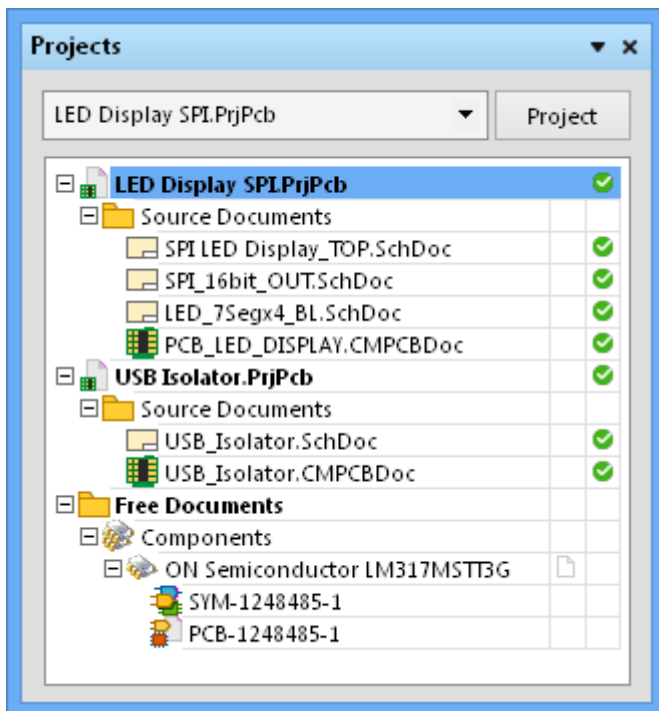


## Projects

Modified by Rob Evans on Jul 28, 2015

Parent page: [Workspace Manager Panels](#)



Two PCB projects and a Component open in the *Projects* Panel, with one of the PCB projects selected. Component projects are opened as temporary free documents..

## Summary

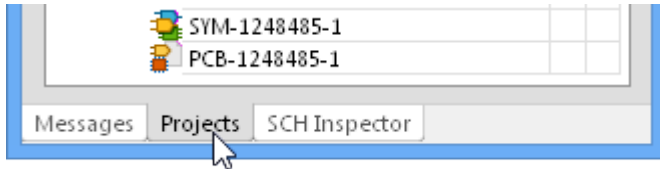
The *Projects* panel displays all projects that are currently open along with their constituent documents and the file Version Control status. Any open documents that have not been created as part of a project or added to an existing open project will also be listed.

## Panel access

To open the *Projects* panel, click the **Projects** button in the **System** group under the **View** tab: **View**

| **System** | 

Panels can be configured to be floating in the editor space or docked to sides of the screen. If the *Projects* panel is currently in the group of docked *Workspace* panels on the left, use the **Projects** tab located at the bottom of the panels to bring it to the front.



## Project documents tree

As you open an existing project or create a new one its entry will appear in the *Projects* panel. Any existing documents that are part of a project and any new ones that have been added will appear under sub-folders, according to their purpose and/or type. For example, the following common folders and content document types can appear under a project:

- **Source documents** – Core design documents such as schematics, PCBs, etc.
- **Generated** - Documents generated as output. Documents are further sub-divided based on type (e.g. BOM documents, text documents, etc). As project output is generated, the corresponding sub-folders in the main tree will be created and become populated.

Any documents that are independent of a project will appear as Free Documents and will exist under corresponding sub-folders. Component documents, for example, are gathered under a Components group and are regarded as free (non project) documents due to their temporary nature.


As well as allowing multiple documents to be open for editing, the environment also supports multiple projects being open at the same time. These could be unrelated projects, or they could be related projects.

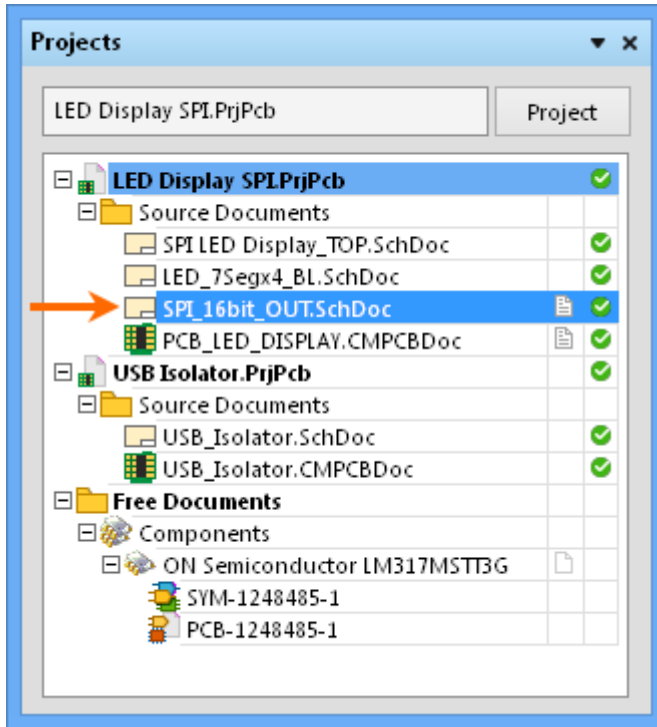


The keyboard shortcuts **Up Arrow**, **HOME**, **END** and **Down Arrow**, can be used to display the previous, first, last and next entry in the panel, respectively. Use the **Right Arrow** and **Left Arrow** keys to expand and collapse a top-level entry or sub-folder therein, respectively.

## Active vs Focused

In the *Project* panel's main tree, the active project is highlighted using the Windows Active Title Bar color for your system. When there are no documents open in the design editor window, a project is made active by selecting it from the list of the projects currently loaded in the panel. As soon as a project document is opened however, the parent project of that document automatically becomes the active project.

The active document in the design editor window will also be highlighted in the panel and accompanied by the file open icon .



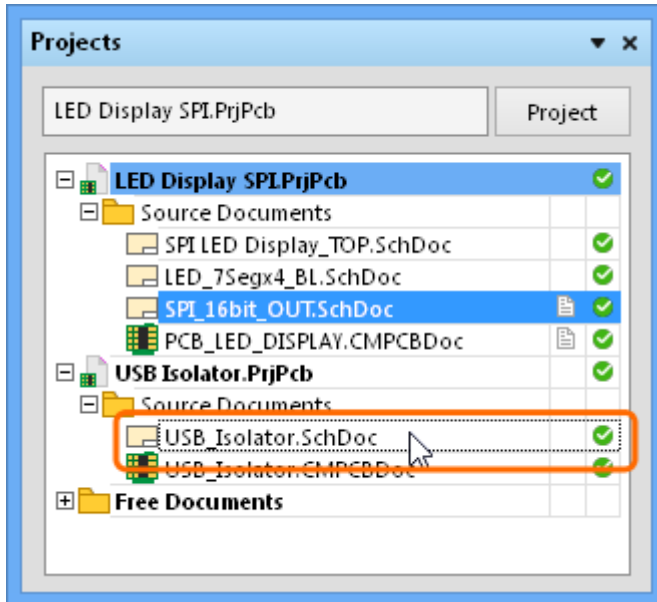
An opened active document reverse highlighted with the system color (blue), which is under the highlighted active parent project.

With no project documents open in the design editor window, opening an additional project will automatically make it the active project in the panel.

With numerous documents open in the design editor window, changing the active document using editor's document tabs will cause the *Projects* panel to update accordingly, to reflect both the active document and the active project (if the document you make active does not reside in the same project as the previously active document). Conversely, clicking on the entry for a document that is already open (and which belongs to a non-active project) will make the parent project of that document the active project.

Only one project and document may be active at any given time, however the panel itself allows you to focus and perform actions on any project or document. Right-clicking on the entry for a non-active project or a non-active document will bring up an associated menu with commands targeting the focused project or focused document, respectively – see [Right-click menus](#) below.

In the case of focusing a document, the document will only become focused if it is closed or hidden, otherwise it will become the active document and its parent project will become the active project. For example, in the image below the active project is LED\_DISPLAY\_SPI.PrjPcb and the active document is SPI\_16bit\_OUT.SchDoc. The focused document is USB\_Isolator.SchDoc (distinguished in the panel by a dotted outline box) from the inactive USB\_Isolator.PrjPcb project.



An opened active document (blue) and a selected focused document, indicated by the dotted outline.



Documents can be hidden in order to prevent clutter in the tabbed area of the design editor window. When a document is hidden, it is still open from the point of view of processes such as compilation/synchronization/annotation, It is just not displayed as a tabbed-document in the design editor window.

A project document can be removed from a project. If it is currently open in the design editor window it will be removed from its parent project but remain open in the panel as a free document.





## Document display icons

Document entries in the *Projects* panel are accompanied by document icons that indicate its open/modified status and Version Control state with respect to the Altium Community Vault. This provides a quick visual summary of which documents are modified, saved or hidden and their relationship to their respective documents in the Vault - are modified, to be added, are in conflict, etc.

The document icons, presented in two columns within the panel, are listed in the two sections below.

### Open/Modified status icons

[blank] The document is closed.

-  Open - The document is open as a tabbed document in the design editor window.
-  Hidden - The document is hidden (open, but not displayed in the workspace).
-  Open/Modified - The document is open and has been modified (yet to be saved).
-  Modified (project) - This icon appears next to the main project document to show that its structure has been modified (yet to be saved).











A modified document or project that has yet to be saved is also distinguished by an asterisk next to its entry in the panel. Modified documents are also indicated by an asterisk inside their tab in the design editor window.

## Version Control status icons

The current version control status of each file is displayed next to the file's name in the *Projects* panel. The version control status icons are as follows:


-  No modification – Your local file copy matches the file retrieved (opened/checked-out) from the Altium Community Vault, and is up to date.
-  Out of date – The file you have open no longer matches the version in the Community Vault, which has changed.
-  Scheduled for Addition – A new document has been locally created and saved, and will be added to the Community Vault when the project is committed.
-  Modified – The retrieved file has been modified and saved locally, and will be updated in (checked in to) the Vault when the project is committed.
-  Modified and locked by someone else (a conflict) – The same file has been committed to the Vault by someone else before your attempt to commit *your* modified version of this file.
-  Scheduled for Deletion – The file has been locally deleted from the project, and will be removed from the Vault during the project commit process.  
\* Only appears in the *Commit* dialog.

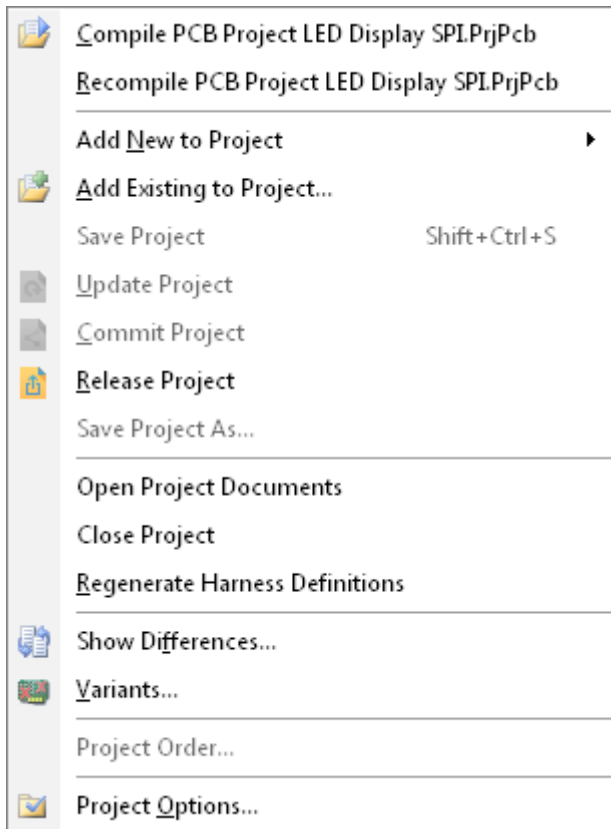
Version Control status icons are not used for Components or their constituent Model documents (Symbol, Footprint etc).

## Right-click menus

### Project file

Right-clicking on a Project file in the *Projects* panel provides access to additional options and commands via the pop-up context menu.

The active state of the commands (active or disabled) will depend on the selected Project's modified and version control status. Note that most commands are also available under **Project** in the main ribbon menu – **Home** | .



Typical right-click context menu for a project.

The right-click context menu commands for a project are:

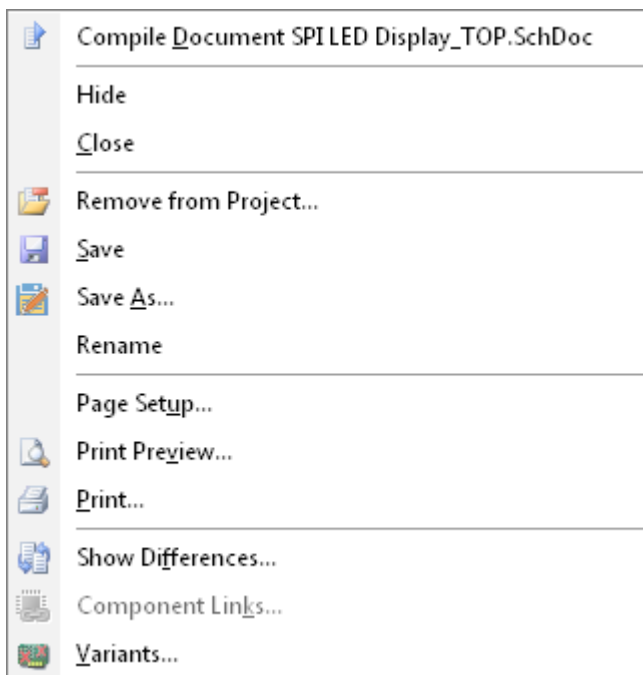
- **Compile PCB Project** – The compiling process detects electrical and drafting violations, and is integral to producing a valid netlist for a project.
- **Recompile PCB Project** – The recompiling process forces a refresh of all linking and net information during the compile process.
- **Add New to Project** – Add a new, blank Schematic, PCB or Text document to the current project.
- **Add Existing to Project** – Add an existing, locally stored Schematic or PCB document to the current project. Other file types (text etc) are also supported.
- **Save Project** – Save the current project document to the local design cache. Locally saved documents (if available) are retrieved from the cache when the project is reopened.
- **Update Project** – Enabled when the local version of the project does not match that in the Community Vault. Provides the choice of retrieving the Vault version of the project or retaining the local version. See [Project Management – Collaboration](#) for more information.
- **Commit Project** – Upload (commit) the complete local project to the Altium Community Vault.
- **Release Project** – Process an already committed project by storing a snapshot that contains the full set of documents plus a set of nominated output documentation (BOM, PDFs, Gerbers etc). See [Project Management – Release](#) for more information.
- **Save Project As** – Re-saves the current project document under the nominated name. Enabled when the project has been edited.
- **Open Project Documents** – Opens all documents contained in the current projects in their respective editors.
- **Close Project** – Closes the project and any active project documents. Prompts to save any documents that have been modified.
- **Regenerate Harness Definitions** – If a project contains a harness connector and entries, its harness definition file will be regenerated so the project synchronization is up to date.
- **Show Differences** – Detect and resolve the inconsistencies in the design structure, or the

difference between two design files. See the [Differences](#) page for more information.

- **Variants** - Opens the *Variants Manager* dialog, which allows variations (different physical component assemblies) to be made of the base design. See the [Variants](#) page for more information.
- **Project Order** - Sort projects and their documents by project order, which is determined in accordance with the positional numbering assigned to each document.
- **Project Options** - Opens the *Options* dialog for this project. Set configurations for error and differences warnings, ECO generation, file paths, etc that override the installation defaults.

## Document file

Right-clicking on a document file in the *Projects* panel provides access to a pop-up context menu offering a range of document-specific options and commands.





Typical right-click context menu for a document.

The right-click context menu commands for a document are:

- **Compile Document** - Check the integrity of a Schematic document, with results displayed in the Messages panel; **View | System | Messages**.
- **Open** - Open the selected document in its design editor.
- **Hide/Close** - A hidden document is active, but not displayed in the design editor window. A closed document will no longer be open in the design editor.
- **Remove from Project** - Remove the document from the parent project.
- **Save** - Save the document locally.
- **Save As** - Save the selected document under the nominated name.
- **Rename** - Rename the selected document under a different name. A modified document must be saved before it can be renamed.
- **Page Setup/Print Preview/Print** - Manage document printing, replicating the functions available from the main Print options; **Outputs | Documentation | Print**.
- **Show Differences** - Detect and resolve the inconsistencies in the design structure, or the difference between two design files. See the [Differences](#) page for more information.
- **Component Links** - (PCB document). Opens the Edit Component Links dialog, where the existing ID-based links between the Schematic and PCB documents can be checked and manually edited. See the [Edit Component Links](#) page for more information.

- **Variants** - Opens the *Variants Manager* dialog, which allows variations (different physical component assemblies) to be made of the base design. See the [Variants](#) page for more information.

The only way to close a Component document or its constituent model documents (Symbol, Footprint etc) is to **Commit** the component to the Community Vault, or **Discard Changes** that have been made to the component - **Home** |  and **Home** |  respectively, when the Component document is active.

---

**Source URL:** [http://documentation.circuitmaker.com/display/CMAK/WorkspaceManager\\_Pnl-Projects\(\(Projects\)\)\\_CM](http://documentation.circuitmaker.com/display/CMAK/WorkspaceManager_Pnl-Projects((Projects))_CM)