Project Management

Contents

What is an Altium Designer Project?

The starting point for every design created in Altium Designer is a project. 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 include 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.

To learn about the different types of projects in Altium Designer, see Types of Projects in Altium Designer

The Role of the Project File (Project Options)

The set of design documents that make up the project are bought together by the project file. The project file stores all project-relevant settings, including a link to each document in the project, and all project-relevant options. Each document in the project is stored as a separate file, which is linked to the project via a relative reference for files on the same logical drive, or an absolute reference for files on a different logical drive. Outputs generated from the project are also referenced in the project file.

The exact set of Project Options stored will depend on the project type.

See Project options for more information

The Projects Panel

The panel that is perhaps the most often used in Altium Designer is the Projects panel. The Projects panel is your view into the project.

To learn more about the Projects Panel including icons and display options, see Projects Panel

Project Insight is a navigation tool available from the Projects Panel. This tool enables you to preview all of the

documents in your project. To learn more about this tool, see [Project Insight](#)

Creating a new Project

You can create a new project in more than one way.

- From the **File** menu, drill down to create a new project.
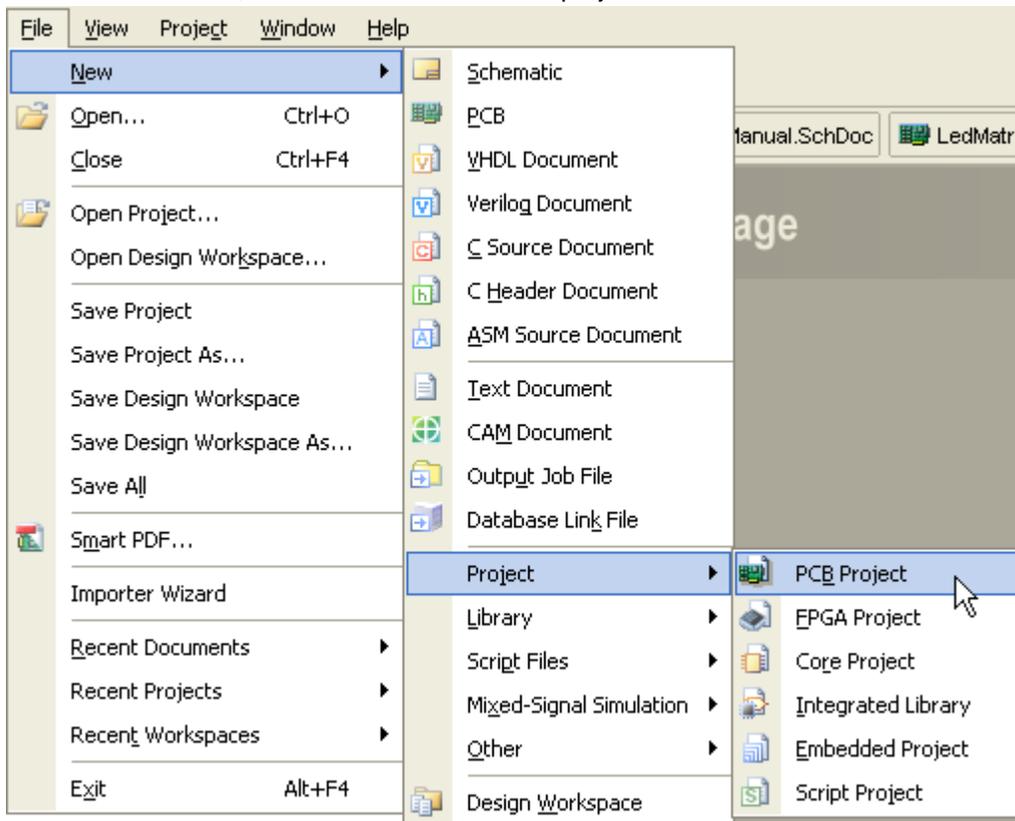

Figure 1. Drill down from the File menu to create a new project.

- Click on the required entry in the **New** section of the Files panel. If this panel is not open, click on the System button at the bottom right of the main application window and choose Files from the pop-up menu that appears.
Note that the project file only exists in memory when first created, use the Save or Save As commands to save it with a suitable name in the required location.
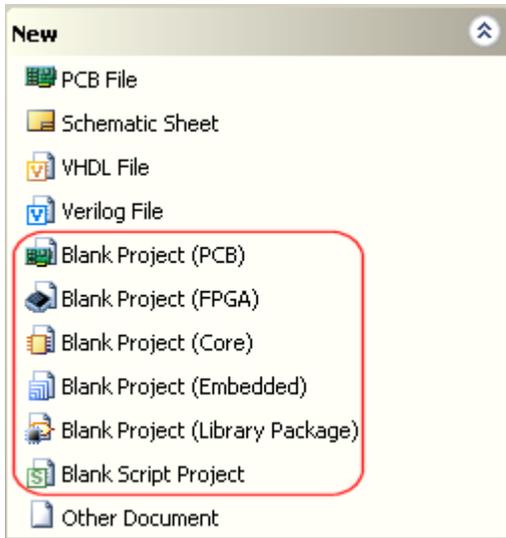
Figure 2. Creating a New Project from the Files Panel

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

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

Note: FPGA, Core and Embedded project filenames should not include any spaces.

Adding and Removing Documents from a Project

Once you have created the project and saved it to the required location, you are ready to start adding the design documents. The easiest way to add new or existing design documents to a project is to right-click on the project name in the Projects panel and use the **Add New to Project** or **Add Existing to Project** menu options.
You can also add other project-relevant documents to your project, such as Word® files or Adobe® PDFs. Add them in the usual way, you will need set the File Type list in the Choose Document to Add dialog to All Files (*.*) to see the files.
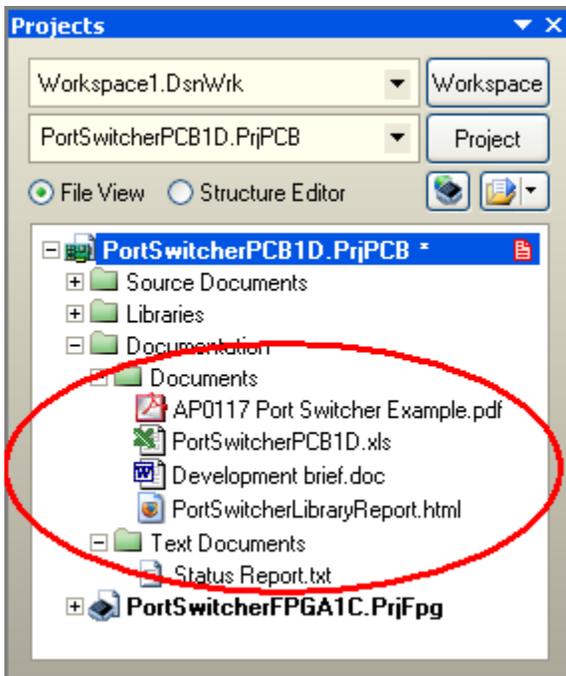
Figure 3. include other types of documents in your project.

Making Changes to the Project

The document that you are currently editing, referred to as the active document, is highlighted in the panel. You will notice in Figure 4 that the project as well as the active document being highlighted, so is the active project. Any change made via a command in the Projects menu will be made to the active project.

When you have multiple projects open for editing, a simple way to perform project related commands on the project of your choice is to right-click on the project name in the Projects panel. When you do, a context sensitive menu will appear, where you can perform project actions regardless of whether the active document belongs to that project or not.

Figure 4 shows that a document in the embedded project is active, but the PCB project has been right-clicked on to display the context sensitive project menu.
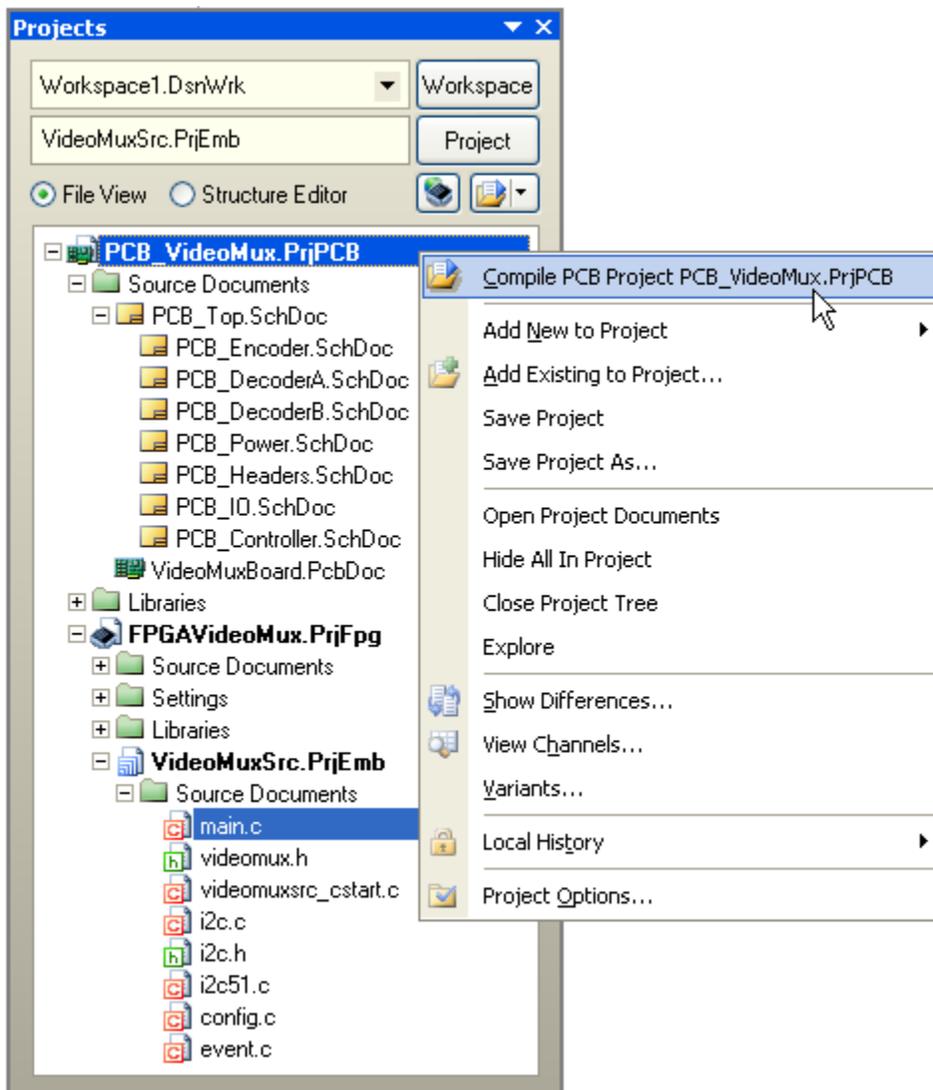
Figure 4. Right-click on the project name to access a set of project-related commands.

Order of Documents in a Project

Documents within a group, such as Source Documents, are displayed by default in the order that they were added to the project (which is the order they are listed in the project file). To change the order of documents within a display folder, click, drag and drop a document to its new location. Once a project has been compiled, the source documents are also displayed showing the hierarchy of the design.
Note that you cannot build project hierarchy by dragging documents in the panel. The parent-child hierarchical relationship between documents in a project is defined by the sheet symbol to sub-sheet definitions.

For more information about project hierarchy, see Connectivity and Multi-Sheet Design

Managing Project Documents

It is important to appreciate that the presentation of documents in the Projects panel does not reflect how they are

stored on the hard drive. One way to think of it is that the Projects panel presents a logical representation of the project. The organization of the document storage is up to you – you might have project documents stored on a common company server, with your own idea or scratch-pad files stored on your own PC. Or you might use Altium Designer with a version control system, checking project documents out from the central repository into your personal workspace on a server.

Sharing Documents between Projects

Because each project document is linked to the project you can share documents between multiple projects. It might be a schematic sheet for a standard power supply you use across different products, or it might be a section of the design that you would like to simulated separately.

Saving Project Documents with a New Name

There are Save As commands that allow you to save a document with a new name. Note that this is not a rename process, after doing this you will have both the old document and the new document on the hard drive. Performing a Save As on a document will also update the project – done to ensure that the document remains a part of the project. If you want to create a back-up copy of a document without affecting the project, use the Save Copy As command.
If a document that belongs to two projects is saved with a new name and both projects are open when you do this, then both projects will be updated to reflect the name change, maintaining the integrity of both of the projects.
Note that you cannot relocate a project by performing a Save As action on the Project document itself and saving it to a new location – this will only save the project document to that location. It will also update all the links to the documents that are in the project, linking from the new project document to the old location where the actual project documents are stored! Again, the Projects panel is not for file management, in this case you must relocate the project and its documents via the operating system.

Project File Management - The Storage Manager

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

To learn more, see Storage Manager Panel

See Also

Working with Documents

Document Management - Auto save, Local History and External Version Control

Design Workspace

Managing Design Variation with Variants

Understanding Version Control, an Introduction