Using Altium Documentation
Modified by Rob Evans on Oct 31, 2018

Parent page: Vault Items

Vaults provide a flexible and secure method of centralizing the storage and management of all types of design data used in Altium Designer. From the schematic model to the component, from managed sheets through to completed PCB designs, an Altium Vault delivers an ideal method of storing and managing your electronic design data.

When the vault concept was initially developed, the idea was that the vault would act as a repository for *released* data - that is, for electronic design content that has been developed, checked and released for use. The thinking was, designers already have systems for storing and managing their source, such as Subversion - what they need now is a simple way of storing and managing the released content, that allows that content to be easily managed and used from within Altium Designer.

Once designers started to use vaults to store and serve their data, such as their completed models and components, it was not long before they started saying, "*hey these vaults are great for storing my released components, how about letting me use the vault to store the source data too! Especially since I have already released a copy of the component source - the schematic and PCB models - into my vault*".

This ability is delivered through the Altium Vault's support for *Direct Editing*. Just what does that mean, 'Direct Editing'? It means you can right-click on a model, or a component, or even a managed sheet in the vault, and select **Edit**. Sure, a file opens in Altium Designer ready for you to edit - opened in a temporary editor - but when you are finished you simply save, then release it back to the vault. As for the file you just edited, well you don't have to worry about it, it is no longer needed since your edits are securely stored back in the vault, so it is destroyed, and the temporary editor instance is closed.

Direct Editing - Concisely Summarized

Direct editing frees you from the shackles of separate version-controlled source data. You can simply edit a supported Item type using a temporary editor loaded with the latest source direct from the vault itself. And once editing is complete, the entity is released (or re-released) into a subsequent planned revision of its parent Item, and the temporary editor closed. There are no files on your hard drive, no questioning whether you are working with the correct or latest source, and no having to maintain separate version control software. The Altium Vault handles it all, with the same great integrity you've come to expect, and in a manner that greatly expedites changes to your data.

Supported Item Types

The following Item types in an Altium Vault are supported with respect to Direct Editing:

- [Component](#)
- [Footprint](#)
- [Symbol](#)
- [Simulation Model](#)
- [Managed Schematic Sheet](#)
- [Layerstack](#)
- [Schematic Template](#)
- [Project Template](#)
- [Component Template](#)
- [Outputjob](#)
- [Draftsman Document Template](#)
- [Draftsman Sheet Template](#)

Editing the Initial Revision of an Item

From the [Vaults panel](#), when you create one of the supported Item types, you have the option to edit and release the associated data entity into the initial revision of that item, after creation. So instead of creating the source data outside of the vault and releasing it, you can start working on the source directly from the moment the empty Item is created, courtesy of direct editing. And when that editing is complete, the entity is released into the initial planned revision of its parent Item, and the temporary editor closed.

To take advantage of this feature, simply enable the **Open for editing after creation** option, found at the bottom of the [Create Item dialog](#). Pre-empting the usefulness of this functionality, the option is enabled by default. The Item will be created and the relevant temporary editor will open, presenting the relevant document as the active document in the main design window. This document will be named according to the Item-Revision, in the format:
`<Item><Revision>.<DocumentExtension>`.

The image below shows this in action for the creation of a new Symbol Item (SYM-001-0003). A temporary Schematic Library Editor is opened, presenting a Schematic Library document (`SYM-001-0003-1.SchLib`) as the active document in the main design window. Use the document to create the entity as required.
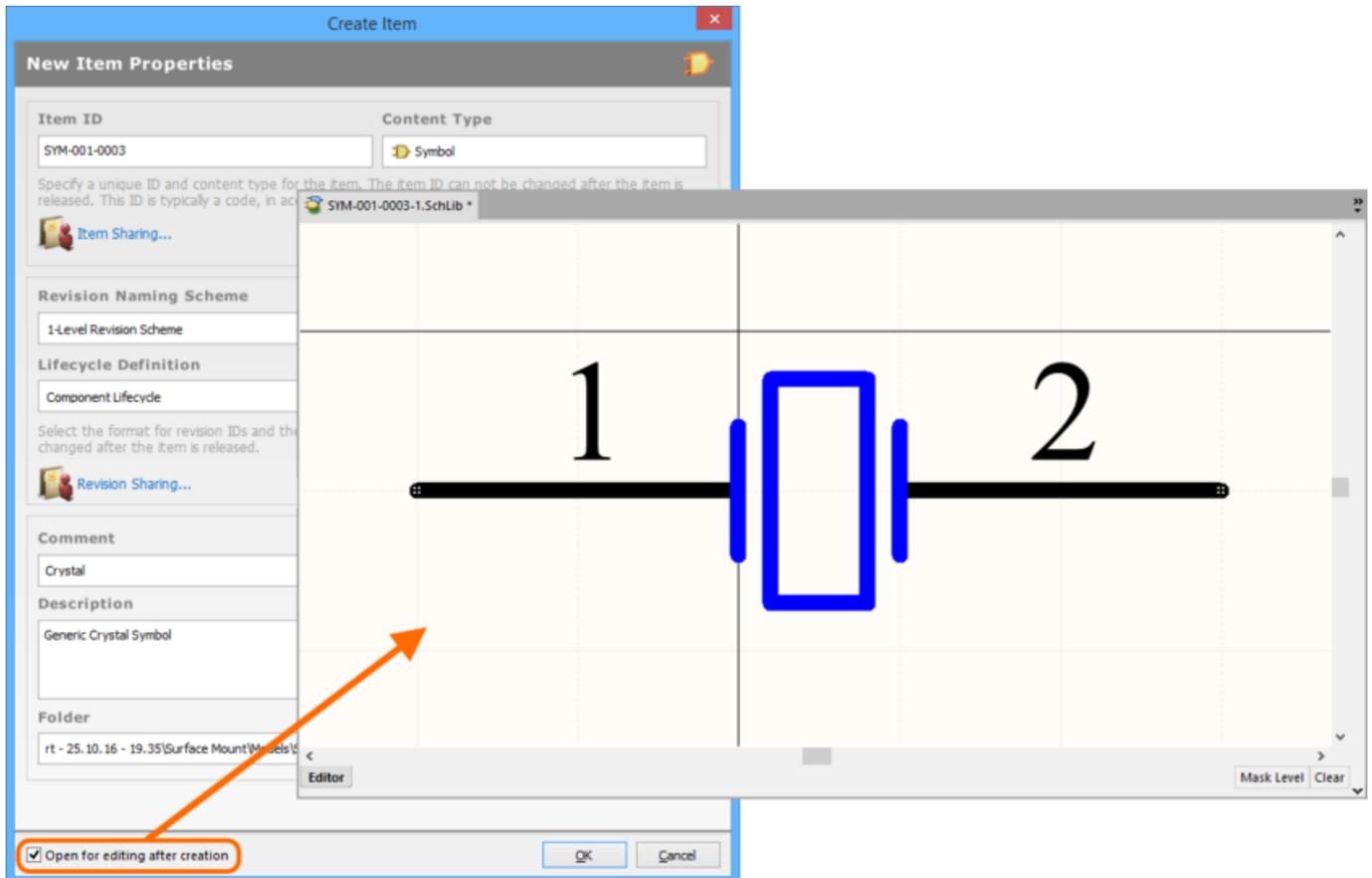
Example of editing the initial revision of a Symbol Item, directly from the vault - the temporary Schematic Library Editor provides the document with which to define your schematic
symbol.

Editing an Existing Revision of an Item

At any stage, you can come back to any revision of a supported Item type in the vault (through the [Vaults panel](#)) and edit it directly. Simply right-click on the revision and choose the **Edit** command from the context menu. Once again, the temporary editor will open, with the document contained in the revision opened for editing. Make changes as required, then commit the release of the document into the next revision of the item.

✅ Right-clicking on the top-level entry for an Item itself, will edit the latest revision of that Item.
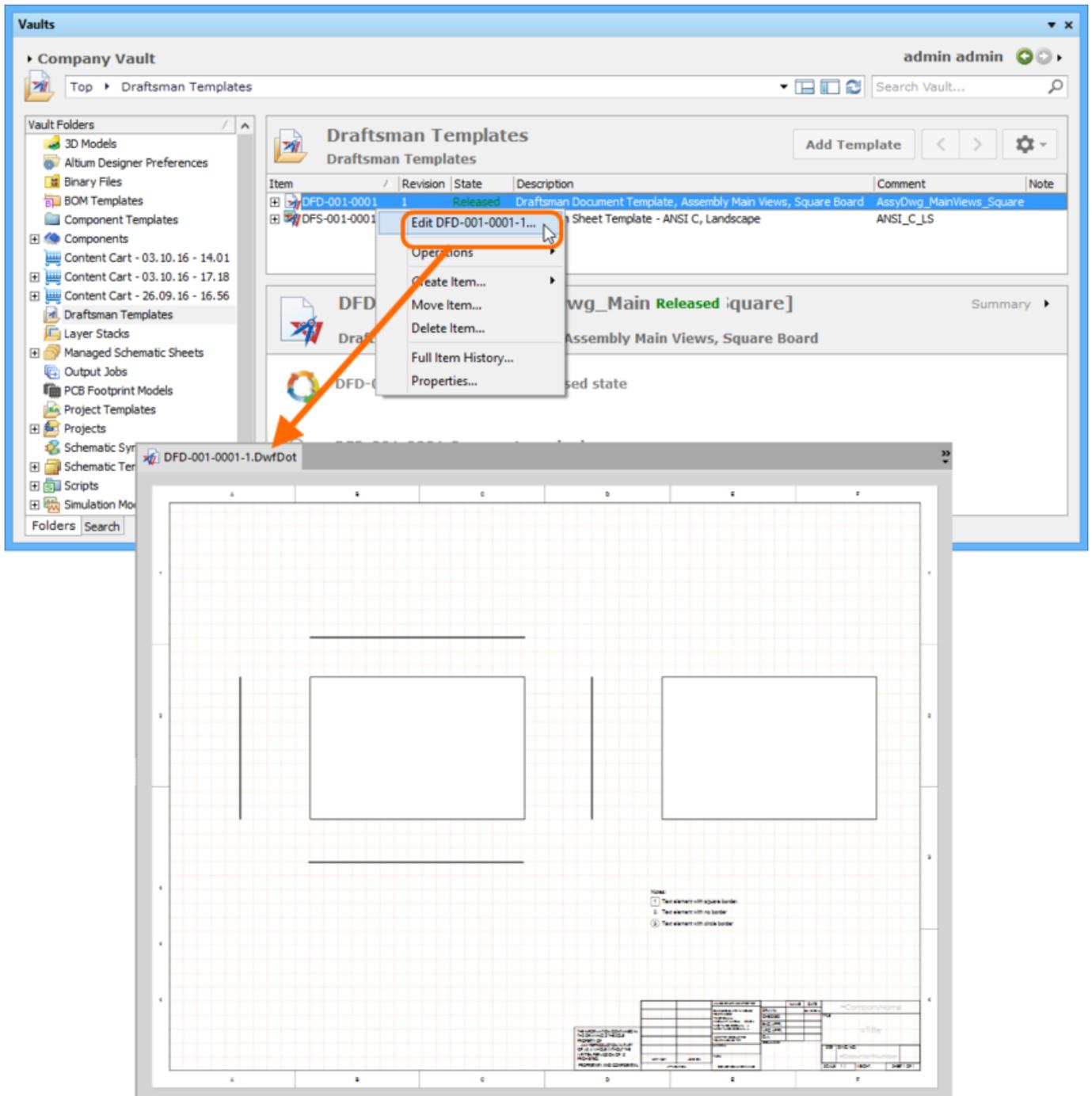
Example of accessing the command to launch direct editing of an existing revision of a Draftsman Document Template Item.

Direct Editing Controls

There are three relevant controls when direct editing, readily available from the **Standard** toolbar for the active temporary:

- 🖫 - *Save Active Document*. Use this button to save the changes made to the document. This is required before you can release the document back to the vault.
- 🗹 - *Release Document*. Use this button to release the defined document to the vault, storing it within the initial (planned) revision of the target Item. The [Create Revision dialog](#) will appear, in which you can change Comment, Description, and add release notes as required. The document and editor will close after the release.
- ✖ - *Cancel Editing*. Use this button if you wish to cancel editing. The document and editor will close, and nothing will be released to the target Item.

These controls are also available as commands - **Save** (Shortcut: **Ctrl+S**), **Release to Vault** (Shortcut: **Alt+Ctrl+S**), and **Cancel Edit** - from the main **File** menu.

> For a revision of a Project Template Item, commands to **Save Project**, **Release to Vault**, and **Cancel Edit**, can only be accessed from the *Projects* panel. Right-click over the entry for the Item Revision to access the relevant context menu.

The Behavior of the Temporary Files

To be able to edit vault data, something must be created for you to edit in Altium Designer. What happens is that temporary files are created. For example when you right-click on a Component Item and select **Edit**, a temporary CmpLib file is created and opened in Altium Designer. When you finish and perform a **Release to Vault**, this file is destroyed as soon as the vault data is updated.

However, if you do not release, but instead **Save** and **Close** the file, it will remain in the [Projects panel](#). This is done so that you do not have to complete the updating process in a single editing session - you can save your work and close the file if you wish, Altium Designer understands that this is a special file so it retains it in the `Altium Vault` folder in the *Projects* panel (☑ Altium Vault), so that you can re-open it when you are ready, and complete the updating and re-releasing process.

> Note that if you exit Altium Designer, all temporary files that were not currently open will be discarded.
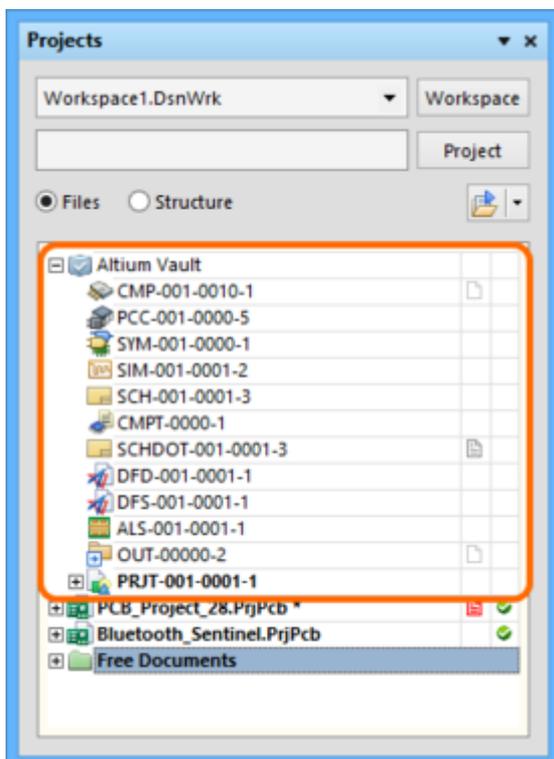
The temporary documents remain accessible, so that you do not have to complete your changes in a single editing session.

> If you change your mind at some point and no longer want to perform the edits, simply right-click on an entry and choose the **Cancel Edit** command from the context menu, to close the temporary file.