



Moving to Altium Designer From P-CAD

Summary

Application Note

AP0130 (v2.9) February 28, 2008

This application note highlights the key differences you need to be aware of when moving from P-CAD to Altium Designer. It will help you ramp up your productivity and quickly take advantage of this powerful and flexible electronic product development environment.

You've made the switch to Altium Designer – a single, unified application that incorporates all the technologies and capabilities necessary for complete electronic product development – and now you're eager to start designing. This application note gives you a jump start on the basics of designing in Altium Designer, and maps out the key differences you need to understand when moving from the P-CAD environment to Altium Designer. It also shows how easy it is to transfer your current P-CAD schematic and PCB designs and libraries into Altium Designer.

Getting Started – Transferring Your P-CAD Design

Translating complete P-CAD designs, including schematics, PCB layout, and library files can all be directly handled by Altium Designer's **Import Wizard** without converting to ASCII first – thus avoiding the need for having P-CAD installed. The Import Wizard removes much of the headache normally found with design translation by analyzing your files and offering many defaults and suggested settings for project structure, layer mapping, PCB pattern (footprint) naming, and more. Complete flexibility is found in all pages of the wizard, giving you as little or as much control as you would like over the translation settings before committing to the actual translation process.

File Translation

P-CAD design files in the **Import Wizard** translate as follows:

- P-CAD PCB (*.PCB) files translate into Altium Designer PCB files (*.PcbDoc).
- P-CAD schematic (*.SCH) files translate into Altium Designer schematic files (*.SchDoc). Each sheet within a .SCH file is imported as a single Altium Designer schematic file (*.SchDoc). Design hierarchy is maintained, including complex hierarchy.
- These files will be grouped into an Altium Designer PCB project (*.PrjPCB) that is automatically created.
- P-CAD PCB files generate an output job document (*.OutJob) if necessary. This document will contain all the print settings from the P-CAD PCB.

P-CAD library (*.LIB and *.LIA) files translate as follows:

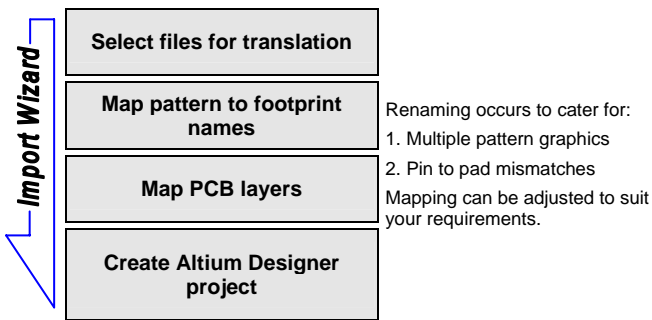
- Libraries that contain solely pattern information translate into Altium Designer PCB library files (*.PcbLib).
- Libraries that contain both pattern and symbol information translate into both Altium Designer PCB library files (*.PcbLib), and schematic library files (*.SchLib) respectively.
- Libraries that contain both component and symbol information translate into Altium Designer schematic library files (*.SchLib). Libraries which contain solely symbol information **do not** import as Altium Designer does not have the same concept of a symbol as P-CAD (described later).

Translated P-CAD libraries are automatically grouped in an integrated library package (*.LibPkg).

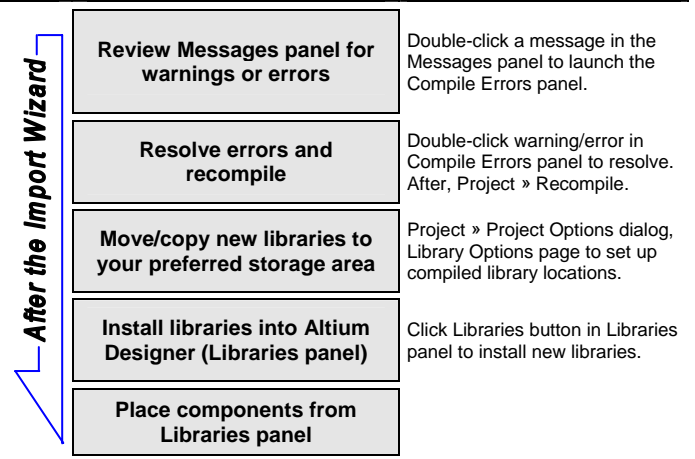
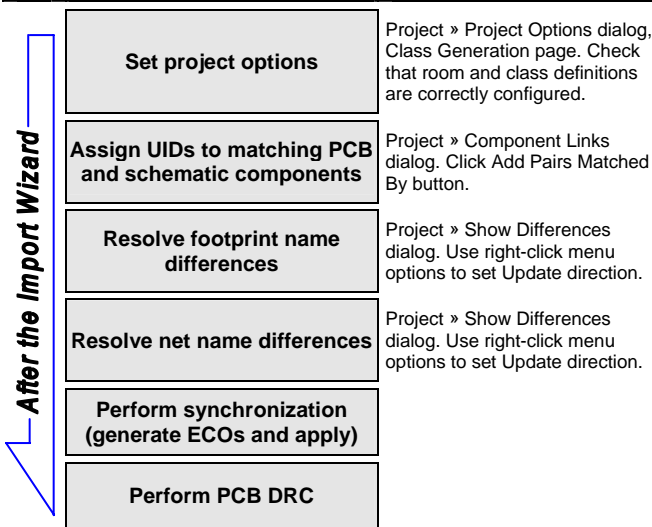
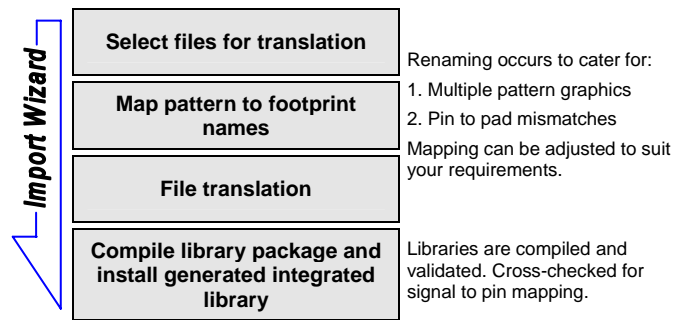
Translation Overview


The steps for translating your P-CAD designs and libraries using the **Import Wizard** look like this:


Translating P-CAD Designs



Translating P-CAD Libraries



 For a tutorial that steps you through all the basics of creating components, read [Creating Library Components](#).

 The fundamentals of how components are defined, their properties, and basic relationships between components, models and library concepts are explained further in [Component, Model, and Library Concepts](#).

From the Knowledge Center in Help, links and videos for understanding the essentials of component, model, and library concepts in Altium Designer can be found in the Documentation Library » Library and Component Management.

Using the Import Wizard

The **Import Wizard** can be launched from the Altium Designer **File** menu. Click on this menu command to launch the wizard as shown below in Figure 1. Right-mouse command menus are available for further control over the translation process through each page of the wizard.

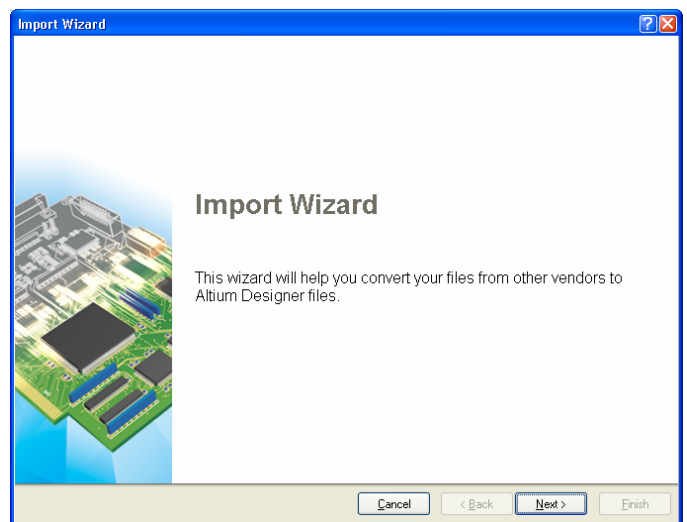


Figure 1. Import Wizard as started from the File Menu. You can also launch this wizard by opening a P-CAD file through File » Open.

The Altium Designer Environment

The Altium Designer environment offers a complete electronic product development environment for all areas of design – from schematic capture to the generation of PCB output, as well as complete FPGA design, development and on-chip debugging.

Perhaps the single biggest difference that you will notice when you start working in Altium Designer is that there is only one application used to create and edit all design files, regardless of the type of file – schematics, PCB, library, text, and so on. No longer will you have to switch between different applications when you want to move from viewing the schematic to the PCB. All the files (also referred to as documents) open in the same executable, each appearing on a separate document Tab within Altium Designer. As you move from one type of document to another the menus and toolbars automatically switch, giving you the right editing environment for that document.

Altium Designer has full support for multiple monitors too. If you have multiple monitors on your PC you can easily drag a document out of Altium Designer and drop it on the second monitor, greatly enhancing your design productivity.

To get you started let's review some of the basic terminology that you'll need to know as you work in Altium Designer.

Working with Documents

In P-CAD, all design work begins on the sheet, the logical working area of the design. There can be multiple schematic sheets within a single P-CAD schematic design file (.SCH file).

In Altium Designer, the logical design area begins with a document, and for each document there is a file stored on the hard drive. This means that for each Altium Designer schematic document (sheet) there is a file, an important conceptual difference to remember.

There can also be multiple design documents of varying types depending on the nature of the design you'll be working on. Getting started, most P-CAD users will be interested in the schematic and PCB document types as these are the files that their designs will be translated to (see Figure 2).

New schematic and PCB document types can easily be created via the **File » New** menu, or by right-clicking on the project in the **Projects** panel (Figure 7).

Workspace Panels

Many elements of the environment will appear intuitive to P-CAD users, helping as you start exploring the system. For example, the **Projects** panel will appear similar to the P-CAD Design Manager; except that since it is not limited to design data it can include the PCB, all libraries, output files, as well as other project documents, such as Microsoft Word or Excel files.

You will also notice that your translated files will be grouped somewhat differently than you are used to seeing. Whether you need to open a specific document such as a schematic, or need information or control to design on a more global, system-wide level, it can all be done using the **Projects** panel.

As you open and make active the documents within various editors you will notice that the resources and available panels will change dynamically; the menus, available panels, and toolbars will quickly change to match the document type you are currently focused on for editing. You'll want to familiarize yourself with how to access these panels, manage, group, and control your display modes to get the most out of the productivity features that are provided here. Press **F1** when the cursor is over a panel for more information on that panel.

Where's all my stuff? Some Basics on Storage Management

All design documents and generated output files, including your translated P-CAD design files are stored as individual files on your hard disk. Your design files can be accessed by opening the project first and then opening the individual documents or any individual document can be opened directly, using the **File » Open** menu command.

Projects Panel

Altium Designer, like P-CAD, also features project management capabilities but there are conceptual differences you'll need to get firm in your mind first. The Altium Designer approach to managing your project is that *all* design documents (schematic, PCB, libraries, etc.) are linked to the project file, both for management and access to certain design features such as design verification, comparison, and synchronization. The Altium Designer presentation through the **Projects** panel provides high

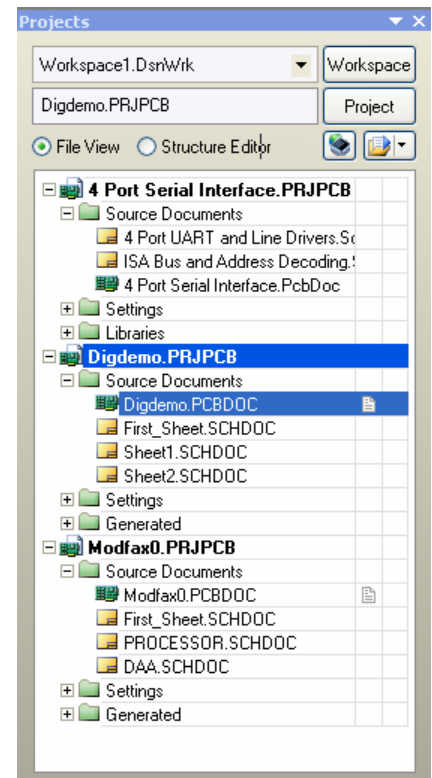


Figure 2. P-CAD design files are displayed immediately after translation in the Projects panel.

visibility and a complete view of everything you need in your project. The project file, which is what you are viewing in the **Projects** panel, contains links to all your documents in your design, as well as any other project-level definitions.

The essentials of project-based design are discussed later in the *Project-based design* section.

Storage Manager Panel

Altium Designer features a dedicated **Storage Manager** panel to allow you greater control over the management of files in your projects. It allows you to navigate the active project in terms of its file storage in Windows. Not only can you see immediately which documents are part of the project and where they are stored, but also see other files that are stored but not explicitly added to the project. The **Storage Manager** is multi-functional and can be used for everything from general everyday file management tasks such as renaming or deleting files, management of backups, through to integrating with your company's version control system. Your project file is listed at the top of the panel in the **Folders** region, corresponding to the root directory, with all other folders and sub-folders contained within that directory displayed below, in their storage hierarchy.

This **Files in Project** region lists all documents currently stored in the root directory or sub-folder thereof, for the active project.

If the active project in the **Folders** region is under version control and you are using either the CVS or SVN version control systems, then selecting a document belonging to that project (in the **Files** region of the panel) will populate the **VCS Revisions** region with a history list for that document.

The **Local History** region presents a local history for the currently focused document in the **Files** region of the panel.

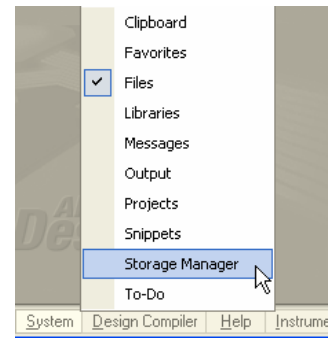


Figure 3. The Storage Manager panel is invoked via the System button at the bottom of the Altium Designer window.

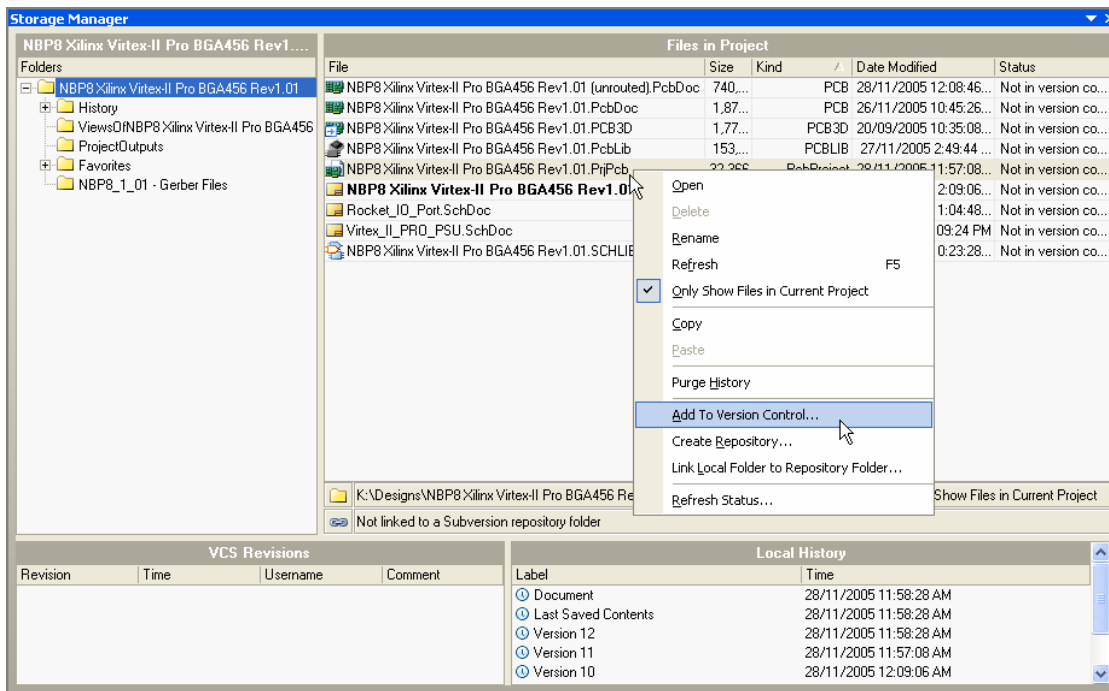


Figure 4. The Storage Manager panel has four key regions: Folders, Files in Project, VCS Revisions, and Local History.

Refer to the document *Welcome to the Altium Designer Environment* for an introduction to Altium Designer and an overview of its unique and unified environment. It provides an illustrated and easy approach to using Workspace panels, storage management, environment customization and much more.

Navigation Toolbar – Direct Document Navigation

Because you can have many design documents and projects open at any time, Altium Designer provides a Navigation toolbar to find the specific design you need quickly. Since everything is integrated into Altium Designer there is no need to switch to another application when you need to view a different type of design file. The Navigation toolbar (Figure 5) is available to assist in the direct navigation of design documents, and can be accessed at any time from within any of the document editors.

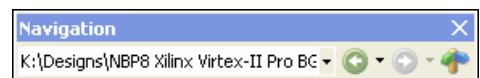



Figure 5. The Navigation Toolbar.

Browsing Viewed Documents

The field at the left of the bar allows you to navigate to any directory or document on a network or local storage directly, as well as any page on the internet. Browsing previously viewed documents is easy using the left and right arrow buttons to go forward and back through previous areas just as you would within an Internet Browser.

Integrated Navigation Home Page

Click the Home Page button  to access the Integrated Navigation Home Page, a top level page where all navigation support pages can be accessed as well as product updates and Altium SUPPORTcenter accounts (Figure 6).

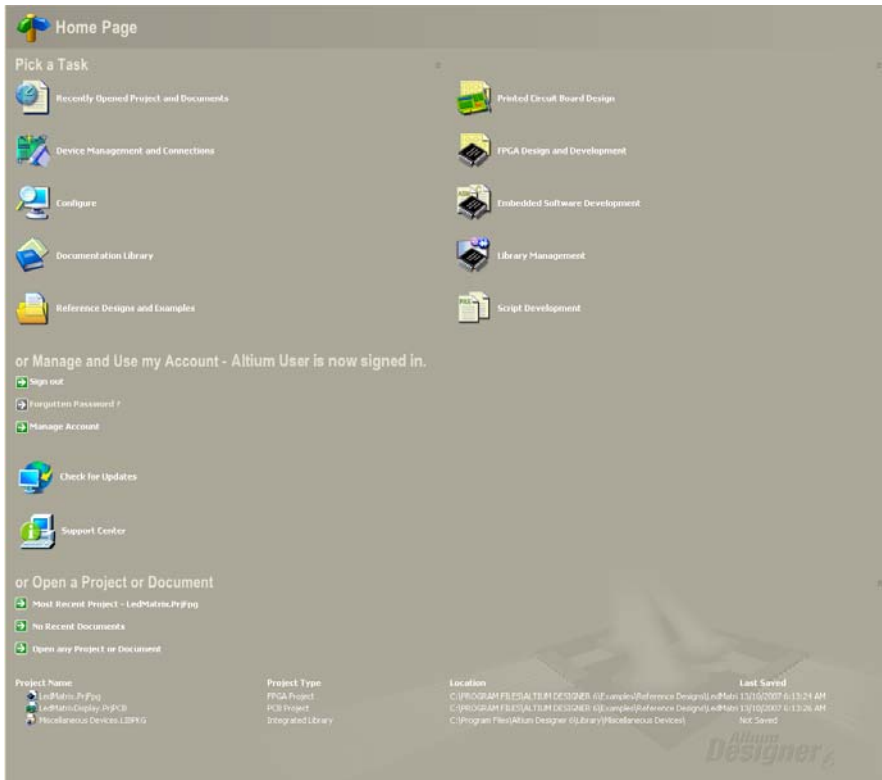


Figure 6. The Navigation Home Page.

Favorites

Like an internet browser, Altium Designer supports the concept of defining Favorites. Once the **Favorites** panel is displayed (via the **System** button on the Status bar) you can right-click in the **Favorites** panel to mark the current view of the active document as a favorite. Double-click on any favorite listed in the panel to return to that document, zoomed to the exact area and location you require.

As well as views of documents in your design, favorites can include links to any directory or document on the network or local storage, as well as any page on the internet.

Immediate Access to Help

For further information about the Favorites panel as well as many other topics in Altium Designer, open the **Knowledge Center** panel (via the **Help** button on Status bar). When the **Knowledge Center** panel is open it will auto-load help on the object, command, or menu entry currently under the cursor if you pause, or alternatively you can press **F1** to load the topic immediately.

Project-based Design

Now that we've covered some of the basics of the Altium Designer environment, it's time to talk about designing. The starting point for every design created in Altium Designer is a project.

It's a simple and important concept – an Altium Designer project is a set of design documents whose output defines a single implementation. For example, the schematics and PCB in a PCB project output the files set required to manufacture a single printed circuit board, while the schematics and HDL in an FPGA project the files set required to program a single FPGA. The project file brings together all those design documents that make up the project.

Altium Designer supports a number of different types of projects, including: PCB, FPGA, Embedded Projects, Core Projects, Integrated Libraries, and Script Projects.

Projects Panel

In Altium Designer, all items related to a project are linked to a Project document, and are easily accessible and manageable in one location. The **Projects** panel is probably one of the more commonly-used panels in day to day work as it allows you to make changes to your project options, add to and remove documents from the project, change the display options of projects, change the order of documents within a project, or even how you would like to display information in the **Projects** panel.

All of your translated files will appear within the **Projects** panel with their own respective projects automatically created for them. Right-click on the project document to display a context-sensitive command menu, giving access to project-relevant editing commands (Figure 7).



Refer to the application note [Project Essentials](#) for all the basics of creating project files, adding and removing files from a project, setting project options, as well as understanding the various project types. It also explains how to group related projects together into a Workspace, ideal for managing multi-board projects.

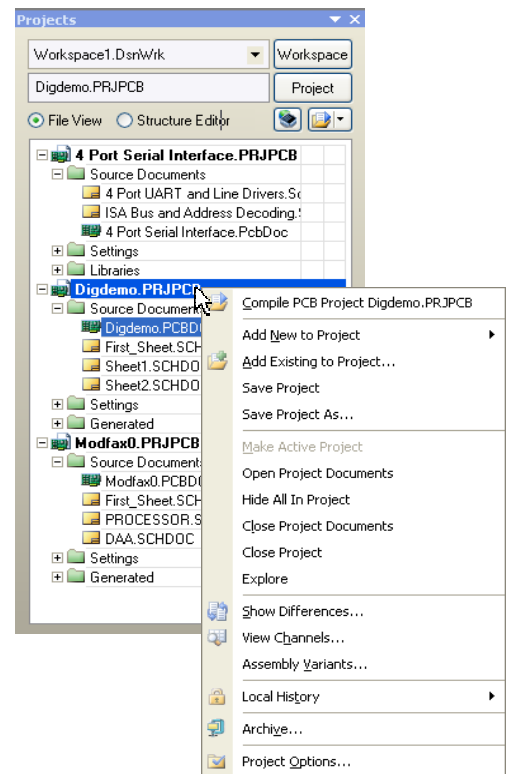


Figure 7. Right-click shows all project-related menu commands.

Compilation – a Cornerstone of Altium Designer

Compilation is a cornerstone concept of the Altium Designer environment, and a fundamental difference from P-CAD. Compilation is a process that allows you to harness many powerful design features and can be done with your translated P-CAD schematics, or even just a netlist! Compilation can also be done on other types of documents such as library documents (described later in this application note).

When you select **Compile Project** from the **Project** menu the compilation process works out the structural relationships between the source schematic (or HDL) documents in the project, then determines the net-level connectivity within each sheet, and finally the connectivity between the sheets. All this component and connective intelligence from your schematics design is written into an internal data structure that can then be used for many post-compilation activities, such as comparing and showing differences between schematics, parameter managing, parametric navigation of your design, cross probing back and forth between the schematics and PCB, and much more.

Where are my nets and components from my design?

You're going to notice that connectivity is not explicit in your design as it was before in P-CAD, but rather has to be extracted from the design using the compilation process. This is available through the right-click menu in the **Project** panel, or using the **Project » Compile Project** menu command.

Once the design is compiled the sheet-level hierarchy, as well as all the components, nets and buses are displayed in the **Navigator** panel. From here you can easily locate any component, bus, net or pin throughout the entire design. And if you hold the **ALT** key as you click on an object in the **Navigator** panel it is highlighted on the PCB as well as the schematic – no longer will you need to inspect netlists to review design connectivity.

Verifying Your Design – Expanded Error Checking

Another benefit that results from compiling a project in Altium Designer is built-in error reporting. This is completely configurable for your needs and can be done before your project is compiled. Right-click either on the project file and invoking the **Project Options** command, or also through the **Project** menu.

The **Error Reporting** tab allows you to fully configure all the errors and warnings that you'd like to run before running a compile, as shown in Figure 8.

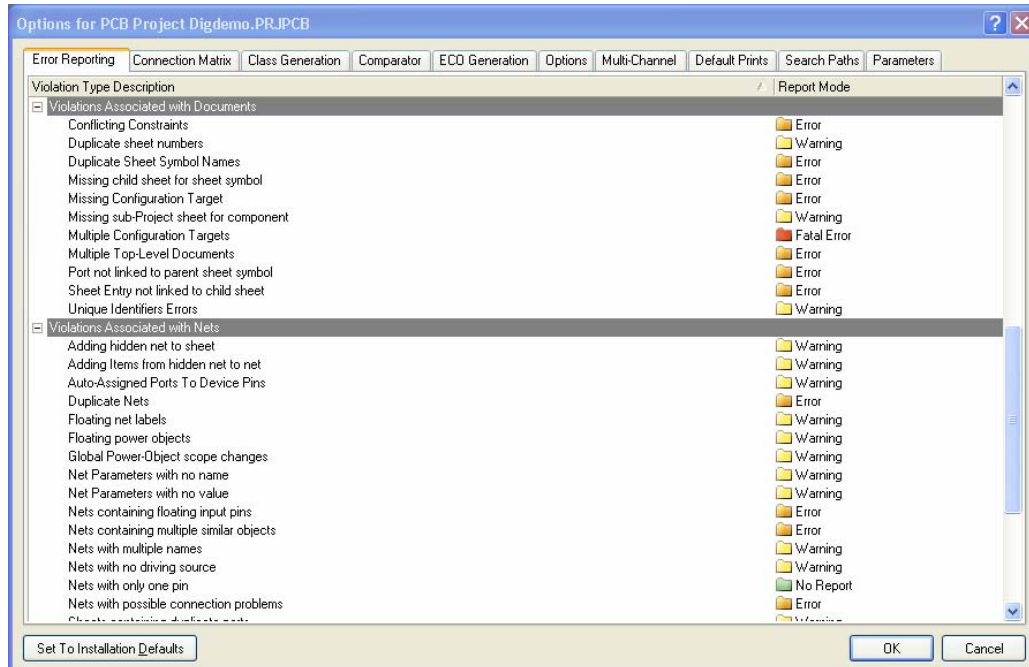


Figure 8. Error Reporting tab in Project Options dialog.



You may wish to get a better picture of the entire development cycle and how it unfolds from an engineer's perspective by reading [An Overview of Electronic Product Development in Altium Designer](#).

Defining Your Net Connectivity – it is Different

In P-CAD, net connectivity is made using sheet connectors, ports, modules, and power parts. Nets between sheets within a single schematic design (.SCH file) are connected through sheet connectors and ports. Hierarchy is represented using modules while power parts are used to create global nets throughout the design. There are no net identifiers for referencing a schematic external to the schematic design file.

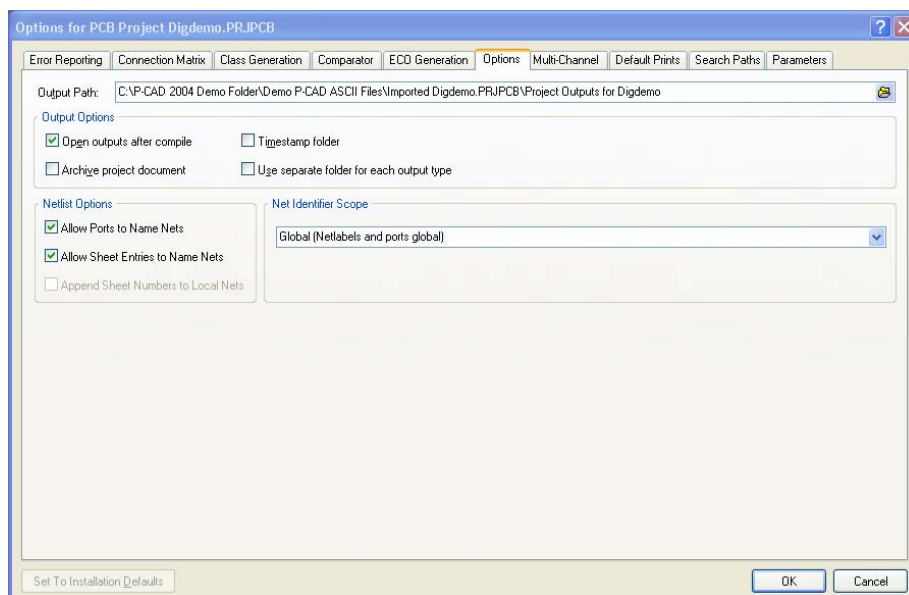


Figure 9. The Net Identifier scope can be set from within the Options tab of Project Options.

Altium Designer uses a slightly different set of net identifiers to create net connectivity. Within a schematic sheet you can use *Wires* and *Net Labels*. Between schematic sheets, nets in a flat design are typically connected using *Ports*, but *Off-Sheet Connectors* are also available. Nets in a hierarchical design are connected from a *Port* on the lower sheet to a *Sheet Entry* of the same name, in the sheet symbol that represents the lower sheet. Power/ground nets are connected using *Power Ports*.

Configuring the Design Connectivity

Altium Designer supports different types of design connectivity, and this must be set to suit the structure of the design. The type of sheet-to-sheet connectivity is referred to as the *Net Identifier Scope*. It is set in the **Options** tab of the *Options for Project* dialog, and saved with the project.

From the **Project** menu, select the **Project Options** menu command, and go to the **Options** tab as shown in Figure 9.

In the **Net Identifier Scope** drop-down you may select from the following connectivity options:

- Automatic (Based on project contents)
- Flat (Only ports global)
- Hierarchical (Sheet entry <-> port connections)
- Global (Net labels and ports global)

The **Import Wizard** handles connectivity automatically through the translation process and will give you the **Automatic (Based on project contents)** configuration by default. This option is an instruction to Altium Designer's design compiler to determine which of the other three options are best suited for the connectivity in your design. Hierarchical modules are mapped as sheet symbols, and they will translate to sheet symbols in Altium Designer. Pins in modules are created as sheet entries in the sheet symbol. In Automatic mode, the design compiler then looks at the sheet symbols on the top sheet. If there are sheet entries (hierarchical pins) in them, it will assume vertical connectivity, and internally use the Hierarchical option. If there are no sheet symbols on the top sheet, or if there are sheet symbols but they do not include any sheet entries, it will assume horizontal connectivity for which there are two ways that Altium Designer supports this: Flat and Global. In order to determine which of these two options to use, the design compiler looks for ports or off-sheet connectors on the sub-sheets. If there are any it uses the Flat option, if there are no ports it uses the Global option.

Remember that you can easily go back and change this configuration after the translation process through the *Project Options* dialog from the **Projects** menu.



Defining net connectivity, net identifiers, scoping and how it all relates to multi-sheet design is a must read for P-CAD users and is fully explained in the article [Connectivity and Multi-Sheet Design](#).

Design Synchronization

Design synchronization is fully integrated in Altium Designer without the need for passing a netlist. Synchronization in Altium Designer is also bi-directional, allowing you to make annotation changes and component property updates in both directions between your schematic and PCB, in a single operation.

Again, an important and fundamental premise of Altium Designer is that the setup of the design's connectivity is driven from the schematic through to the PCB. If you are making connectivity changes in the opposite direction (from PCB to Schematic), a report is generated and these updates can then be performed on the schematic.

The synchronization feature is used when you first transfer from the schematic to the new blank board, or when you make design changes that need to be passed over.



For more information on design transfer and design synchronization, read the article [Finding Differences and Synchronizing Designs](#).

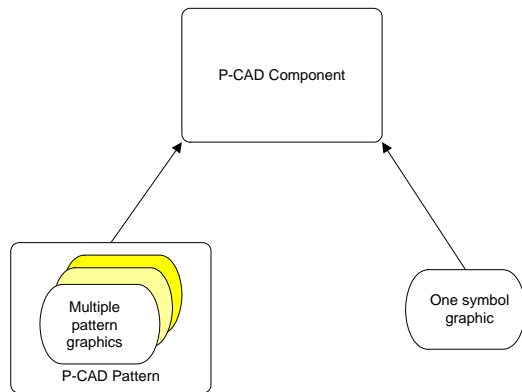
As well as being able to detect electrical differences, such as changed designators, component values or net connectivity, Altium Designer also include a physical difference engine, which can find schematic and PCB layout changes – ideal for examining changes between different revisions of a board.

The Schematic Symbol *Is* the Component...

As an expert P-CAD user, you'll know that components form the basic building blocks of design in P-CAD, and the symbol is merely a graphical representation of that component in the schematic. But in Altium Designer the symbol *is* effectively the component for all phases of design, and not just the schematic capture portion of it. A little comparison will help show the differences of how the two are modeled between the respective systems for a better understanding.

From the Knowledge Center in Help, links and videos for understanding the essentials of component, model, and library concepts in Altium Designer can be found in the Documentation Library » Library and Component Management.

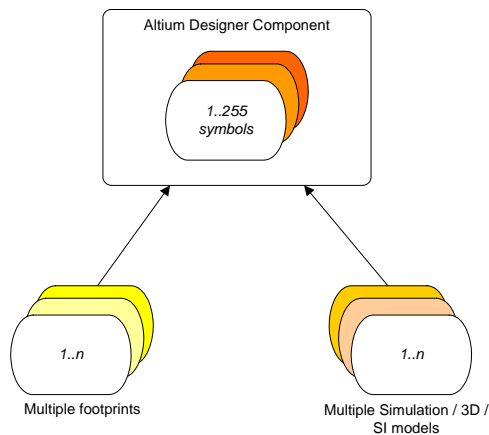
P-CAD Components



In P-CAD, all of the logical and electrical data that is held in the component can be seen in Library Executive in the *Pins View* dialog. Pin and gate swapping component pin to symbol pin, and pattern pad mapping, along with the pin's electrical and logical data is the only component information available. Because this information relates primarily to the pins and is somewhat limited, there are inherent restrictions to the number of ways that P-CAD components can be represented throughout the design process. An Altium Designer component, on the other hand, contains more information and is more flexible in terms of how it can be represented.

Figure 10. P-CAD components have a single symbol graphic and one or more pattern graphics for each pattern

Altium Designer Components



In Altium Designer, the logical symbol is assumed to be the essential starting point of a component. It can be initially defined at minimum as a name in a schematic library to which pins and any graphical symbols or alternative display options needed for implementation may be added. This flexibility allows a component to be represented in different ways during the design and capture process. This may not only be as a logical symbol on the schematic, but also be a footprint on the PCB or even as a SPICE definition for simulation.

Figure 11. Altium Designer symbols can have multiple footprints (pattern graphics) and symbol models



The fundamentals of how components are defined, their properties, and basic relationships between components, models and library concepts are explained further in the article [Component, Model, and Library Concepts](#).

True Multi-channel Design Finally Arrives

Complex hierarchy is the general term used throughout the industry to describe the process of using multiple instances of the same sheet in a schematic hierarchy. This important concept is supported by Altium Designer.

Multi-channel Design

Traditionally, a design that included complex hierarchy had to go through a process of ‘flattening’ or ‘expanding’ the hierarchy at some point, to uniquely instantiate every component and net. Altium Designer does not need to do this, so this multiple-instantiation capability is referred to as multi-channel design instead of complex hierarchy.

Like complex hierarchy, multi-channel design is the ability to reference a child sheet multiple times. It can be done by placing multiple sheet symbols, each referencing the same sub-sheet, or it can be done by placing a single sheet-symbol and using the **Repeat** statement to generate an array of sub-sheets. This is built on the complex hierarchy architecture of multiple instances, but in this case the parent object is expanded by the design compiler at the time of compilation.

Multi-channel design also supports multiple levels. For example, a 32-channel design could be structured over two levels, having 4-banks of 8-channels, to create the final 32-channels. Additionally you can wire signals to either all of the channels or use a bus where one member of the bus goes to each channel. Altium Designer is the only electronic design platform to offer this concept.



There are several example multi-channel designs that come with Altium Designer that you may wish to look at. These include the *Multi-Channel Mixer*, *Peak Detector* and *PortSwitcher*. All three designs can be found in the `\Examples\Reference Design` folder. Once you have opened one of the examples you should compile it, and then look for the tabs at the bottom of each schematic sheet.



For more information on multi-channel designs, refer to the article [Multi-Channel Design Concepts](#).

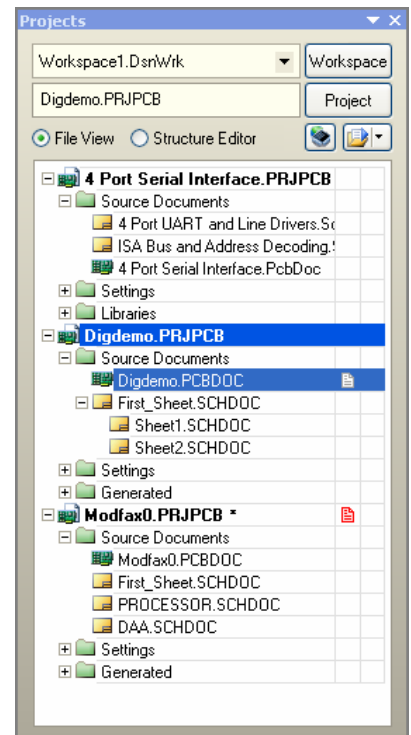


Figure 12. Project hierarchy is easily viewed in the Projects panel.

Libraries

Altium Designer supports working directly from the source symbol or model libraries, an ideal approach when the schematic and PCB are designed by separate organizations. There are also integrated libraries, a term used in *both* Altium Designer and P-CAD, yet each having distinct differences.

In P-CAD, an integrated library is a single library file that has both the pattern (called a footprint in Altium Designer) and symbol information. Components within these libraries have the logical pin designators and pin data specific to the type of component. As explained earlier there are some fundamental differences about how a component is defined between the two environments. In P-CAD, the component definition provides the symbol and patterns with any connectivity intelligence; patterns and symbols alone do not contain any logical information such as pin designator values because they are only a graphical representation.

Altium Designer Libraries

In Altium Designer an integrated library, like in P-CAD, is also one where the source symbol, footprint, and all other information (e.g. SPICE and other model files) are contained in a single library file. However, all this information must be compiled before it can be accessed. During compilation checks are made to see how relationships are defined, validating the relationship between the models and the symbols and bundling them into a single integrated library. Unlike P-CAD, this file can not be directly edited in Altium Designer, offering portability and security.

You can create an integrated library through the **Project** panel via the command **Compile Project** (Figure 7). All of Altium Designer’s 70,000+ components are supplied in integrated libraries, from which the source libraries can be extracted at any time if required. All libraries may be viewed and managed at any time from the **Projects** and **Library** Panels.

Library Types

There are four types of libraries used in the Altium Designer environment: model, schematic, integrated and database.

Model

These libraries contain the models for each component representation as per their design domain and are each stored in their respective “model containers”, called model libraries. In some domains, there will be typically one model per file and they are referred to as model files (*.mdl, *.ckt). In other design domains, models are usually grouped into library files according to how the user has grouped them such as PCB footprints grouped into package-type libraries (*.PcbLib).

Schematic

These libraries contain source schematic components and their model interface definitions (*.SchLib).

Integrated

An integrated library (*.IntLib) is a compiled file, that includes schematic libraries along with all models referenced in the symbols' model interface definitions; which could include footprint model libraries, simulation model files, and three-dimensional model libraries.

Database

Database libraries provide similar functionality to a component information system. When you place from an installed database library (*.DBLib) all data in the component comes from the referenced database.

Where are my libraries? Some Basics on Library Management

You'll be able to view your source schematic and PCB library files immediately after translation through the **Projects** panel. Your translated P-CAD libraries are automatically grouped into one PCB project.

Like P-CAD, libraries are installed (added) to the Altium Designer environment, making their components available in all open projects. Display the **Libraries** panel, from there you can install and remove libraries. Libraries can also be linked to any project, and you can also define project search paths, useful for referencing simulation models.


You'll be able to view your source schematic and PCB library files immediately after translation through the **Projects** panel. Libraries are not automatically translated as Altium Designer integrated (compiled) libraries. They will, however, be automatically grouped into integrated library projects. To create an integrated library from an integrated library project, you will need to do this manually through the **Project** panel via the **Compile Project** command in the right-click menu (Figure 7) when you select a source library.


 Refer to the article [Enhanced Library Management Using Integrated Libraries](#) for a further discussion on library management.

A Brief Note on Database Linking


Appreciating the fact that many designers like to link from the components in their electronic design software to their company database, Altium Designer has strong support for linking and transferring database data through the design process and into the Bill of Materials.

Two techniques are supported, one where the Altium Designer library symbol holds all model references and also includes links into an external database, the second where the database holds all model references and other company information. While database connections in Altium Designer are set up for MS Access databases (*.mdb files) by default, any ODBC-compliant database can be accessed.

 To learn more about linking from Altium Designer components to an external database, refer to the application note [Linking Existing Components to Your Company Database](#).

 For information on placing components directly from a company component database, read the application note [Using Components Directly from Your Company Database](#).

How do I setup this new workspace? Designing the PCB

 A tutorial that covers all the basics of PCB design transfer, including the topics mentioned and more can be found in [Getting Started with PCB Design](#).

PCB Board Wizard

Before you can transfer your design from the Schematic Editor to the PCB Editor, you'll need to have at the very least a blank PCB with at least a board outline. The **PCB Board Wizard** allows you to easily create a basic PCB design using many industry-standard board outlines as well as create your own custom board sizes.

The **PCB Board Wizard** is launched from the **Files** panel in the **New from template** section. At any stage you can use the **Back** button to check or modify previous pages in the wizard.

Placement Grid and Units

All options for the placement grid, measurement units, sheet position, and designator display are found in the *Board Options* dialog. With a PCB document active in the main design window (for this and all of the following context-sensitive dialogs), select **Design » Board Options** [shortcut: **D, O**] from the main menu to open the *Board Options* dialog.

PCB Preferences

Preferences that assist in positioning components, such as Online DRC, Snap to Center and Selection preferences are found in the *Preferences* dialog. Select **Tools » Preferences** [shortcut: **T, P**] from the main menu to open the *Preferences* dialog.

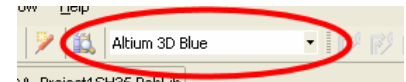
View Configurations (Layers and Colors)

View configurations are settings that control many PCB workspace display options for both 2D and 3D environments, and apply to the PCB and PCB Library Editors. The view configuration last used when saving any PCB document is also saved with the file. This enables it to be viewed on another instance of Altium Designer using its associated view configuration. View configurations can also be saved locally and be used and applied at any time to any PCB document. Any PCB files that you open which do not have an associated view configuration are displayed using a system default one.

Note: The *View Configurations* dialog provides access to 2D color settings for layers and other system-based color settings – these are *system settings*, that is, they will apply to all PCB documents and are not part of a view configuration. Color profiles for the 2D workspace can also be created and saved, similarly to view configurations, and can be applied at any time.

Select **Design » Board Layers & Colors** [shortcut: **L**] from the main menu to open the *View Configurations* dialog. This dialog enables you to define, edit, load and save view configurations. It has settings to control which layers to display, how to display display common objects such as polygons, pads, tracks, strings etc, displaying net names and reference markers, transparent layers and single layer mode display, 3D surface opacity and colors and 3D body display.


You can apply view configurations using the *View Configurations* dialog or by selecting them directly from the drop-down list on the **PCB Standard** toolbar.



You can quickly navigate between the layers of your design by selecting the layer tabs at the bottom of the main design window. Helpful shortcut keys from the numeric keypad include the '+' and '-' for cycling through all visible layers, and the '*' to cycle through visible signal layers.

Layer Stack Manager

Layers are easily managed using the *Layer Stack Manager* dialog. Select **Design » Layer Stack Manager** [shortcut: **D, K**] from the main menu to open it. You can associate nets to planes, change the number of layers, define layer and substrate thickness and reassign electrical layer data in this dialog.

 For further information on setting up your board, refer to the tutorial [Preparing the Board for Design Transfer](#).

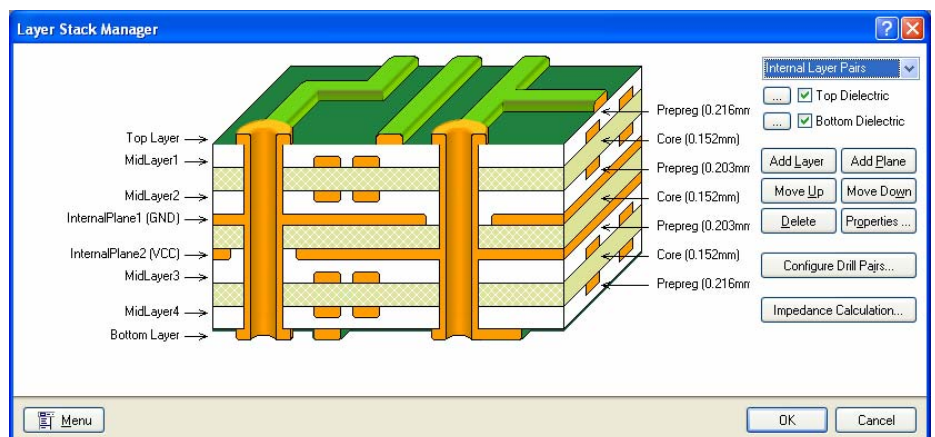
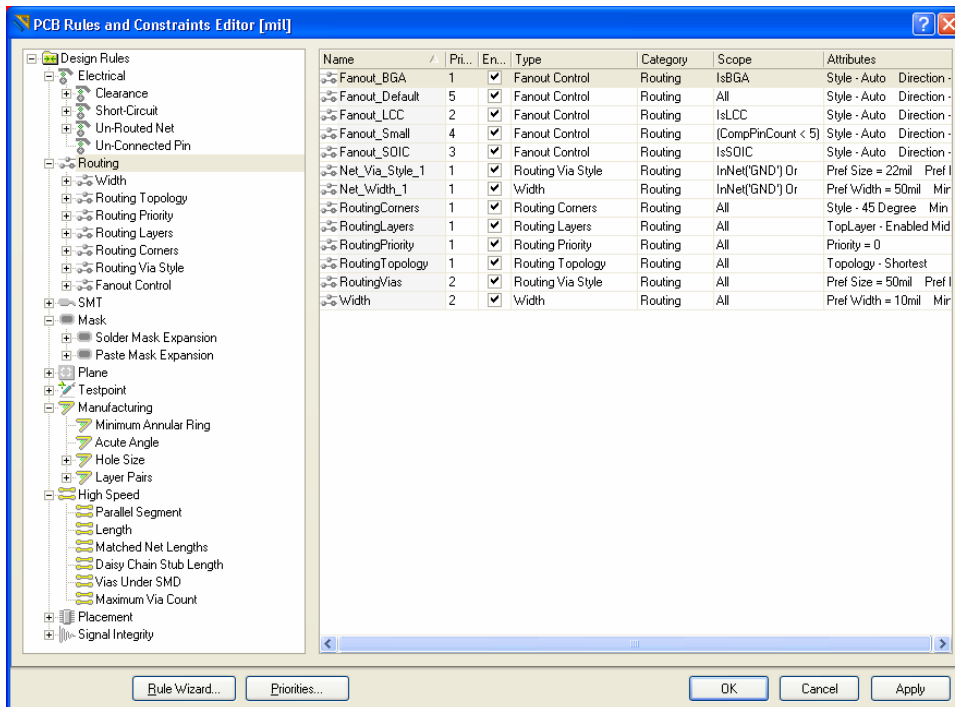


Figure 13. The *Layer Stack Manager* dialog shows a cross-section of the board as you design. Layers may be added or redefined in this dialog.

Design Rules

The PCB Editor is a powerful and dynamic rules-driven environment. This means that as you work in the PCB Editor and do things that change the design (such as placing traces, moving components, or routing the board), the PCB Editor constantly monitors each action and checks to see if the design still complies with the design rules. If it doesn't an error is immediately flagged for your attention.



With the PCB as the active document, select **Design » Rules** from the main command menu to launch the *PCB Rules and Constraint Editor* dialog as shown in Figure 14.

Figure 14. The PCB Rules and Constraint Editor dialog, where all design rules can be managed.

One of the powerful features of Altium Designer's design rule system is that multiple rules of the same type can be defined, each targeting different objects. This is called *scoping*, a new concept for P-CAD users, it allows you to exactly target rules to objects in your design. To say it another way, the exact set of objects that each rule targets is defined by that rule's *scope*. The hierarchy of rules is also user-defined – this is the priority setting that you can see in Figure 14. This combination of rule scoping and priority gives an unprecedented level of control that allows you to precisely target the design rules for your board.

Interactive Route

As with P-CAD, you must have a signal layer active before you can begin routing. Enable the layer that you would like to start on by pressing the **L** shortcut key to display the *View Configurations* dialog. Enable the **Show** option to display a layer – the same as you would do in P-CAD. Once you are showing your signal layer, the tab for it will display in the PCB Editor window. Click on the layer tab at the bottom of the workspace to make it the current or active layer, ready to route on.

There are various interactive routing modes available in Altium Designer. These can be invoked through the **Place** menu or clicking the required routing button in the toolbar.

The following tips will assist you to get a quick start for placing traces (many of these will be the same as in P-CAD):

- **ENTER** or Left-click – Places a start or end vertex in the trace. Placed trace segments appear in the appropriate layer color.
- **SPACEBAR** – Allows you to toggle between the start and end modes for the trace you are placing.
- **SHIFT + SPACEBAR** – Allows you to change the corner mode of your current route
- **END** – Allows you to redraw the screen at any time.
- Shortcut keys **V, F** – Redraw the screen to fit all objects (View Extent).
- **PAGEUP, PAGEDOWN** – Allow you to zoom in or out, centered on the cursor position. The mouse wheel will help you to pan left and right, holding the **CTRL** key down to zoom in and out with the mouse-wheel.
- **BACKSPACE** – Will let you unplace the last trace segment.
- **ESC** or Right-click – Will complete your trace.



Display the **Shortcuts** panel for a dynamic list of shortcut keys available for use wherever you are currently working in Altium Designer, including context-sensitive shortcuts available while running a command. The **Shortcuts** panel can be enabled by clicking the **Help** button in the Status bar.

So how do I get my board manufactured? Setting Up Output Files

Manufacturing an electronic product requires a multitude of output files to be generated. These can include schematic prints for the handbook, assembly drawings, fabrication files, drill files, pick-and-place files, test point reports and the Bill of Materials (BOM).

To manage these outputs Altium Designer includes a specialized output job document (*.OutJob) that can be added to a project to centralize output generation. Any number of outputs can be configured within an output job document, and multiple output jobs can be added to a single project. Once you have defined the sets of output files you require, you can then generate all outputs in a single batch process.

Because output jobs are stored within the project, you can easily regenerate output files at any time. You can work on multiple projects simultaneously, maintaining independent output configurations and file sets for each project. Output configurations can be saved and reused in subsequent projects, allowing you to set up multiple configurations for different manufacturers and produce the correctly configured file set for each fabrication and assembly house.

OutputJobs Editor

The OutputJob Editor allows you to define and manage Output Job Configuration files (*.OutJob). The Output Job file allows you to define all your design output configurations – assembly, fabrication, reports, netlists, etc, all in a single location. You can even create multiple Output Job files and add them to your projects to create a separate assembly output from the fabrication output.

You can create a new file of this type for any active project by using either the **File » New » Output Job File** (Figure 15) command or right-clicking on a project in the **Projects** panel and choosing **Add New to Project » Output Job File** from the pop-up menu that appears.

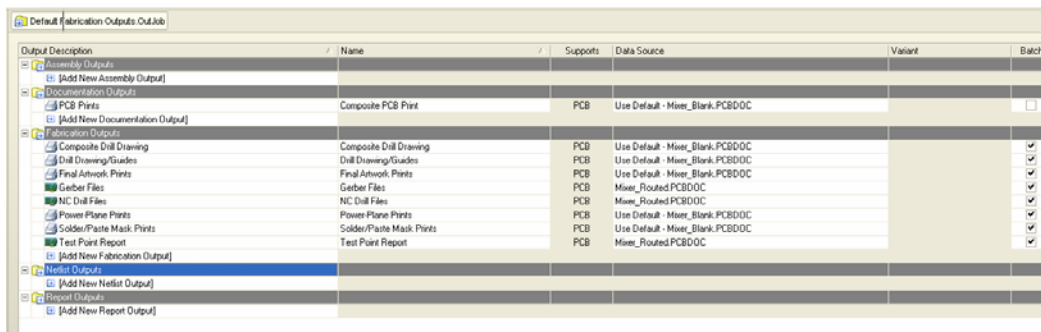



Figure 15. A fabrication output job file for the Multi-Channel Mixer.

The Output Job file is divided into categories that reflect the function of the output. These include Assembly, Documentation, Fabrication, Netlist, and Report Outputs. You'll want to familiarize yourself with how to configure for these and more.

 A comprehensive technical reference for setting up and configuring your output jobs through the OutputJobs editor may be found in the [OutputJob Editor Reference](#). You can open this by pressing **F1** when the cursor is over an open OutJob.

Smart PDF

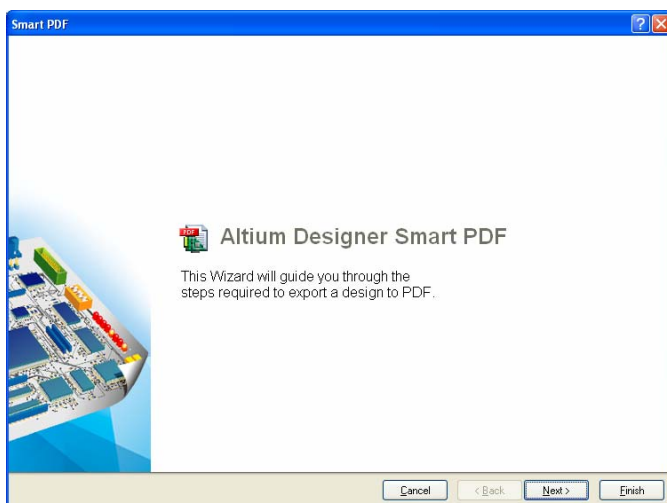


Figure 16. Use Smart PDF to generate live, bookmarked PDFs of your designs, ideal for design reviews and product documentation.

Smart PDF is a built-in PDF generation wizard that quickly generates a PDF of a single schematic sheet, drawings of the PCB, or all the schematics and PCB in a project, complete with clickable bookmarks to each component, net and pins in your design.

The Altium Designer **Smart PDF** wizard is launched from the **File** menu, and will guide you through the steps required to export a design to PDF.

Bonus Technologies

The Bonus Technologies form a more integrated solution in the Altium Designer environment than they previously did in P-CAD. As a result of that there are some key differences to launching and using them that you should be aware of.


Situs Autorouter

Altium's Situs Topological Autorouter engine is fully integrated into the PCB editor.

The Situs engine uses topological analysis to map the board space. Topological mapping provides greater flexibility in route path determination and allows non-orthogonal routing corridors to be more efficiently exploited. Altium Designer also includes full bi-directional support for the SPECCTRA autorouter. During export you can automatically preserve existing board routing, control the mapping of Altium Designer via rules to SPECCTRA pad stacks, and propagate net classes to SPECCTRA to allow efficient generation of class-based routing constraints as you did with P-CAD.

With your active PCB document complete and everything positioned, you're ready to start!


Select the command **Auto Route » All** to open the *Situs Routing Strategies* dialog, just as before in P-CAD. Selecting **Route All** from this dialog will launch the autorouter.

 Routing a board can be a big challenge. For further points on your board setup, configuring your design rules, and running the autorouter, take a look at [Situs Autorouting Essentials](#).

Fabrication and Assembly Files

Altium Designer provides a wide choice of outputs that allow you to generate appropriate files for any manufacturing requirements in the PCB editor. This includes support for all popular Gerber formats, with full control over generated layers and the manual listing or automatic embedding of aperture information.

NC drilling and routing files, automatic pick and place files, and all other relevant manufacturing files can be easily generated, checked and edited from the command **File » Fabrication Output**.

 A tutorial that explores the Quick Load command by importing supplied Gerber, ODB++, and other file types into the CAM Editor is available in [CAM Editor Imports and Exports](#).

Mixed-signal Circuit Simulation

Mixed-signal circuit simulation is a unified part of the capture process, and is fully integrated with the schematic editing environment.


Before you launch the Mixed-signal circuit simulation, make sure that any schematic sheet within the project is open. Simulation models need to be associated with every component in the schematic document before simulation can be launched from the command **Design » Simulate » Mixed Sim**. Simulation is done for the entire project.

If you need to do simulation for only one schematic sheet, then you can create a separate project and add only that sheet to the project. As before in P-CAD, the *Analyses Setup* dialog will launch first so that you can setup all analyses information before running simulation.

Signal Integrity

Signal Integrity is fully integrated into Altium Designer, unlike P-CAD where it is a stand-alone application. It requires that you have, at the very least, a project file that contains at least one PCB and one schematic source document. This is a slightly different requirement from P-CAD where only a PCB design is required.

Signal Integrity is run from the command **Tools » Signal Integrity** where the *Model Assignments Analyses* dialog will launch before Signal Integrity will run.

 A full tutorial that covers the setting up of design parameters like design rules, and Signal Integrity models, starting up Signal Integrity from the schematic and PCB editors, and configuring your tests further can be found in the tutorial [Performing Signal Integrity Analyses](#).

Viewer Edition License






A Viewer Edition license is also available to replace the P-CAD viewer that you may have used before for design exploration and as a cross probing tool, putting the right set of functionality into the hands of designers and engineers across your organization.

The Viewer Edition licensing option of Altium Designer provides quick, easy, and secure read-only exploration of design projects and documents that have been created using Altium Designer. Users can view, print and interrogate all aspects of a design created by Altium Designer making design data more accessible to the entire design-chain. The Viewer Edition not only enhances collaboration within an organization, but also between the design team and external parties, greatly improving design work flow and project productivity.

The Altium Designer Viewer Edition is provided free of charge. You can deploy the Viewer Edition across your organization using your existing Altium Designer license. For non-license holders, contact your nearest Altium sales office to apply for a Viewer Edition license. The license is activated online and registered to the user every 12 months.

For Further Reference

Below are references to other articles and tutorials in the Altium Designer Documentation Library that talk more about the conceptual information as well as walking you through specific tasks. Remember, you can also browse through the Help contents, and use **F1** and **What's This** at any time in a dialog for more details.

-  For more PCB project options, refer to the tutorial, [Getting Started with PCB Design](#)
-  For more FPGA project options, refer to the tutorial, [Getting Started with FPGA Design](#).
You can find both of these in the Getting Started with Altium Designer book.
-  A tutorial that steps you through all the basics of creating components may be found in [Creating Library Components](#).
-  A tutorial that steps you through all the basics of editing multiple objects may be found in [Editing Multiple Objects](#).
-  For getting started with embedded software, refer to the tutorial [Getting Started with Embedded Software](#).

Revision History

Date	Version No.	Revision
02-Feb-2006	2.0	New document release
08-May-2006	2.1	Content reviewed and updated
11-May-2006	2.2	Content reviewed and updated
20-Jun-2006	2.3	Content reviewed and updated
07-Dec-2006	2.4	Content reviewed and updated
21-Mar-2007	2.5	Content reviewed and updated
9-Nov-2007	2.6	Content updated for 6.8
7-Jan-2008	2.7	Updated view configurations info.
11-Feb-2008	2.8	Component body references changed to 3D body.
28-Feb-2008	2.9	Converted to A4

Software, documentation and related materials:

Copyright © 2008 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, Design Explorer, DXP, LiveDesign, NanoBoard, NanoTalk, P-CAD, SimCode, 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.