **F1** to load content with autoupdate disabled.

The **Knowledge Center** panel is a portal, from the concise help summary displayed in the top of the panel there are links to PDF based reference and applied documents.

The lower section in the panel has a navigation tree, use this to browse through the PDF-based documentation, and open a document of interest.

The **Knowledge Center** panel includes a powerful PDF searching feature, available at the bottom of the panel. Pages that include all words in the search string are returned (except common words such as and, or, etc). The search scope is

 T] or click the Interactive Routing button Start interactive routing as before, but use the * key to . 		
This command is used to place track objects onto the current document. A track is a straight solid-filled line with a defined width. Interactive routing is the process of placing track segments. Tracks are generally placed on a signal layer to form the electrical interconnection between component pads on a PCB. However, tracks are also used as general-purpose line drawing elements to create board outlines, component outlines, polygon planes, keepout boundaries, etc. Use After launching the command, the cursor will change to a crosshair and you will enter interactive routing mode. Simply click or press Enter to anchor the starting point for the track you wish to route. If you place a track on a design object that has a net name, the track you are placing will adopt the net name, becoming part of that net. If you click on a connection line, the cursor will jump to the nearest pad and then keep the connection line attached to the end of the track you are placing. Documentation Library > Board Layout > Search results # AP0107 Impedance-Controlled Routing # AP0135 Interactive and Differential Pair Routing # AP0136 Internal Power and Split Planes # AR0126 Situs Autorouting Essentials # TR0104 Altium Designer Panels Reference # AR0128 Situs Autorouting Essentials # TR0114 Design Rules Reference # TR0114 Design Rules Reference # TR0115 Design Rules Reference # TR0116 Design Rules Reference # TR0116 Design Rules Reference # TR0117 Getting Started with PCB Design > Select Place > Interactive Routing from the menus [shortor > T] or click the Interactive Routing button > T] or click the Interactive Routing button > T] or click the Interactive Routing button > T] or click the Interactive Routing button	Ir	nteractively Route Connections
A track is a straight solid-filled line with a defined width. Interactive routing is the process of placing track segments. Tracks are generally placed on a signal layer to form the electrical interconnection between component pads on a PCB. However, tracks are also used as general-purpose line drawing elements to create board outlines, component outlines, polygon planes, keepout boundaries, etc. Use After launching the command, the cursor will change to a crosshair and you will enter interactive routing mode. Simply click or press Enter to anchor the starting point for the track you wish to route. If you place a track on a design object that has a net name, the track you are placing will adopt the net name, becoming part of that net. If you click on a connection line, the cursor will jump to the nearest pad and then keep the connection line attached to the end of the track you are placing. Documentation Library > Board Layout > Search results		
is the process of placing track segments. Tracks are generally placed on a signal layer to form the electrical interconnection between component pads on a PCB. However, tracks are also used as general-purpose line drawing elements to create board outlines, component outlines, polygon planes, keepout boundaries, etc. Use After launching the command, the cursor will change to a crosshair and you will enter interactive routing mode. Simply click or press Enter to anchor the starting point for the track you wish to route. If you place a track on a design object that has a net name, the track you are placing will adopt the net name, becoming part of that net. If you click on a connection line, the cursor will jump to the nearest pad and then keep the connection line attached to the end of the track you are placing. Documentation Library > Board Layout > Search results	Т	his command is used to place track objects onto the current document.
After launching the command, the cursor will change to a crosshair and you will enter interactive routing mode. Simply click or press Enter to anchor the starting point for the track you wish to route. If you place a track on a design object that has a net name, the track you are placing will adopt the net name, becoming part of that net. If you click on a connection line, the cursor will jump to the nearest pad and then keep the connection line attached to the end of the track you are placing. Documentation Library > Board Layout > Search results AP0107 Impedance-Controlled Routing AP0135 Interactive and Differential Pair Routing AP0136 Interactive and Differential Pair Routing AR0126 Internal Power and Split Planes AR0128 Situs Autorouting Essentials AR0128 Editor and Object Reference AR0129 CB Editor and Object Reference TR0114 Altium Designer Panels Reference CT0117 Getting Started with PCB Design Select Place > Interactive Routing from the menus [shortor 	is a p: d	the process of placing track segments. Tracks are generally placed on signal layer to form the electrical interconnection between component ads on a PCB. However, tracks are also used as general-purpose line rawing elements to create board outlines, component outlines, polygon
After launching the command, the cursor will change to a crosshair and you will enter interactive routing mode. Simply click or press Enter to anchor the starting point for the track you wish to route. If you place a track on a design object that has a net name, the track you are placing will adopt the net name, becoming part of that net. If you click on a connection line, the cursor will jump to the nearest pad and then keep the connection line attached to the end of the track you are placing. Documentation Library > Board Layout > Search results AP0107 Impedance-Controlled Routing AP0135 Interactive and Differential Pair Routing AP0136 Interactive and Differential Pair Routing AR0126 Internal Power and Split Planes AR0128 Situs Autorouting Essentials AR0128 Editor and Object Reference AR0129 CB Editor and Object Reference TR0114 Altium Designer Panels Reference CT0117 Getting Started with PCB Design Select Place > Interactive Routing from the menus [shortor 	U	98
are placing will adopt the net name, becoming part of that net. If you click on a connection line, the cursor will jump to the nearest pad and then keep the connection line attached to the end of the track you are placing. Documentation Library > Board Layout > Search results AP0107 Impedance-Controlled Routing AP0135 Interactive and Differential Pair Routing AP0138 Pin and Part Swapping with Dynamic Net Assignment AR0126 Internal Power and Split Planes AR0128 Situs Autorouting Essentials AR0128 Situs Autorouting Essentials AR0124 Difference AR0124 Delidior and Object Reference AR0125 Difference AR0126 Place > Interactive Routing from the menus [shortor Select Place > Interactive Routing button Start interactive routing as before, but use the * key to .	At yo	fter launching the command, the cursor will change to a crosshair and ou will enter interactive routing mode. Simply click or press Enter to
and then keep the connection line attached to the end of the track you are placing.		
 [™] AP0107 Impedance-Controlled Routing [™] AP0135 Interactive and Differential Pair Routing [™] AP0138 Pin and Part Swapping with Dynamic Net Assignment [™] AR0126 Internal Power and Split Planes [™] AR0128 Situs Autorouting Essentials [™] AR0128 Situs Autorouting Situs Autorouting Essentials [™] AR0128 Situs Autorouting Situs Autorouting Situs [™] AR0128 Situs Autorouting Situs Autorouting Essentials [™] AR0128 Situs Autorouting Situs Autorouting Situs Autorouting Situs [™] AR0128 Situs Autorouting Autorouting Situs [™] Autorouting Situs Autorouting Asset Autorouting	a	nd then keep the connection line attached to the end of the track you are
 [™] AP0135 Interactive and Differential Pair Routing [™] AP0138 Pin and Part Swapping with Dynamic Net Assignment [™] AR0126 Internal Power and Split Planes [™] AR0128 Situs Autorouting Essentials [™] AR0124 Altium Designer Panels Reference [™] TR0104 Altium Designer Panels Reference [™] TR0112 PCB Editor and Object Reference [™] TR0116 Design Rules Reference [™] TR0117 Getting Started with PCB Design ▶ Select Place > Interactive Routing from the menus [shortc ▶ T] or click the Interactive Routing button 		Documentation Library > Board Layout > Search results
AP0138 Pin and Part Swapping with Dynamic Net Assignment AR0126 Internal Power and Split Planes AR0128 Situs Autorouting Essentials AR0128 Situs Autorouting Essentese Essente AR0128 Situs Autorouting Essentese AR0128 Situ		🗉 🖄 AP0107 Impedance-Controlled Routing
 ☆ AR0126 Internal Power and Split Planes ☆ AR0128 Situs Autorouting Essentials ☆ TR0104 Altium Designer Panels Reference ☆ TR0112 PCB Editor and Object Reference ☆ TR0116 Design Rules Reference ☆ TU0117 Getting Started with PCB Design > Select Place > Interactive Routing from the menus [shortc T] or click the Interactive Routing button > Start interactive routing as before, but use the * key to . 		et al la construction de la constru
 [™] AR0128 Situs Autorouting Essentials [™] [™] TR0104 Altium Designer Panels Reference [™] [™] TR0112 PCB Editor and Object Reference [™] [™] TR0116 Design Rules Reference [™] [™] TR0116 Design Rules Reference [™] [™]		🗄 🎦 AP0135 Interactive and Differential Pair Routing
 		🗉 🖄 AP0138 Pin and Part Swapping with Dynamic Net Assignment
 [™] TR0112 PCB Editor and Object Reference [™] TR0116 Design Rules Reference [™] TU0117 Getting Started with PCB Design Select Place > Interactive Routing from the menus [shortd > T] or click the Interactive Routing button Start interactive routing as before, but use the * key to . 		🗄 🖄 AP0138 Pin and Part Swapping with Dynamic Net Assignment 🗄 🖄 AR0126 Internal Power and Split Planes
 [™] TR0116 Design Rules Reference [™] TU0117 Getting Started with PCB Design Select Place > Interactive Routing from the menus [shortd [™] Select Place > Interactive Routing from the menus [shortd [™] Select Place > Interactive Routing from the menus [shortd [™] Select Place > Interactive Routing from the menus [shortd [™] Select Place > Interactive Routing button Start interactive routing as before, but use the * key to . 		
 Mathematical Started with PCB Design Select Place > Interactive Routing from the menus [shortd T] or click the Interactive Routing button Start interactive routing as before, but use the * key to . 		AP0138 Pin and Part Swapping with Dynamic Net Assignment D' AR0126 Internal Power and Split Planes D' AR0128 Situs Autorouting Essentials D' TR0104 Altium Designer Panels Reference
 Select Place > Interactive Routing from the menus [shortd T] or click the Interactive Routing button Start interactive routing as before, but use the * key to . 		AP0138 Pin and Part Swapping with Dynamic Net Assignment AR0126 Internal Power and Split Planes AR0128 Situs Autorouting Essentials D' TR0104 Altium Designer Panels Reference D' TR0112 PCB Editor and Object Reference
Start interactive routing as before, but use the * key to .		AP0138 Pin and Part Swapping with Dynamic Net Assignment AR0126 Internal Power and Split Planes AR0128 Situs Autorouting Essentials D TR0104 Altium Designer Panels Reference D TR0112 PCB Editor and Object Reference D TR0116 Design Rules Reference
		AP0138 Pin and Part Swapping with Dynamic Net Assignment AR0126 Internal Power and Split Planes AR0128 Situs Autorouting Essentials Trollo4 Altium Designer Panels Reference Troll2 PCB Editor and Object Reference Pow TR0116 Design Rules Reference Down TR0117 Getting Started with PCB Design
Laborative Deutine		 ☑ AP0138 Pin and Part Swapping with Dynamic Net Assignment ☑ AR0126 Internal Power and Split Planes ☑ AR0128 Situs Autorouting Essentials ☑ TR0104 Altium Designer Panels Reference ☑ TR0112 PCB Editor and Object Reference ☑ TR0116 Design Rules Reference ☑ TU0117 Getting Started with PCB Design Select Place > Interactive Routing from the menus [shorter T] or click the Interactive Routing button
Interactive Routing 🗸 Search		 ☑ AP0138 Pin and Part Swapping with Dynamic Net Assignment ☑ AR0126 Internal Power and Split Planes ☑ AR0128 Situs Autorouting Essentials ☑ TR0104 Altium Designer Panels Reference ☑ TR0112 PCB Editor and Object Reference ☑ TR0116 Design Rules Reference ☑ TR0117 Getting Started with PCB Design ▶ Select Place > Interactive Routing from the menus [shortcd

Figure 22. Use the Knowledge Center panel to learn about what you are doing, or use it to browse and search the extensive PDF-based documentation library.

determined by your current location in the navigation structure.

Access to Shortcuts

Perhaps the single thing you can do to become more productive in any software environment is to learn the shortcut keys. Keystrokes are more efficient than carefully positioning a mouse over a button or drilling through menus, and once learned become second nature. In a multi-editor environment like Altium Designer it can be hard to remember the shortcuts, particularly those special-purpose ones that are available when you are running a command. To help with this Altium Designer provides a shortcut menu, that can be used from within all interactive Schematic and PCB commands. When a command is running, for example **Place Wire**, simply press the tilde key (~) and a menu will appear, listing all of the valid shortcuts for that stage of the interactive command.

		sharp Er	11 3 3 2 0
_	Help	F1	
	Zoom In	PgUp	
-	Zoom Out	PgDn	
_	Redraw	End	
	Zoom Pan	Home	r
	Move cursor Left by one grid (Shift D	own ×10) Left	
-	Move cursor Right by one grid (Shift [Down x10) Right	
_	Move cursor Up by one grid (Shift Do	wn ×10) Up	
_	Move cursor Down by one grid (Shift	Down ×10) Down	
	Move cursor Left by ten grid	Shift+Left	
ne	Move cursor Right by ten grid	Shift+Right	
	Move cursor Up by ten grid	Shift+Up	
	Move cursor Down by ten grid	Shift+Down	
	Display popup menu		
	Remove vertex	BkSp	
	Toggle wire start/end mode	Space	
	Cycle wire placement mode	Shift+Space	
	Change through dialog box	Tab	
	Insert vertex	Ins	nt C
		NOT	

Figure 23. Fast access to shortcuts available during an interactive process

Use the menu to refresh your memory about the shortcuts available, or use it in the traditional menu sense, to select the required option with the mouse.

By keeping the **Shortcuts** panel visible, you can quickly peruse the shortcuts available to you, depending on what you are working on. For example, simply having a schematic document open will show the various Schematic Editor shortcuts.

By Name - By Category - By Shortcut					
Align menu	A			U_ARM	
Align Objects Bottom	Shift+Ctrl+B			Sharp LH79520	with ARM720T
Align Objects Left	Shift+Ctrl+L				
Align Objects Right	Shift+Ctrl+R			PER_DATA[31.0]	
Align Objects Space Equally Horizontal	Shift+Ctrl+H			PER_WEB[3.0]	
Align Objects Space Equally Vertical	Shift+Ctrl+V			PER_WE	
Align Objects To The Current Snap Grid	Shift+Ctrl+D				
Align Objects Top	Shift+Ctrl+T			ARM7 SYS RESET	
Change Single Object	Left_Dbl_Click			PER_CLK	
Clear	Delete			ARM7_SYS_CLK	
Clear All Underlines	Shift+Ctrl+C			PER_INT[4.0]	
Clear Current Filter	Shift+C				
Close Active Document	Ctrl+F4				
Compile the current project	Ctrl+F9				
Сору	Ctrl+C	U2			
Cut	Ctrl+X	Wishbone I	nterconnect		
DeSelect menu	х				
Design menu	D	-< \$0_STB_0 -< \$0_CYC_0	m0_STB_I <		ME_STB_0 PX ME_CYC_0 PX
Display Files Panel	Ctrl+N	-> s0_CYC_0	m0_ACK_0		ME_ACK_I <
Drag Single Object	Ctrl+Left_Click		m0_ADR_I[23.0] => m0_DAT_0[31.0] =>	IO_ADR_0[23.0]	ME_ADR_0[31.0] (***) ME_DAT_[31.0] (***
Duplicate Selected Object	Ctrl+D	s0_DAT_0[7.0]	m0_DAT_I[31.0]	IO_DAT_0[31.0]	ME_DAT_0[31.0]
Edit In-Place	F2	\$0_SEL_0[3.0] =< \$0_WE_0	m0_SEL_I[3.0] => m0_WE_I =>	IO_SEL_O[3.0]	ME_SEL_OB.0
Edit menu	E	✓ <	III0_WE_1	IO_WE_0	ILE_WE_O

Figure 24. The Shortcuts panel – enhanced productivity at your fingertips.

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.

Designator		<u> </u>
	- Width 6mil	Height 60mil
Properties	Eccation A. Grosnin	1. Yotohii
Text	C17	Hide 🔲
Layer	TopOverlay 🗸	Mirror No
Font	Component text can be mirrored inc	년 lependently of the component
Autoposition	outline. If this option is checked, the will be mirrored along the x-axis. Th have the designator text appear on component outline.	component designator text his is useful if you wish to

Figure 25. Using the What's This Help feature to access dialog option information

Web Update

In order to keep your Altium Designer software, libraries and documentation up-to-date, a web update feature is provided (**DXP** » **Check For Updates**).



Figure 26. Web Update - ensuring you have the latest tools

As well as checking for available updates on the Altium Website, you can also configure the feature to check a network location – ideal if your company has multiple installations of Altium Designer. You can also enable automatic checking for updates. All configuration is carried out from the System – Altium Web Update page of the *Preferences* dialog (**DXP** » **Preferences**).

Preferences	
E Constant General View	🥰 System – Altium Web Update
Akium Web Update Akium Web Update Akium Web Update Avigation Backup Projects Panel Projects Panel Proje	Select source for updates Image: Control of the second
Set To Defaults 💌 Save	Load Import From OK Cancel Apply

Figure 27. Configuring the Web Update feature

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 28). Access to this dialog is made by choosing the **System Info** command from the main **DXP** menu.

		tly installed in De e installed server:				🖺 <u>M</u> enu
ArngeCmp	AutoPlacer	CAMtastic	CompMake	CoreBuilder	EditConstraints	1
EDIF	EditEmbedded	EditScript	EditSim	EditVHDL	FpgaFlow	
920 HelpAdvisor	HSE dit	ImporterMan	IntegratedLib	CayerStack	LoadPCAD	
in the second se	₫₽₽ ₽₽₽ MakeLib	PCB	PCB3D	PCBMaker	PdifImporter	
PinSwapper	Placer	ReportGener	Route CCT RouteCCT	SavePCAD	Sch	
اللہ کی کھی کھی کہ	C ScriptingSys	SignalIntegrity	SIM	SimView	SpecctralF	
Targets	TextEdit	VersionControl	in the second se	WaveSim	Workspace	
The second		board are arrang		- 11-1-1-1-1-1-1-1-1-1-1-1-1-1-1-1-1-1-		

Figure 28. 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 29).

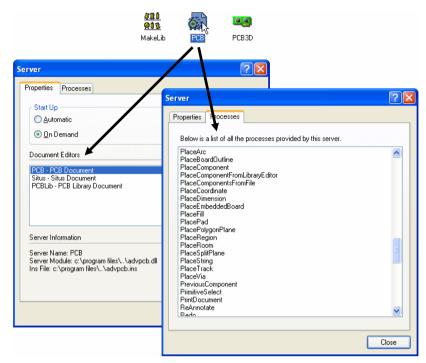


Figure 29. Interrogating server properties

Getting Started with Altium Designer

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

Introduction to Altium Designer -	this article details how and where to find the answers to design questions, and how to get the most out of the v pool of knowledge available for and within Altium Design	ast
An Overview of Electronic Product - Development in Altium Designer	this article describes an entire design from the Engineer perspective, dividing the job between PCB and FPGA design, then using Altium Designer's integrated tools to complete both.	
Getting Started with PCB Design -	this tutorial is designed to give you an overview of how create a schematic, update the design information to a and generate manufacturing output files. It also investi the concept of projects and integrated libraries.	PCB
Getting Started with FPGA Design -	this tutorial gives an overview of how to create an FPG design. It outlines how to create a schematic and then compile, synthesize, build and program a physical FPC 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.
Getting Started with Embedded Software	-	this tutorial shows how to create an embedded software project with Altium Designer.
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.
Connectivity and Multi-Sheet Design	-	this article discusses the structural and connective considerations involved in multi-sheet design, then describes the different browsing tools that let you verify net connectivity across source documents.
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.
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-based panels, which together provide a powerful technique for finding and editing design objects.
Verifying Your Design in Altium Designer	-	this article discusses everything that the designer has to do to be confident that their design is ready for PCB Layout, from the kinds of problems that are common, why they happen, and strategies to assist with building the product right.
Preparing the Board for Design Transfer	-	this tutorial shows how to define the board shape, configure the drawing sheet, setup the layers, and define any keepout requirements, in preparation for transferring the design from the Schematic Editor.
Specifying the PCB Design Rules and Resolving Violations	-	this article introduces the PCB Design Rules System, in particular how rules are created and applied to objects in a design. It also describes how defined rules are checked using online or batch DRC tools and discusses navigating and resolving violations.

Interactive and Differential Pair Routing	-	this application note describes how Altium Designer's interactive routing features can be used to help you efficiently and accurately route your board.
Finding Differences and Synchronizing Designs	-	this article provides an overview of the process used by Altium Designer to keep your schematic and PCB designs synchronized. The facilities for detecting and resolving design differences are covered in particular.
FPGA Designer's Quickstart Guide	-	this guide gives an overview of using the Altium Designer environment to develop an FPGA design.
Design Portability, Configurations and Constraints	-	this article describes what is required for design portability and the role of configurations and constraints in achieving this portability.
Implementing a Simple Processor- Based Design in an FPGA	-	this tutorial describes how to implement a processor-based design in an FPGA. It describes the creation of FPGA and Embedded projects, creating C files, setting up processor and compiler options and then configuring and programming the design to an FPGA device.
Component, Model and Library Concepts	-	this article defines components, models and libraries, and their relationships. The search sequence for locating models is explained, as well as options that make this search more restrictive for specific models.
Creating Library Components	-	this tutorial covers the creation of schematic components and PCB footprints using the Schematic and PCB Library Editors in Altium Designer.
Using Components Directly from Your Company Database	-	this application note provides detailed information on using components from a database using Altium Designer's database library feature.
Customizing the Altium Designer Resources	-	this tutorial describes how to customize your Altium Designer resources, such as commands, menus, toolbars and shortcut keys. It covers rearranging the existing menus and toolbars, adding and deleting toolbar or menu commands, creating new drop-down menus or toolbars and working with shortcut key tables.
Moving to Altium Designer from Protel 99 SE	-	this article outlines the process you go through to transfer a Protel 99 SE design into the Altium Designer environment.
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
30-Nov-2005	1.1	Reviewed and updated for Altium Designer 6
19-Jan-2006	1.2	Image updates and additional document links in the section 'Getting Started with Altium Designer'.
19-Jun-2006	2.0	Updated for Altium Designer 6.3.

Software, hardware, documentation and related materials:

Copyright © 2006 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, Board Insight, CAMtastic, CircuitStudio, Design Explorer, DXP, LiveDesign, NanoBoard, NanoTalk, Nexar, nVisage, P-CAD, Protel, SimCode, Situs, TASKING, and Topological Autorouting and their respective logos are trademarks or registered trademarks of Altium Limited or its subsidiaries. All other registered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.