



Moving to Altium Designer From OrCAD®

Summary

Application Note

AP0132 (v2.5) February 11, 2008

This application note highlights the key differences you need to be aware of when moving from OrCAD® 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 keen to get on with the design process. 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 OrCAD® environment to Altium Designer. It also shows how easy it is to transfer your current OrCAD Capture® and Layout® designs and libraries into Altium Designer.

Getting Started – Transferring Your OrCAD Design

Translating complete OrCAD designs, including Capture schematics, Layout PCB files, and library files can all be handled by Altium Designer's **Import Wizard**. 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 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 file translation settings, before committing to the actual translation process.

File Translation

Files in the Import Wizard translate as follows:

- OrCAD Layout (*.MAX) files translate to Altium Designer PCB files (*.PcbDoc).
- OrCAD Capture (*.DSN) files translate to Altium Designer schematic files. Each page within a .DSN file will be imported as a single Altium Designer schematic file (*.SchDoc). Design caches within a .DSN file will be imported as a schematic library (*.SchLib). Design hierarchy is maintained, including complex hierarchy.
- These files will be grouped into an Altium Designer PCB project (*.PrjPCB) that is automatically created.

OrCAD library files translate as follows:

- OrCAD OLB (schematic library) files will be translated into Altium Designer schematic library files (*.SchLib).
- OrCAD LLB (PCB library) files will be translated into Altium Designer PCB library files (*.PcbLib).
- Translated OrCAD libraries are automatically grouped into one PCB project.

Pin Name and Number Movement in Schematic

It's worthwhile to note as a matter of convenience that if you receive an **Unrecognized Project File Version** error after importing your designs, this error occurs due to the addition of a new feature to OrCAD Capture v10.x, the ability to move pin names and numbers independently from the pin. If the OrCAD Capture schematic is saved normally it will be completely incompatible with OrCAD Capture 9.x and below, and also Altium Designer.

To fix this problem you'll need access to OrCAD Capture v10.x. Make sure the DSN is selected in the project panel, then run **File » Save As**. You will see a small check box saying **Remove Pin Name and Number Movement**. This check box will appear if pin name and numbers have been moved in 10.x. Check it and save the DSN file.

The DSN file can now be imported into Altium Designer.

Default Layer Mapping for PCB

It should also be noted about how the layers are mapped on import for PCB designs. To facilitate the batch import process of multiple designs, there is **Default Layer Mapping**. The Default Layer Mapping is simply a mapping between the names of the foreign PCB layers and Altium Designer PCB layers. Of course you can add, change, or remove as many mappings as you want. This default mapping is then used by the import wizard to build the layer mapping for each PCB, that can then be individually customized. The rationale here is that should you wish to import ten PCB designs and you want to map the layer *Assembly 1* to *Mechanical Layer 1*, you would not have to customize each of the ten PCB designs in order to get the right layer mapping.

The advantage to importing in this manner is that batch management of layer mapping can save a lot of time when importing multiple designs. In this instance the default layer mapping will be saved to your preferences. The disadvantage to using this is that Default Layer Mapping is not always intelligent with differing structures in designs, and so some manual changes may be needed afterwards. You'll need to decide what is best for your situation.

Using the Import Wizard for OrCAD Files

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

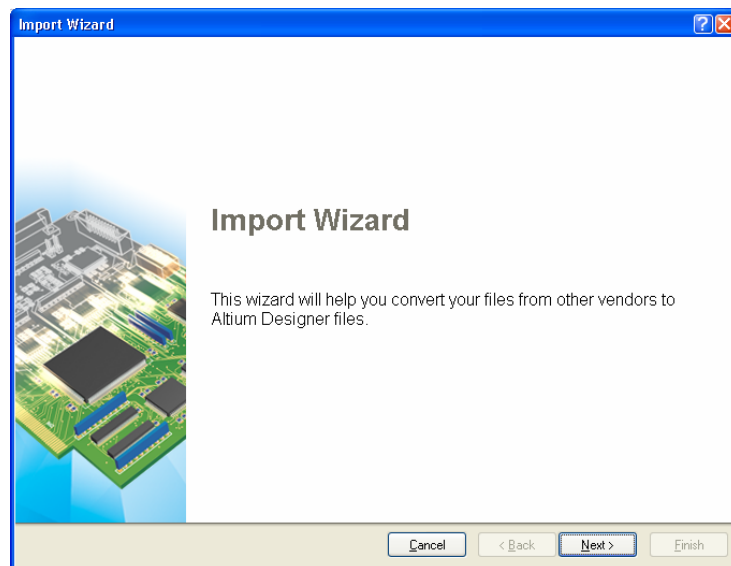


Figure 1. Import Wizard as started from the File Menu.

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 OrCAD Capture all design work begins on the page, the logical working area of the design. There can be multiple schematic pages within a single OrCAD schematic design file (.DSN 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 sheet (page) there is a file, an important conceptual difference to remember.

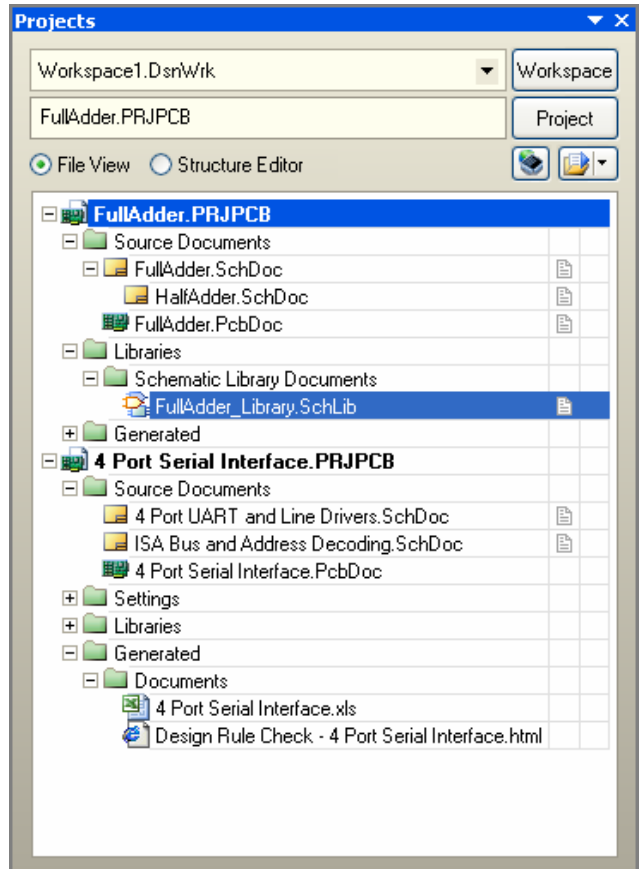


Figure 2. Translated design files are displayed immediately after translation in the Projects panel. Once the project has been compiled the schematic hierarchy will be shown.

There can also be multiple design documents of varying types, depending on the nature of the design you are working on. Getting started, most OrCAD 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 6).

Workspace Panels

Many elements of the environment will appear intuitive to OrCAD users, helping as you to start exploring the system. For example, the **Projects** panel will appear similar to the OrCAD Project Manager, except that since it is not limited to schematic design data it can include the PCB, all libraries, output files, as well as other project documents, such as MS 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 OrCAD design files, are stored as individual files on your hard disk. Your design documents 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 OrCAD, 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 visibility and a complete view of everything you need in your project, not just the schematic part of it. 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

Altium Designer features a dedicated *Storage Manager* to allow you greater control over the management of files in your projects. The *Storage Manager* can be invoked at any time simply by clicking on the **System** button at the bottom of the Altium Designer application window and choosing **Storage Manager** from the pop-up that appears.

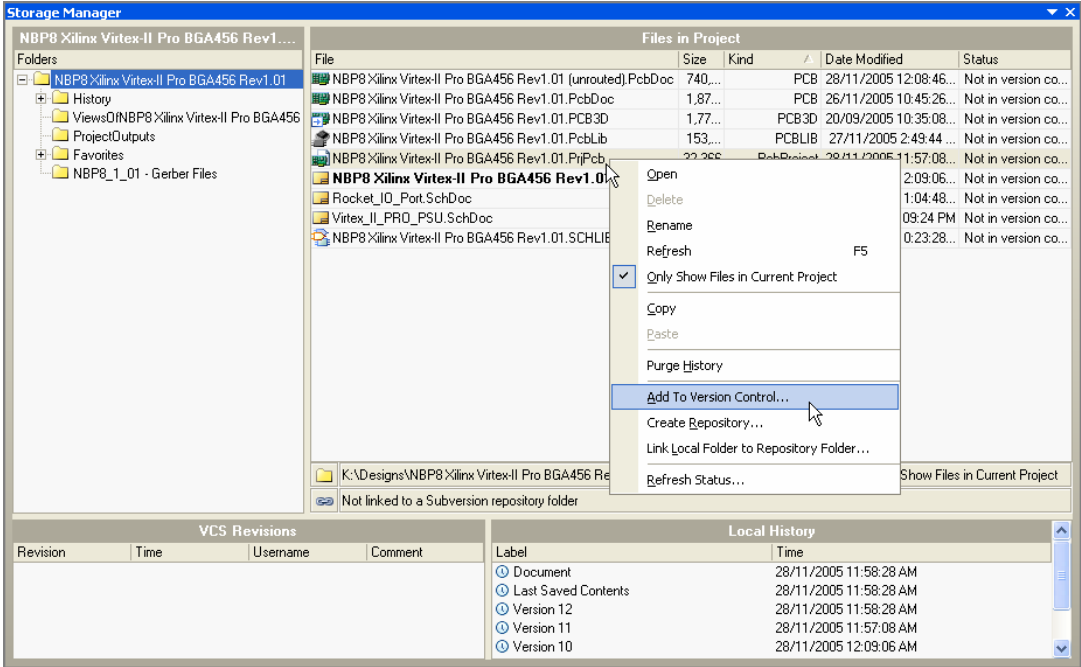


Figure 3. The Storage Manager.

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, as shown above in Figure 3.



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 4) 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.

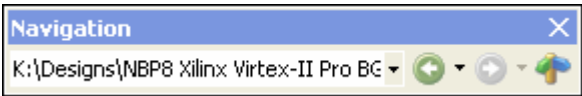



Figure 4. The Navigation Toolbar.

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 area, just as you would within an Internet Browser.

Integrated Navigation Home Page

Click the Go to Home Page button  to access the Integrated Navigation Home Page, a top level page where all navigation support pages can be accessed, as shown in Figure 5.

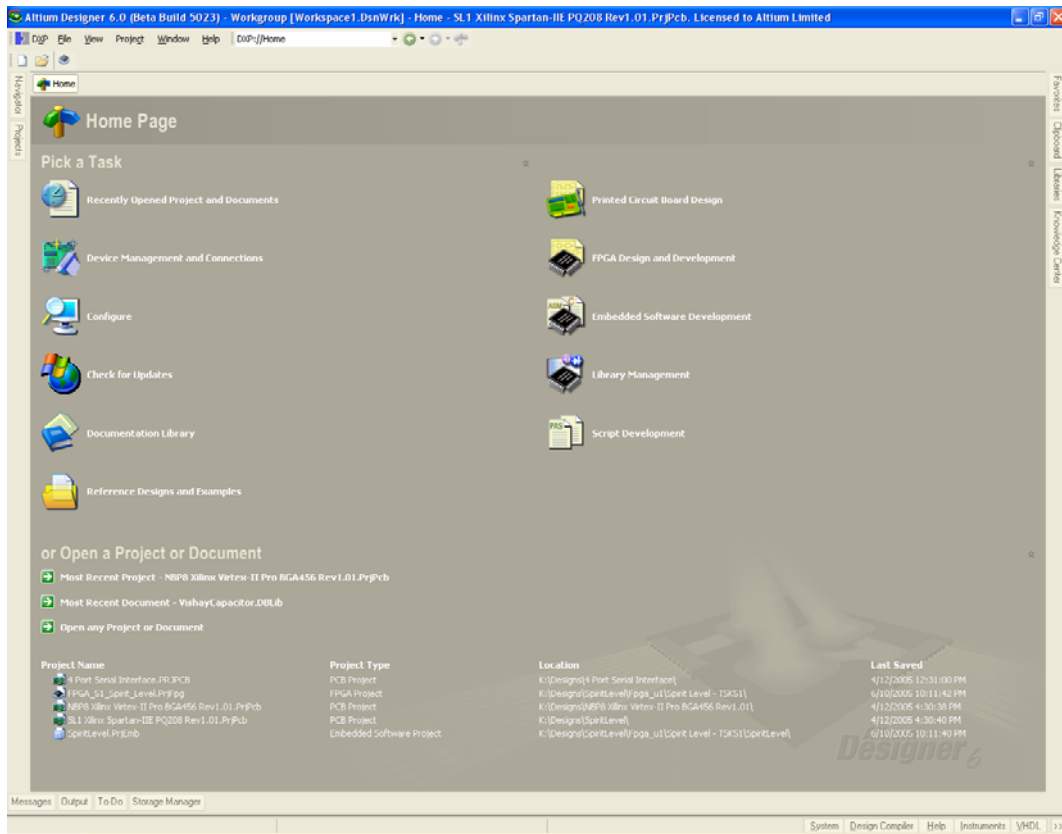


Figure 5. The Navigation Home Page.

Favorites

Like your internet browser, Altium Designer supports the concept of defining Favorites. Once the **Favorites** panel is displayed (click the **System** button on the Status line) you can right-click in the **Favorites** panel to mark the current view of the active document as a favorite. Simply double-click on a favorite 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 (click the **Help** button on the 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 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 is 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 fileset required to manufacture a single printed circuit board, while the schematics (and HDL) in an FPGA project output the fileset required to program a single FPGA. The project file brings together all of the 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.

Back to the 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 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 design files will appear in the **Projects** panel, organized into their respective projects that were 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 6).



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.

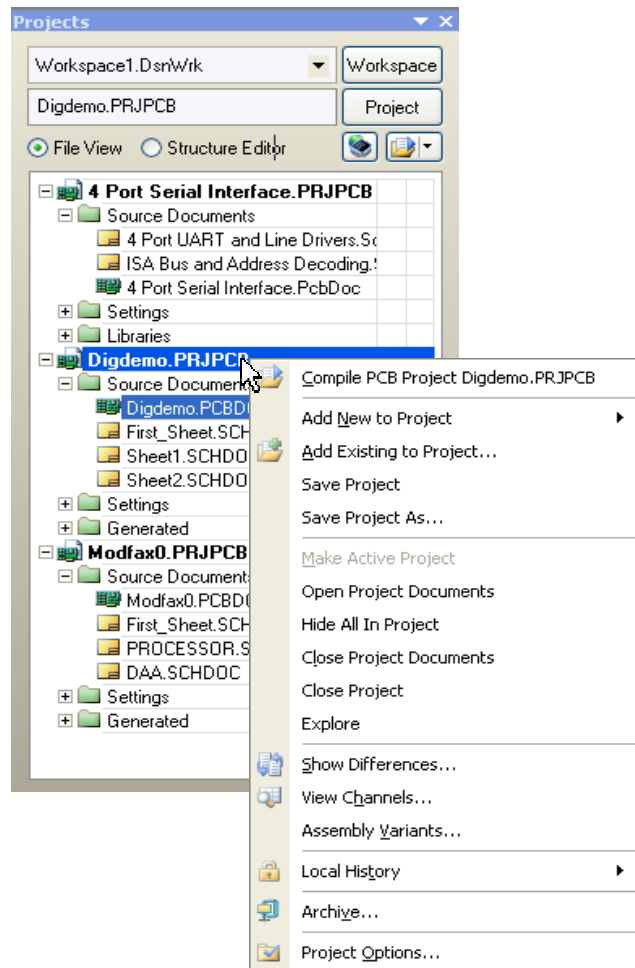


Figure 6. Right-click to show all project-related commands.

Project Hierarchy

For any project involving multi-sheet design, there are basically two choices that need to be made – defining the structural relationship between the schematic sheets (flat or hierarchical), and determining the method of electrical connectivity between the circuitry on those sheets. Display of the project hierarchy in Altium Designer's **Projects** panel is quite similar to OrCAD Capture's Project Manager, with some differences in naming conventions and how hierarchies are graphically represented.

Defining Your Sheet Structure in OrCAD Capture's Project Manager

Like Altium Designer, Capture supports flat and hierarchical designs. Both use a block-like symbol to define sheet-to-sheet structure in a hierarchical design, called a *Sheet Symbol* in Altium Designer, and a *Hierarchical Block* in Capture. In both the symbol references the lower level schematic. In Altium Designer this is simply another schematic sheet, in Capture it can be more complex.

Capture has another layer of design partitioning that affects hierarchy. In Capture there is a *schematic*, which present as a folder icon in Capture's Project Manager, and there are *pages*, which present as a schematic sheet icon. Each Capture schematic can be made up of one or more pages. The Capture hierarchical block points to the schematic below, which means the block can actually reference circuitry divided over multiple pages.

Typically a flat Capture design is one schematic (folder), with the design being drawn on as many pages as required in that schematic (folder). For a hierarchical design, the hierarchical block symbol (or part with an attached schematic sheet or model) is the mechanism used to partition the major functional regions of a design.

For a simple hierarchy, each hierarchical block, or part with an attached schematic folder, or VHDL model, represents a unique design module. The Hierarchy tab in OrCAD Capture's Project Manager displays a simple hierarchical design as a tree of schematic pages. The schematic folder or VHDL entity at the top of a hierarchy, which directly or indirectly refers to all other modules in the design, is called the root module. In the OrCAD Project Manager's File tab, the root module has a backslash on its folder icon (Figure 7). The root module folder, as well as any other module folder, can contain as many schematic pages or VHDL models as required.

Now let's look at Altium Designer.

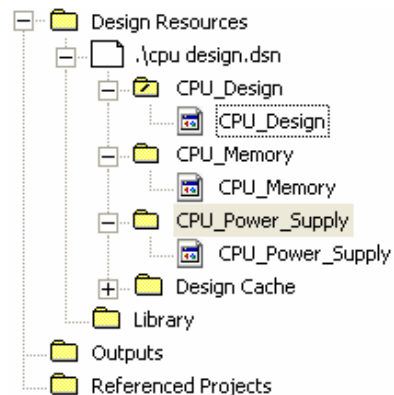


Figure 7. The structure of an example CPU_Design project as it would appear in OrCAD Capture.

Sheet Structure in Altium Designer's Projects Panel

In Altium Designer, hierarchical designs can likewise be viewed and navigated also as a tree structure through the **Projects** panels. Once the project has been compiled at least once, the **Projects** panel will show the hierarchical structure. In a hierarchical design you can think of the first sheet as the parent and those represented by sheet symbols as children (note that child sheets can have their own children too). With that idea in mind, the tree view of the hierarchy makes it easy to navigate and get the overall

picture of your design, the hierarchy of the example CPU Design shown in Figure 7 is shown in Figure 8 as it appears in Altium Designer.

A multi-sheet design project in Altium Designer can also be arranged as a hierarchical structure of logical blocks, where each block can be either a schematic sheet or a HDL file (VHDL or Verilog). At the head, or top, of this tree structure is a single master schematic sheet, more commonly referred to as the project's top or parent sheet.

The structure of the sheets is formed through the use of a special symbol called a sheet symbol. Each of the source documents that make up the design are represented on the parent sheet by a sheet symbol. The Filename property of each sheet symbol references the schematic sub-sheet that it graphically represents. In turn, a schematic sub-sheet can also contain further sheet symbols referencing lower schematic sheets or HDL files. In this way you can define a structural hierarchy of source documents that can be as simple or complex as your needs require.

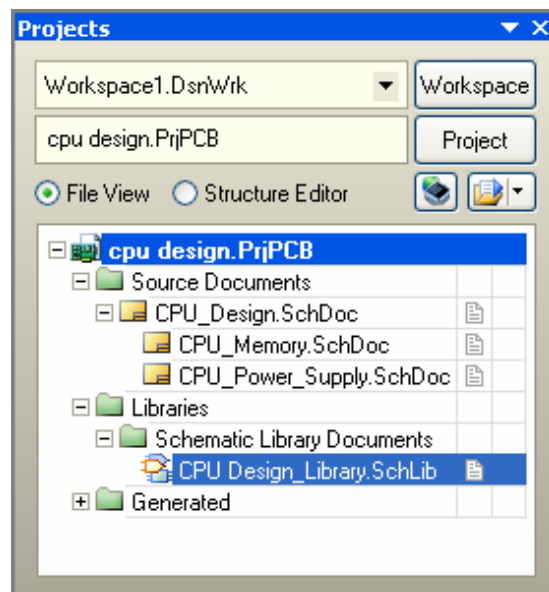


Figure 8. The same project CPU Design is easily viewed after import in the Projects panel of Altium Designer.

Defining Your Net Connectivity – it is Different

In OrCAD Capture, net connectivity is made using net aliases, off-page connectors, hierarchical blocks and hierarchical ports, and globals. Nets between schematic pages within a single schematic folder are connected through the off-page connectors while the hierarchical blocks and ports connect the nets between the schematic folders. Globals are used to connect power/ground nets throughout the design.

Altium Designer uses a similar 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.

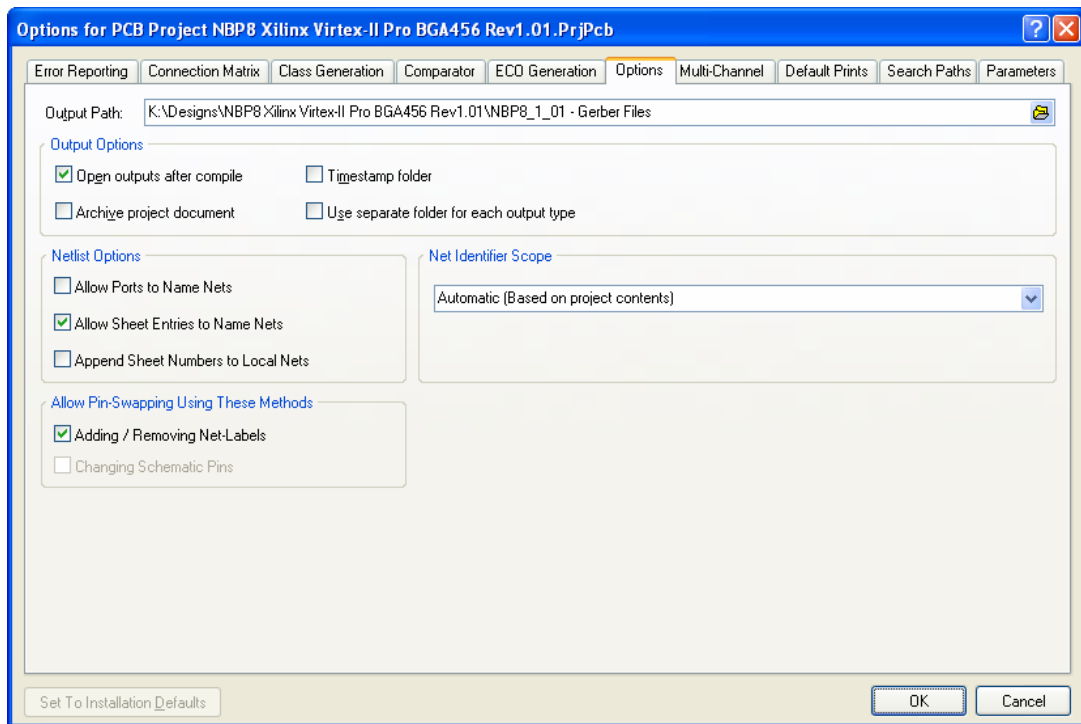


Figure 9. The Net Identifier scope is set in the Options tab of Project Options.

In the **Net Identifier Scope** dropdown you can 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 simply 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 blocks are mapped as sheet symbols, and they will translate to sheet symbols in Altium Designer. 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 subsheets. 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. The Import Wizard also allows OrCAD

users to determine how they want their junctions to import, and also log any errors or warnings that you can check later after importing.



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

Design Synchronization

Design synchronization is fully integrated in Altium Designer without the need for passing a net list. 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.

Complex Hierarchy

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 as well as OrCAD.

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 (discussed below).

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, 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](#).

Parametric Multi-channel Design

Support for multi-channel design – designs where the same section of circuitry is repeated – is an outstanding strength of Altium Designer. The ability to be able to make each channel different by passing parameters to it from the parent sheet symbol is also supported, and is referred to as parametric hierarchy.

Using parametric hierarchy you can parametrically define the component value, supporting the situation where a component does not have the same value in each channel. Parametric components are defined by declaring their value as a parameter of the sheet symbol above, and then referencing that parameter on the target component.



A full tutorial that shows how to create a multi-channel design in the Schematic Editor, including the use of sub-sheets, sheet symbols, and the **Repeat** command may be found in the tutorial, [Creating a Multi-channel Design](#).

Compilation – a Cornerstone of Altium Designer

Compilation is a cornerstone concept of the Altium Designer environment, and a fundamental difference from OrCAD. Compilation is a process that allows you to harness many powerful design features and can be done with your translated OrCAD schematics, or even just a net list. 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 as explicit in your design as it was before in OrCAD, 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 net lists 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. Simply 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 10.

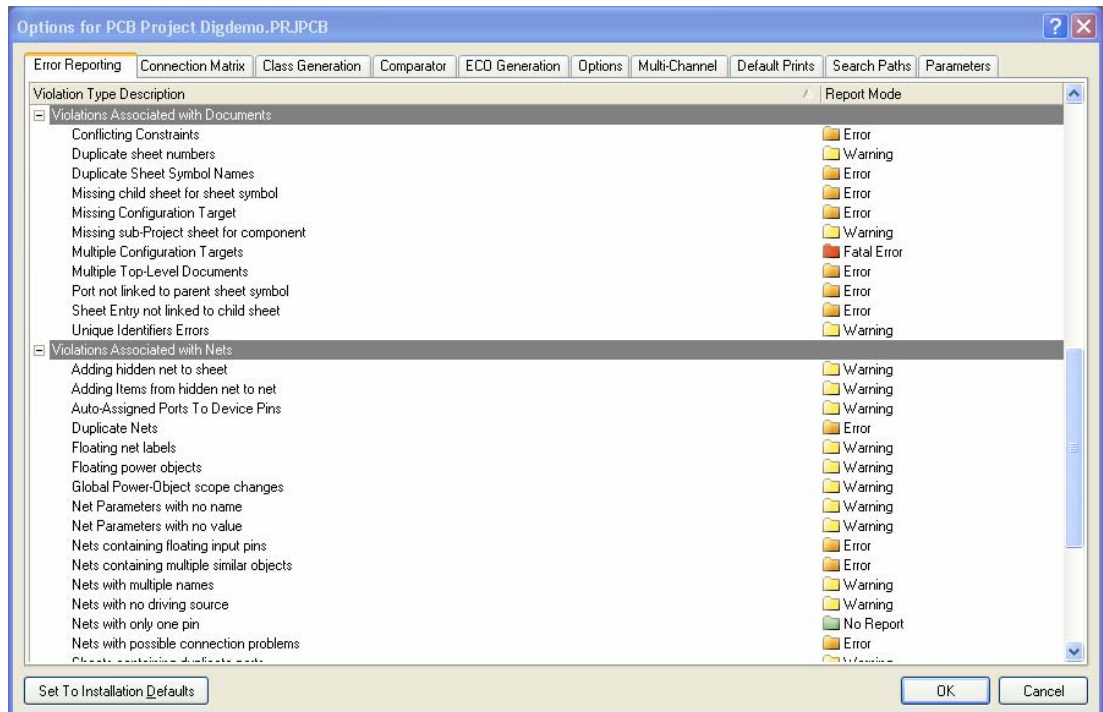


Figure 10. Error Reporting tab in Project Options dialog.



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](#).

The Schematic Symbol Is the Part...

As an expert OrCAD user, you'll know that parts form the basic building blocks of design in OrCAD.

In OrCAD Layout (PCB), a part can represent one or more physical components; or it may represent a function, or simulation model. The part's inherent behavior is described through a simulation model, or other means. Parts in PCB designs usually correspond to physical objects: gates, chips, connectors, objects that come in packages of one of more parts. Multiple-part packages are physical objects that are comprised of one or more parts.

In OrCAD's Capture, a part is a logical entity that is described by symbol graphics, pins and various properties. As parts are placed in a schematic design, Capture maintains the identity of the part for back annotation, net listing, bills of materials, and so forth. At the very minimum, a part requires a part name, a part reference prefix, and a name of a PCB footprint.

These two definitions that use the same term depending on the context of design may initially cause some confusion in the new environment which uses the term *component* instead. But it is not unlike how things work in Altium Designer except that the schematic symbol is effectively the component for all phases of design, and not just the schematic capture portion of it.

Altium Designer Components

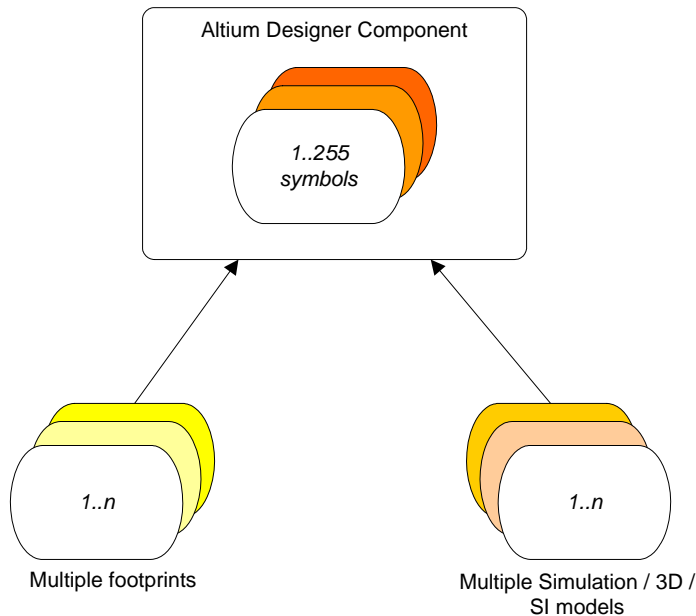



Figure 11. Altium Designer symbols can have multiple footprints and symbol models.

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.

 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](#).

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 concept that will be new for OrCAD users. All libraries may be viewed and managed at any time from the Projects and Library Panels.

Altium Designer Libraries

An integrated library in Altium Designer is one where the source symbol, footprint, and all other information (e.g. SPICE and other model files) are compiled into a single file. During compilation checks are made to see how relationships are defined, to validate the relationship between the models and the symbols and to bundle them into a single integrated library. This file can not be directly edited after compilation, offering portability and security.

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.

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 the OrCAD Capture CIS. 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 OrCAD libraries are automatically grouped into one PCB project.

Like Capture, 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.



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

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, offering the same flexibility you previously had with OrCAD.



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 much 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 easier such as Online DRC, Snap to Center, Selection preferences may be found by invoking the Preferences dialog for PCB Documents. Select **Tools » Preferences** [shortcut: **T, P**] from the main menu command to open the *Preferences* dialog.

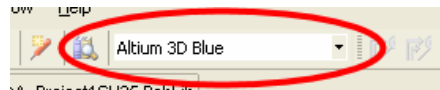
View Configurations (Layers and Colors)

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.



It's worth mentioning that you can easily 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 includes 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.

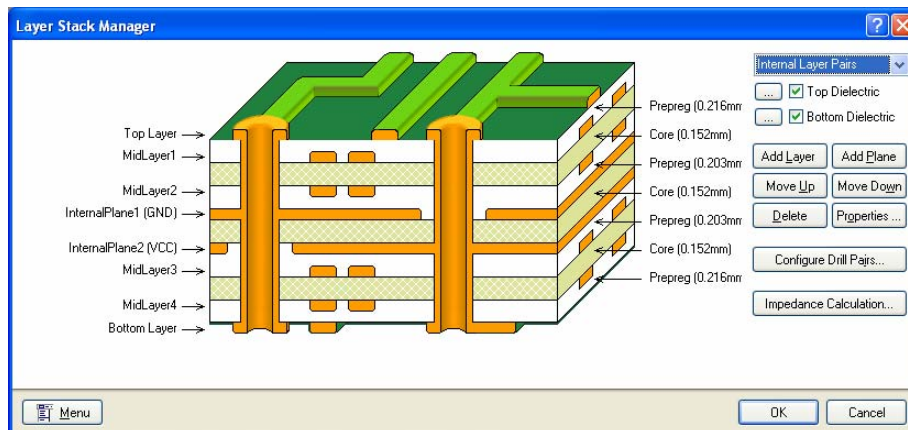



Figure 12. The *Layer Stack Manager* dialog shows a cross-section of the board as you design. Layers may be added or redefined in this dialog.

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

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 tracks, 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 invoke the *PCB Rules and Constraint Editor* dialog as shown below in Figure 13.

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 OrCAD 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 13. 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.

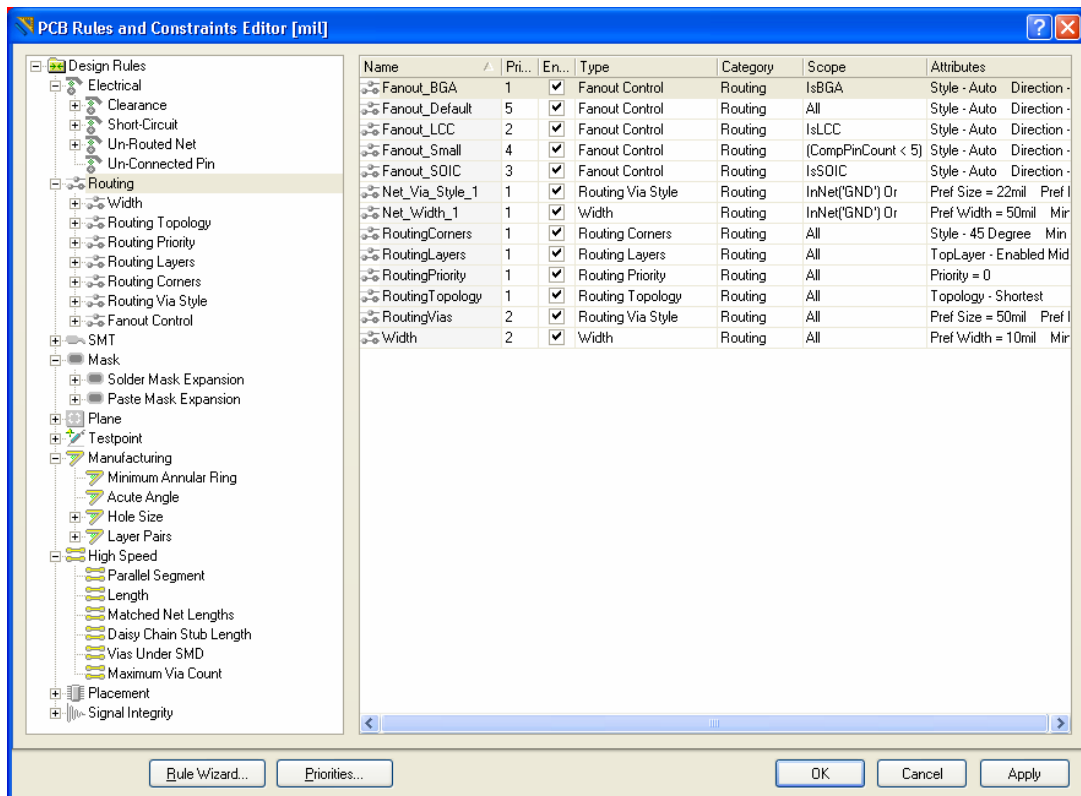



Figure 13. The PCB Rules and Constraint Editor where all design rules can be managed.

Interactive Route

As with OrCAD, 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 Board Layers and Colors dialog. Click on the **Show** checkbox to enable a layer. Once you have enabled your signal layer, the tab for it will display in the main design window. Click on the Layer tab at the bottom of the workspace to make it the current or active layer, ready to route on.

Interactive Routing in Altium Designer can be invoked through either the **Place** command in the main menu, selecting the **Interactive Routing** command, or selecting the  button in the **Wiring** toolbar.

The following tips will assist you to get a quick start for placing traces:

- Left-click or **ENTER** key – Places a start or end vertex in the trace. Placed trace segments will show 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 Extents).
- **PAGE UP**, **PAGE DOWN** – 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.
- Right-click or **ESC** key – 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 line.

Situs Autorouter

The Situs topological autorouting system is fully integrated into Altium Designer. Once your active PCB document is completed and everything positioned, you're ready to start.

Select **Auto Route » All**, from the Situs Routing Strategies to select a suitable routing strategy, and then click on **Route All** to start autorouting. The **Messages** panel displays the routing progress.



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

Signal Integrity

Signal Integrity is also integrated into Altium Designer. Because of this, it requires that you have at the very least, a project file that contains at least one SCH source document. Signal Integrity is run from the main menu 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 SCH and PCB Editors, and configuring your tests further can be found in the tutorial [Performing Signal Integrity Analyses](#).

So how do I get my board manufactured? Setting Up Project Outputs

The setting up of all project outputs is consolidated through a single interface – the **OutputJobs Editor**. Because the output settings are stored in a document (*.OutJob) it offers the convenience of being portable between multiple and different projects.

OutputJobs Editor

The OutputJobs 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 project, for example 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** (as shown in Figure 14) 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.

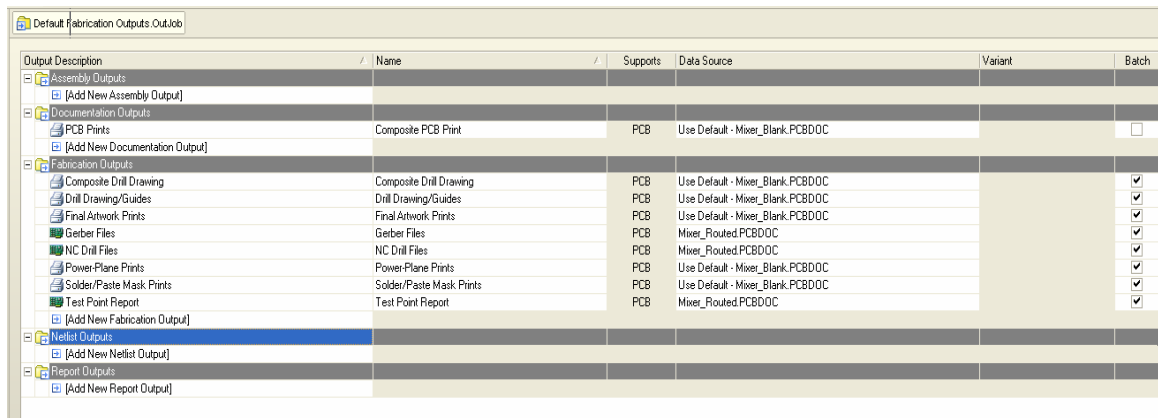


Figure 14. Fabrication output job file for the Multi-Channel Mixer project.

The Output Job file is divided into a number of 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 the output options that you require.



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 OutputJob.

Smart PDF

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 pin 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.

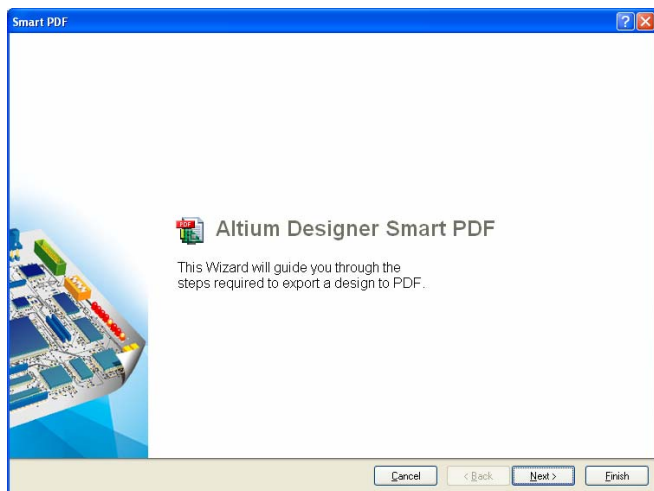


Figure 15. Use Smart PDF to generate live, bookmarked PDFs of your designs, ideal for design reviews and product documentation.

Viewer Edition License




A viewer edition license is also available to replace the OrCAD 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 through 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, simply 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](#).
-  For a tutorial that steps you through all the basics of creating components, read [Creating Library Components](#).



For a tutorial that steps you through all the basics of editing multiple objects, read [Editing Multiple Objects](#).



For an overview of Altium Designer's FPGA design, development and debugging capabilities refer to the [FPGA Designers QuickStart Guide](#).

Revision History

| Date | Version No. | Revision |
|-------------|-------------|---|
| 2-Feb-2006 | 2.0 | New document release |
| 21-Feb-2006 | 2.1 | Content reviewed and updated |
| 2-Jul-2007 | 2.2 | Content reviewed and updated |
| 9-Nov-2007 | 2.3 | Content reviewed and update for 6.8 |
| 7-Jan-2008 | 2.4 | Updated view configurations info. |
| 11-Feb-2008 | 2.5 | Component body references changed to 3D body. |

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.