Altium Vault Evaluation Guide

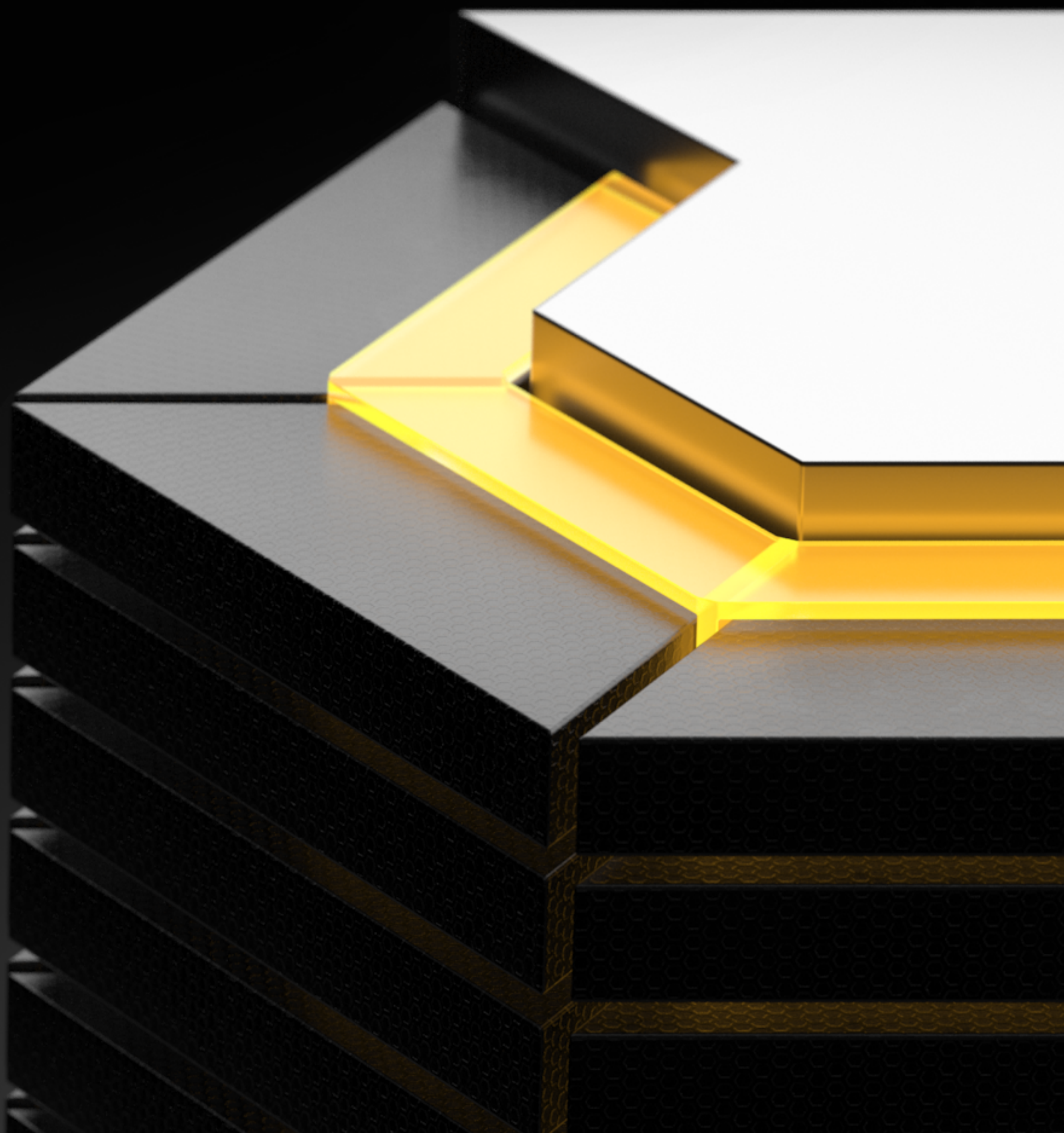


TABLE OF CONTENTS

INTRO TO ALTIUM VAULT

Data management has always been a tedious part of the design process, but ignoring data management leads to wasted efforts, disorganization, and faulty designs. As engineers, we tend to focus on the constructive aspect of design. We want to create the best products and we tend to forget about the small details that make great products possible. No one wants to start from scratch on a new design, especially when dealing with components that we have already proven to be successful. We want to deal with new problems and push technology to new places, but without a way to leverage our past experience, what are we supposed to do? What if you could access all your existing component definitions, templates, and supply chain information in one place?

Altium Vault® provides a simple system to streamline data management so that you can benefit from your previous efforts by building a bridge between your existing and future designs. A centralized maintenance system for all your ECAD data allows everyone to access the information they need without having to worry about local copies and using the wrong revisions. Your workflow remains the same with the added benefit of built in validation and organization systems, so you don't have to spend time doing the work manually. Altium Vault provides an automated, disciplined data management system to automate the design and release processes while making ECAD data easy to access.

FOUR STEPS TO UNDERSTANDING

This document will guide you through a successful evaluation of Altium Vault. It will introduce you to an example dataset so you can learn more about how Altium Vault can work for you. You will be provided with an evaluation instance of Altium Vault, which will be populated with default data structures and recommended configurations guiding you through the four core principles of use:

FIND	APPLY	RELEASE	CORRECT

Disclaimer: By default, Work-In-Progress design sources are committed to the Vault Server using SVN protocol. Although the login and session with the Vault database on port 9785 is secured over HTTPS, the SVN commit actions are not. Therefore, we recommend for the evaluation instance that you do not commit any sensitive intellectual property or designs.

LOGIN FROM BROWSER INTERFACE

Basic **user** and **roles** setup of the **Altium Vault** can performed with the **browser interface**. This will usually be done by the IT department, as they generally maintain user databases and are the best qualified to provide access to specific users as part of the installation. Since the basic setup is done through a web interface, you only need access to **Altium Vault**.

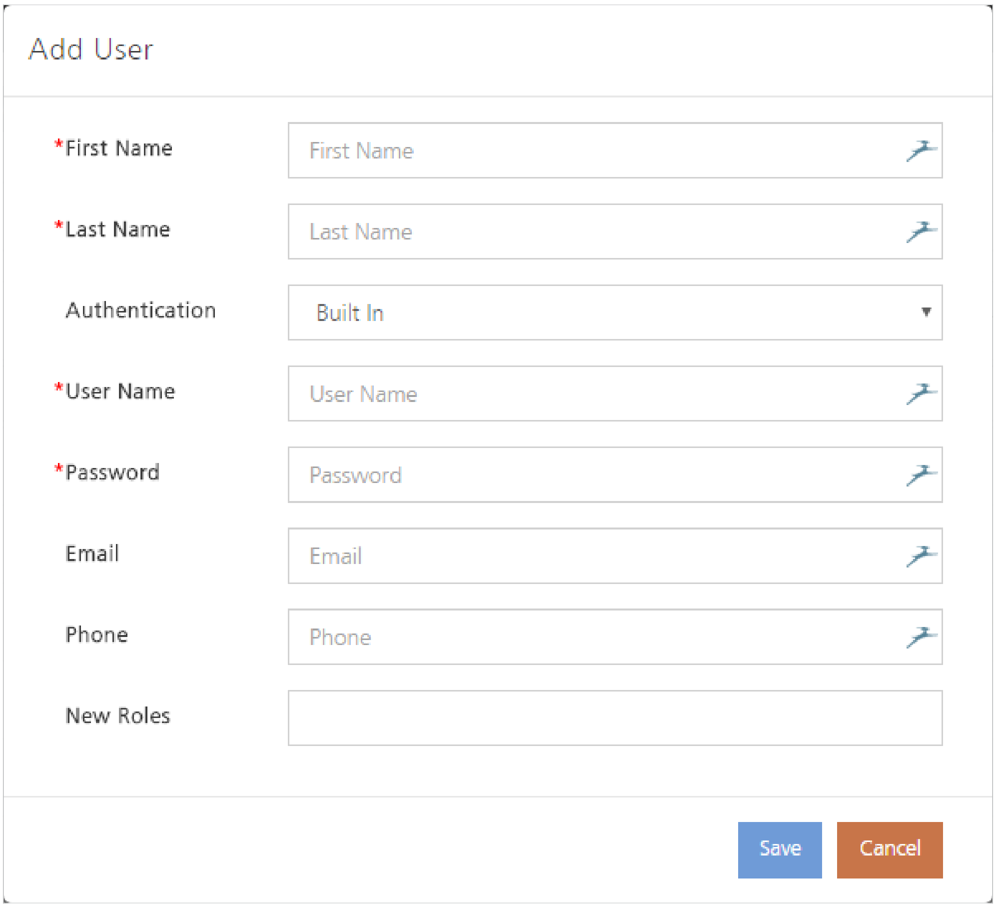
1. **Input** the address **ServerName:9785** into your web browser.
2. **Login** with the **user name** and **password** provided in your evaluation startup email.

After you login, you can see the interface with the menu at the top. Menu item access depends on **user permissions**, so each user can only access the functions specifically made available to them.

SETTING UP USERS

Log into the **browser interface** with your emailed default credentials. The first thing you'll want to do is change the **default admin user** to whomever you want to be the **admin** for the program. This can be accomplished in the ***Users*** menu. It provides a simple list of users that can be modified in a standard way. Make sure you only log in with a **User profile** in one location or licensing issues will prevent proper usage. You can create additional user credentials by creating new **User** profiles.

1. **Click** the **Users** tab in the **browser interface**.
2. **Select** the **Users** tab.
3. **Click** on **Add User**.
4. **Fill** in **First Name** and **Last Name** of **User**.
5. **Select authentication** type **Built In**.



Add user dialog

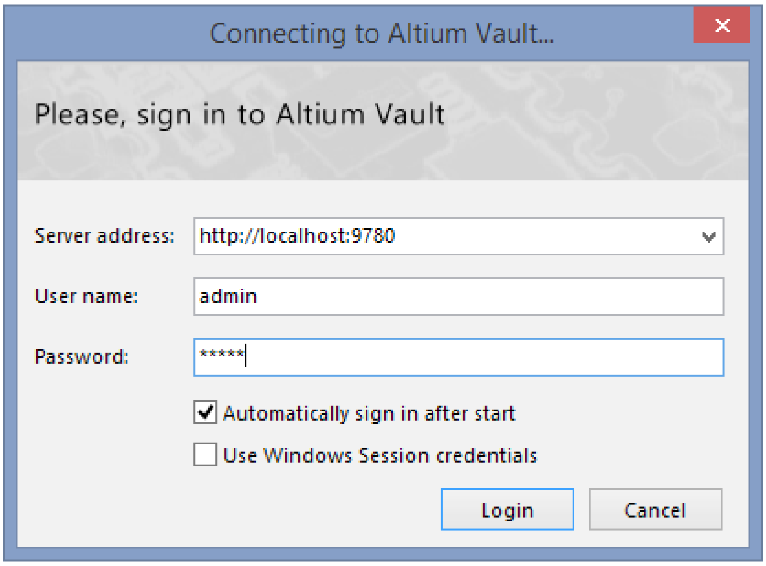
BUILT-IN AUTHENTICATION

6. **Fill in User Name** and **Password** to complete the process.
7. **Set New Roles** to **Engineers** and **Librarians**.
8. **Click Save**.

Setting **authentication type** to **Windows** will allow you to link your computer credentials and domain information to **Altium Vault**, but the topic will not be covered in this guide.

CONNECTING ALTIUM VAULT TO ALTIUM DESIGNER

1. Open **Altium Designer**.
2. Go to **DXP >> Sign into Altium Vault**.



Altium Vault sign in

3. Fill in your **Server address**, **User name**, and **Password**.

   Note: If your **Altium Vault** login credentials have been linked to **Windows** credentials, check the **Use Windows Session credentials** box.

4. Click **Login**.

FIND THE PARTS YOU WANT

ORGANIZED BROWSING

Maintaining a centralized organization structure allows everyone to know the location of your ECAD data and where to place new data. Every folder must include the following **Folder Name**, **Folder Type**, and **Naming Scheme**. Creating these three basic attributes allows every user to understand the purpose of folder and the rules for adding ECAD data. The **sample data folder** structure can be seen below. Please note that the 3-level structure structure is recommended for large number of components and ECAD data. For most users, the 2-level structure provides a sufficient structure for your ECAD data.

1. **Go** to **DXP >> Vault Explorer**.
2. **Click** the the folders you want to view in **Vault Explorer**.

Note: Expand folders using the **+ key**. Some directories will not display any items because items are located in their sub-folders.

Managed Content

- Managed Schematic Sheets
- Templates
  - BOM Templates
  - Component Templates
  - Draftsman Templates
  - Layer Stacks
  - Output Job Templates
  - Capacitors
  - Connectors
  - Crystals & Oscillators
  - Diodes
  - Fuses
  - Inductors
  - Integrated Circuits
  - LED
  - Mechanical
  - Optoelectronics
  - Radio & RF
  - Relays
  - Project Templates
  - SCH Templates

Projects

- BC0001

Trash

- Miscellaneous

Unified Components

- Components
  - Audio
  - Resistors
  - Sensors
  - Switches
  - Transformers
  - Transistors

Generic Components

- Capacitors
- Resistors

Models

- Footprints
- Symbols

COMPONENT SEARCH

Quickly find the exact component you need with extensive search capabilities that allow you to save queries and filter results. You can search for parameters including filtering by footprint, manufacturer part numbers, and more. Let's search for a specific component found in our reference design: BC0001.

SEARCHING FOR COMPONENTS

1. **Open** the **Vault Explorer** under **DXP >> Vault Explorer**.
2. **Click** the **Search tab** at the bottom of **Vault Explorer**.
3. Make sure you have the **Generic Search** selected in **Saved Searches**.
4. **Type Micro Board Stacker**, into the search bar and hit **Enter**.
5. **Right click** on **CMP-002-00046-1** and select **Navigate to**.

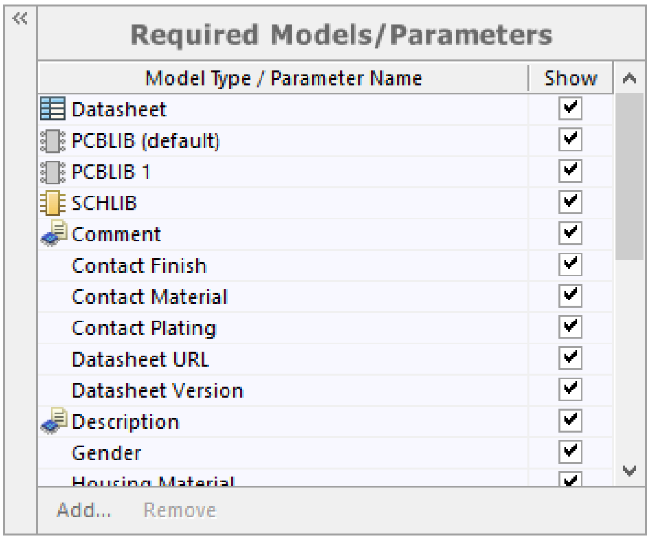
CLONING A COMPONENT

Sometimes you need a variant of an existing component with minimal changes. When you clone a component, the component will inherit component parameters, links, and templates. You can clone an existing component and change any of the associated data for component variant. This in turn will lead to creation of components with the proper naming convention without the need of user input. However, you can still change the naming of component ID if you desire a specific naming convention for cloned components as seen in following example.

6. **Right click** on **CMP-002-00046-1** and select **Operations >> Clone CMP-002-00046-1**.

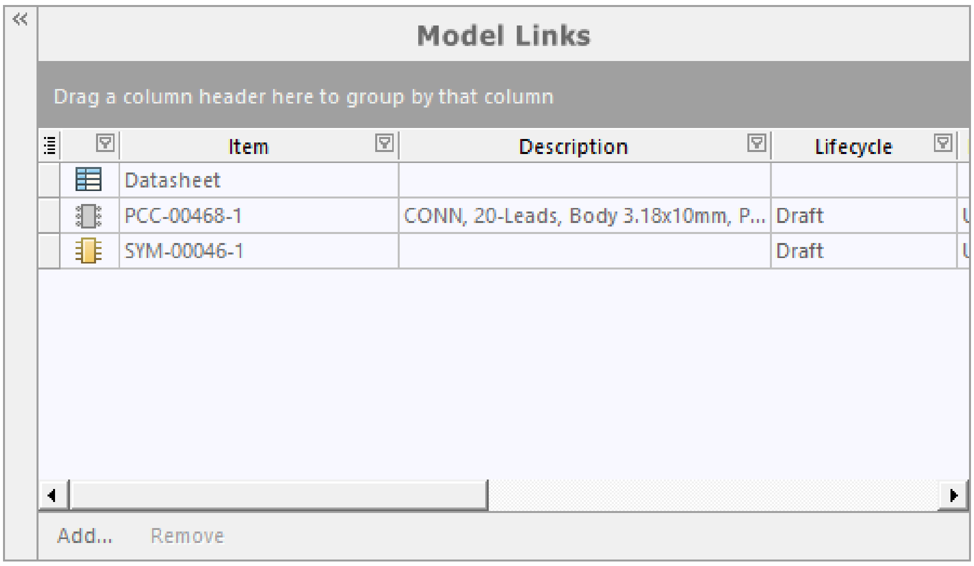
Note: This step is a continuation of the previous section. The CmpLib will open in Altium Designer for modification. The following steps explain how parts should be organized for release.

7. In the **Altium Designer** workspace, you can change **Required Models/Parameters** for your cloned component.
8. **Close** the **Vault Explorer**.



Required Models/Parameters in CmpLib

9. In the **Model Links** sections, you can make any changes to **linked symbols** and **footprints**.



Model Links in CmpLib

10. **Click** the **Projects** tab on the bottom left.

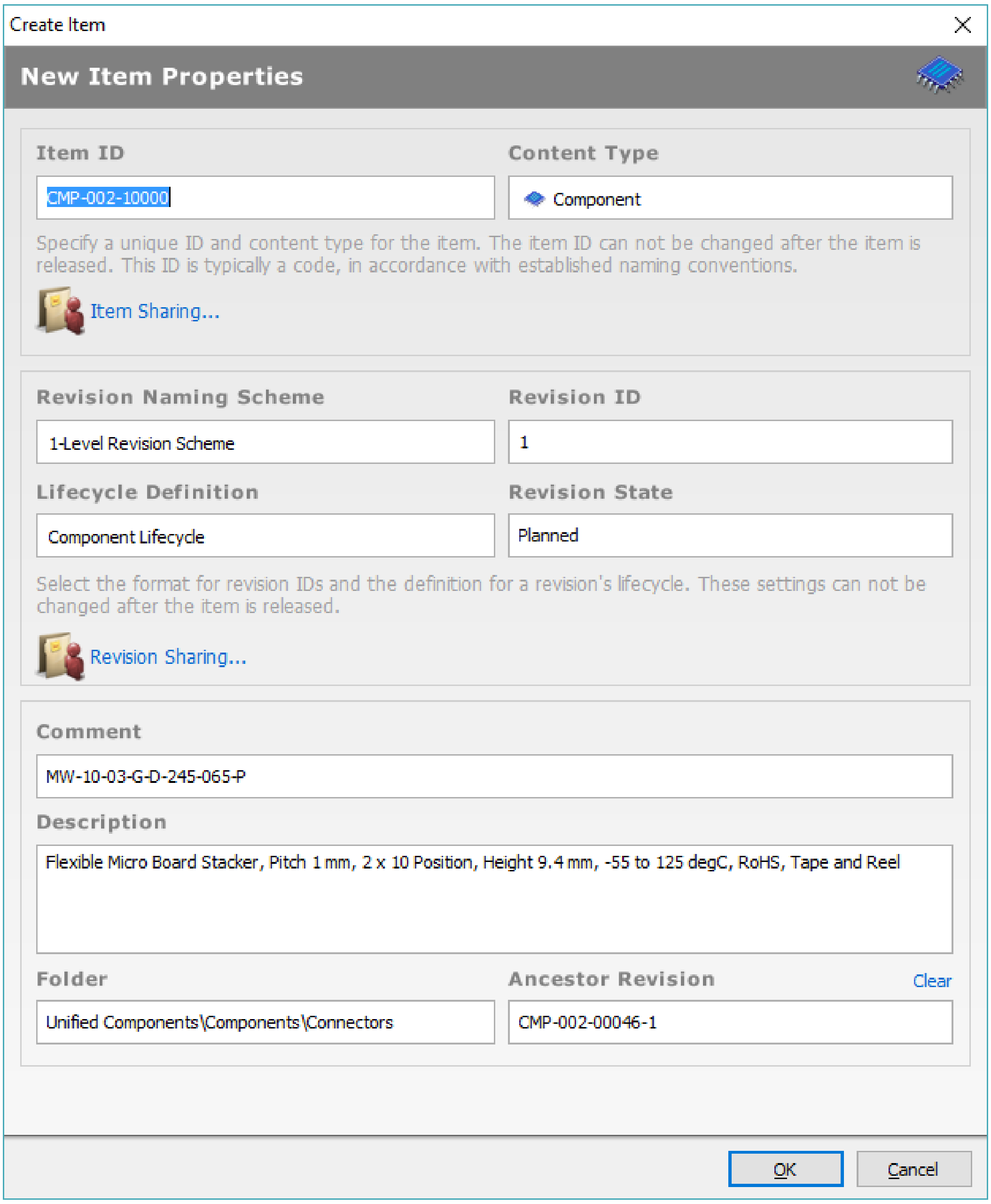
Note: If you don't see the **Projects** tab, open the panel by **clicking** on **System >> Projects** in the bottom right corner.

11. **Right click** on **Copy of CMP-002-00046-1** in the **Projects Panel** and **Save**.

12. **Right click** on **Copy of CMP-002-00046-1** and select **Release to Vault...**

13. **Change** the **Item ID** of the component to **CMP-002-10000**.

Note: The component name will automatically set to the next available number following the component naming scheme. The manual name change creates a known component name for use later in the guide and should not be used in normal operation.



Model Links in CmpLib

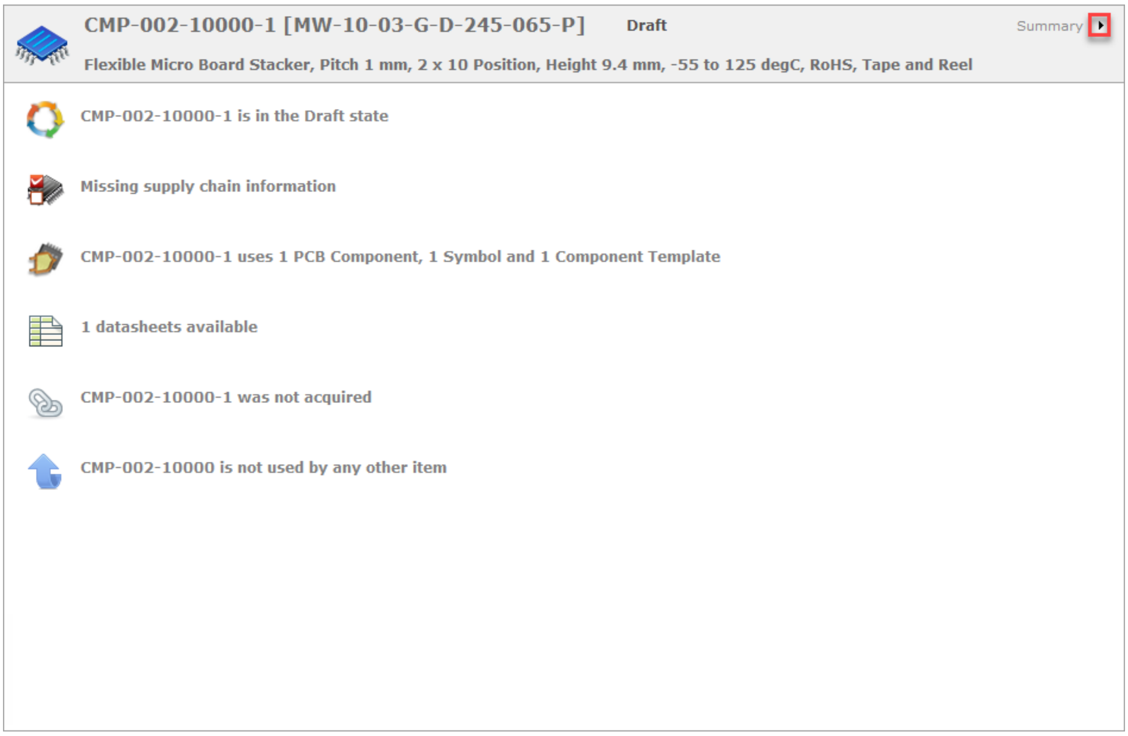
14. **Click OK** in the **Create Item** window.

NAVIGATING COMPONENT DATA

Once you have identified a potential component, it is important to verify more information about the component. The bottom half of **Vault Explorer** contains different data views depending on the item type selected. The following section takes a look at the **Summary**, **Preview**, **Datasheet**, and **Supply Chain** views for components.

1. **Open** the **Vault Explorer** under **DXP >> Vault Explorer**.
2. **Search** and **select** the component you just cloned **CMP-002-10000**.

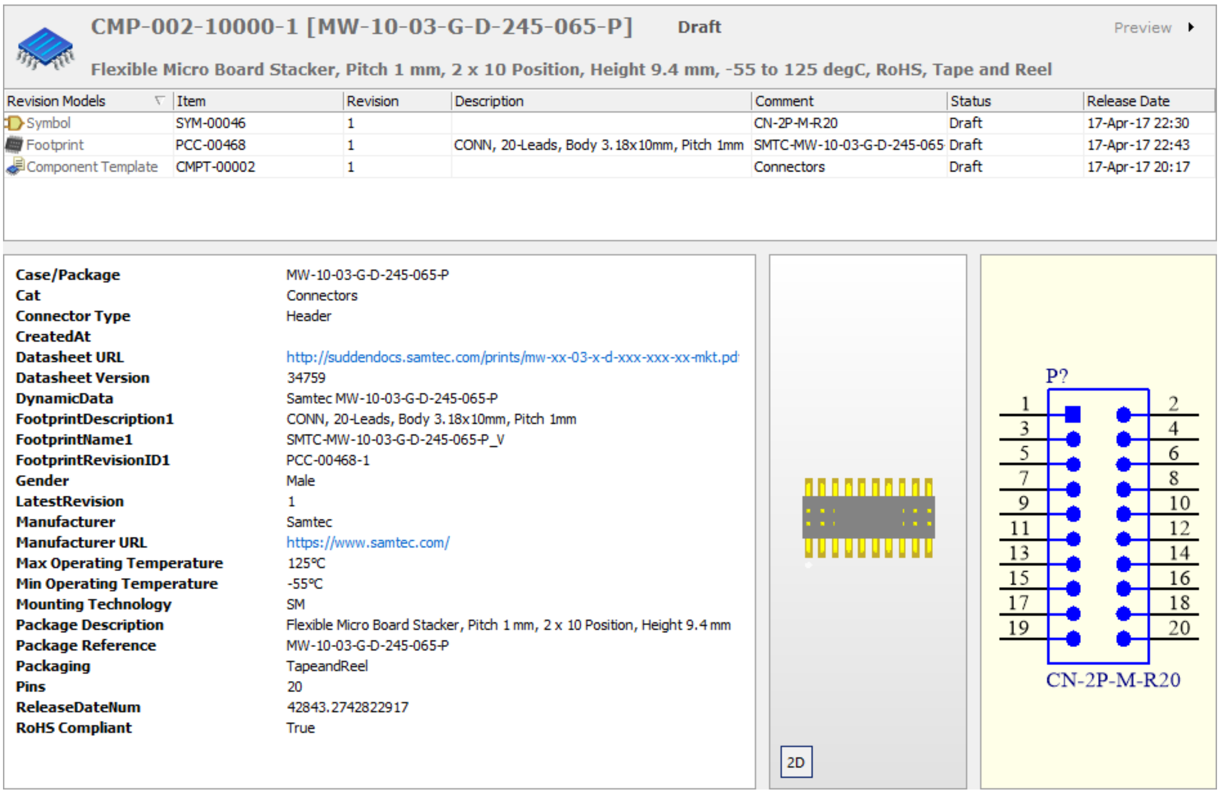
Note: The lower right window should be in **Summary** view by default. The **Summary** view provides insight and links to the other data views.



Altium Vault component summary

3. **Click** the **Black Triangle** at the top right corner (highlighted in the illustration with a red square) of the information section and select **Preview**.

Note: The **Preview** view shows linked **model items**, **parameters**, **3D Models**, and **Symbols** associated with a component.



Altium Vault component preview of the pane

4. **Click** the **Black Triangle** at the top right corner and select **Datasheet**.

Note: The **Datasheet** view displays attached datasheet files to the component to provide direct access to component specifications. Cloned components do not copy original datasheets to ensure you don't accidently transfer the wrong datasheet

5. **Click** the **Black Triangle** at the top right corner of the information section and select **Supply Chain**.

Note: The **Supply Chain** view shows dynamic **supplier links** with **pricing**, **availability**, and **manufacturer**, as well as **supplier**, **part numbers** for a given component. *However, the supply chain information for a new or cloned component will be blank and must be populated with a designated supplier.*

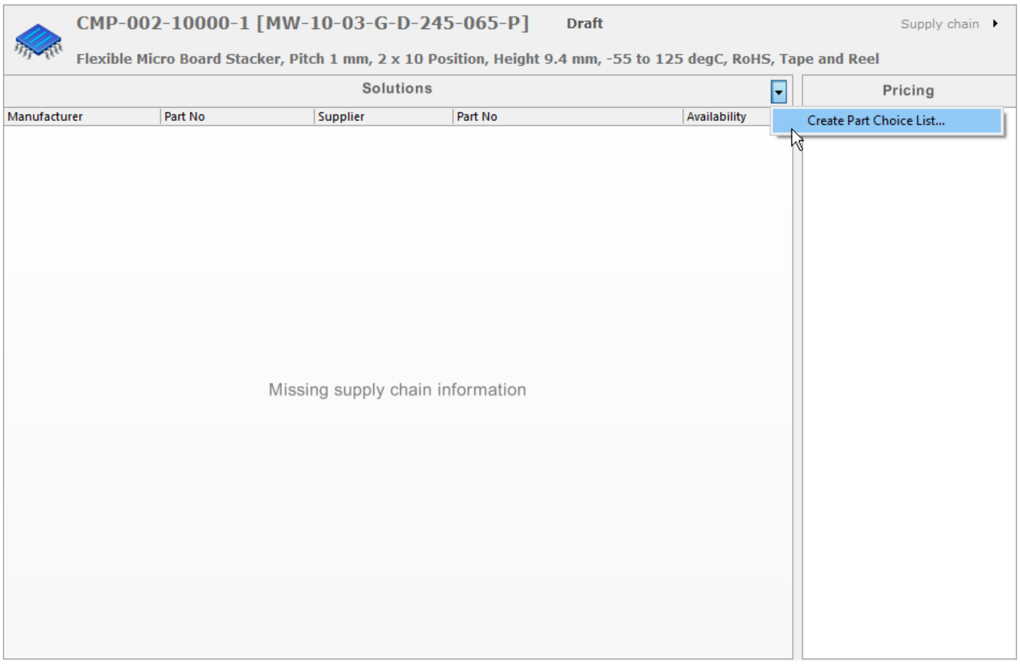
APPLY THE PARTS YOU NEED

Now that you have a better understanding of a component, you can see how it can be used in a design. The cloned component from last section is missing supply chain information that must be added. The same process can be used if there is ever a problem with part availability from the original supplier. Finding an alternate supplier and linking supply chain data takes less than a minute.

LINKING SUPPLY CHAIN SOLUTIONS

Note: This step is a continuation of the previous section.

1. In the **Supply Chain** view that you just navigated to, **click** the **Black Drop down Arrow** in the "**Solutions**" header (see figure). Then click **Create Part Choice List...**

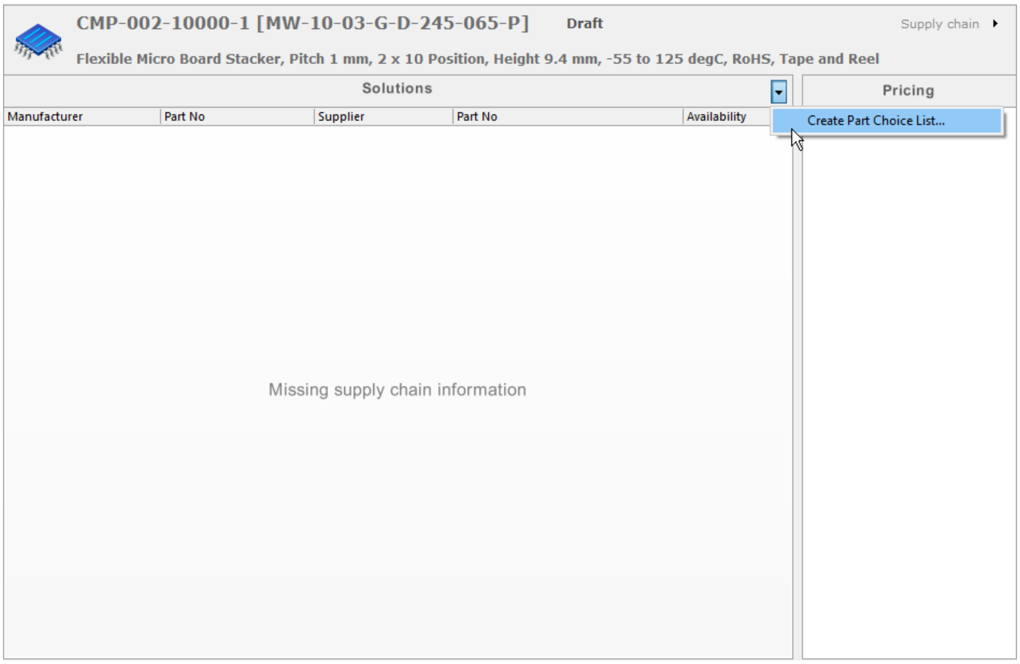


Altium Vault component supply chain

2. In the **Keywords** search bar, ensure that the manufacturer's part number **MW-10-03-G-D-245-065-P** is populated.

Note: The **Keywords** search bar will auto-populate with the content of the **comments** section of each item.

3. As shown in the figure, select one or more resulting parts from the search, then **click** the **>>** to add to the **Manufacturer Part Choices** list.



Manufacturer part choice selection for Altium Vault component

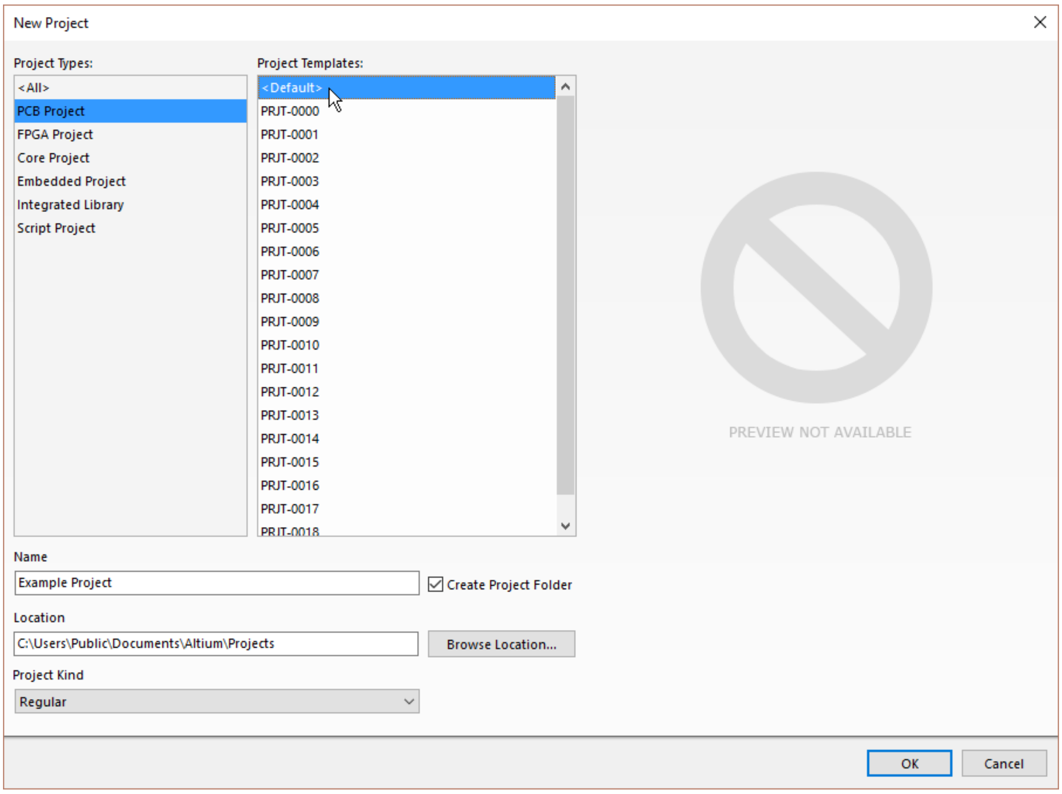
4. **Click OK**.

These steps link the manufacturer part number to the Altium Vault item. The supply chain view will then pick up on that part number (or part numbers) and show the availability based on Altium Verified suppliers and your supplier choices.

ADDING A COMPONENT TO A DESIGN

Once a component has the relevant information, you can use it in a design. Using an **Altium Vault** component is simple as dragging and dropping it into a schematic. Dock the **Vault Explorer** panel by dragging it to the desired area of **Altium Designer**. Using a component, such as the **ATMEGA16U2 microcontroller**, in a schematic takes seconds to find and use.

1. **Create** a **new project** by going to **File >> New >> Project...**
2. **Select PCB Project** as your **Project Type**.
3. **Select <Default>** as your **Project Template**.



New project dialog

4. **Name** the project **Example Project**.
5. **Set Location** to a common location. For example, **C:\Users\Public\Documents\Altium\Projects**.
6. **Set Project Kind** field to **Regular**.

Note: The **Project Kind** can be altered to create **managed** projects.

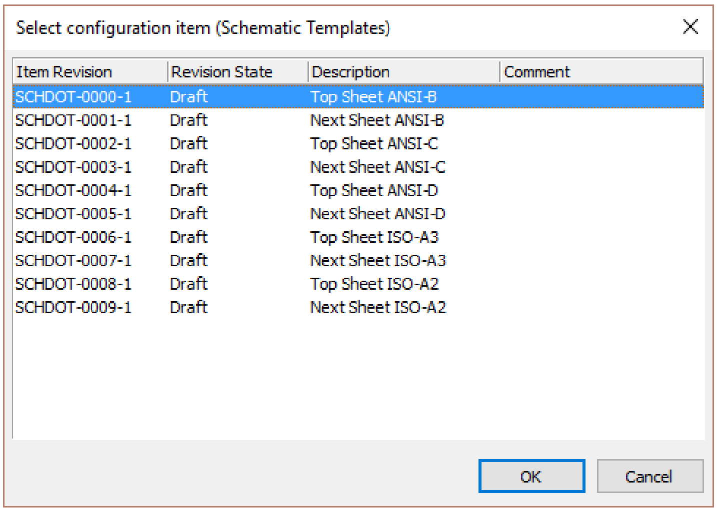
7. **Click OK**.
8. **Create** a **Schematic document** by **right clicking** on the project in the **Projects Panel** and selecting **Add New to Project >> Schematic**, **select** template **SCHDOT-0000-1** from **Altium Vault** and press **OK** to create the schematic.

9. **Search** and **select** the latest component revision of **ATMEGA16U2** from **Vault Explorer Search**.
10. **Click** and **Drag** the **latest component revision** to the **open document**.
11. **Release** on the location you would to **place** the **component**.

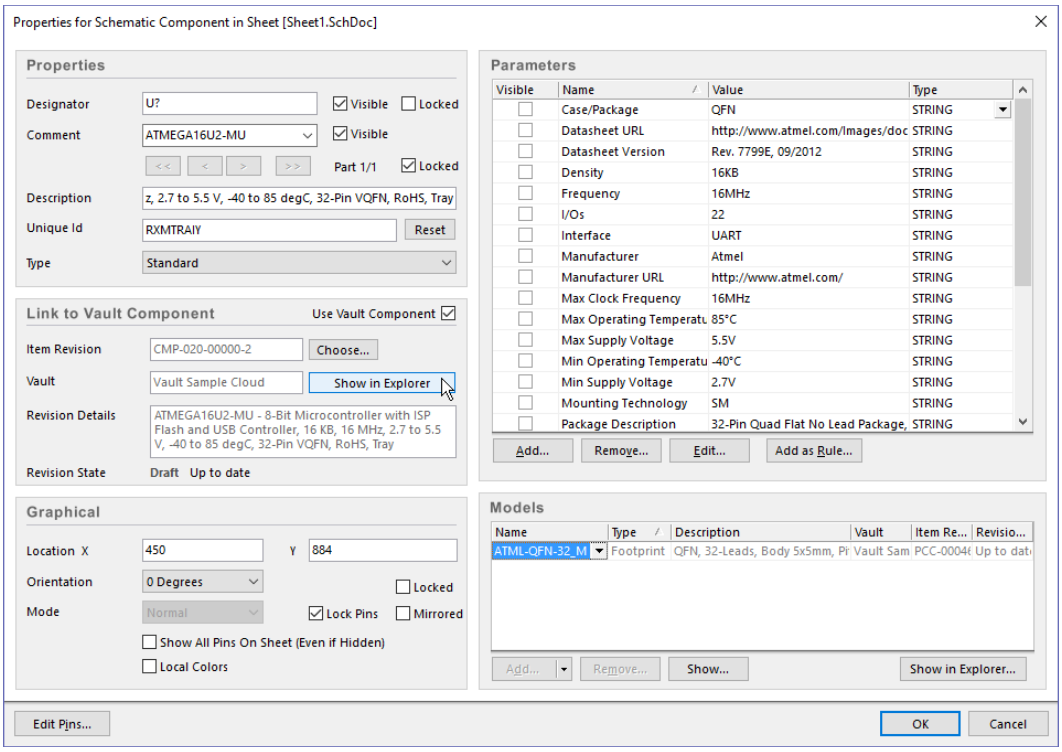
OPEN LINKED DATA FROM SCHEMATIC

Finding the source of any placed component is simple.

1. **Double Click** the **component** placed in the last section to open **Component Properties**.
2. **Click** the **Show In Explorer** button.



Schematic template selection



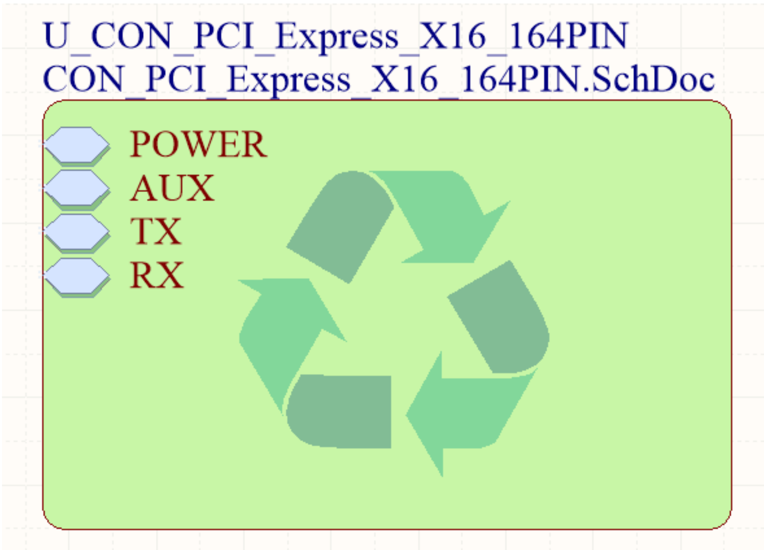
Component properties

3. Press **OK** to close **Component Properties**.
4. The **component** will be seen in **Vault Explorer**.

DESIGNING WITH REUSE BLOCKS

Everyone has validated circuit logic that they would like to reuse for new designs. Managed sheets provide you with the opportunity to manage and reuse design blocks. You can open up the managed sheets and bring in your validated circuit logic into your schematic design sheets. When properly constructed managed sheets will contain links to vault components, which are easily updateable to the latest component revisions. This linking means you never have to worry about obsolete components in new designs.

1. **Open** the **schematic sheet** from created **Example Project** from last section.
2. **Open Vault Explorer**.
3. **Select** the **Managed Content >> Managed Schematics Sheets** folder in the **Vault Explorer**.
4. **Right Click** on **SCH-0001** and **select Place SCH-0001**.
5. **Place SCH-0001** on the **schematic** document.



Managed sheet

RELEASE YOUR ECAD DATA

Everything in your design, from component datasheets to output files, comprises your ECAD data. It is important to know how to build things from scratch. Starting from components all the way to output job release.

BUILD A COMPONENT FROM SCRATCH

Say that you need a component that hasn't already been created in Altium Vault. You can initiate a **Part Request** for any component that requires creation.

PART REQUEST

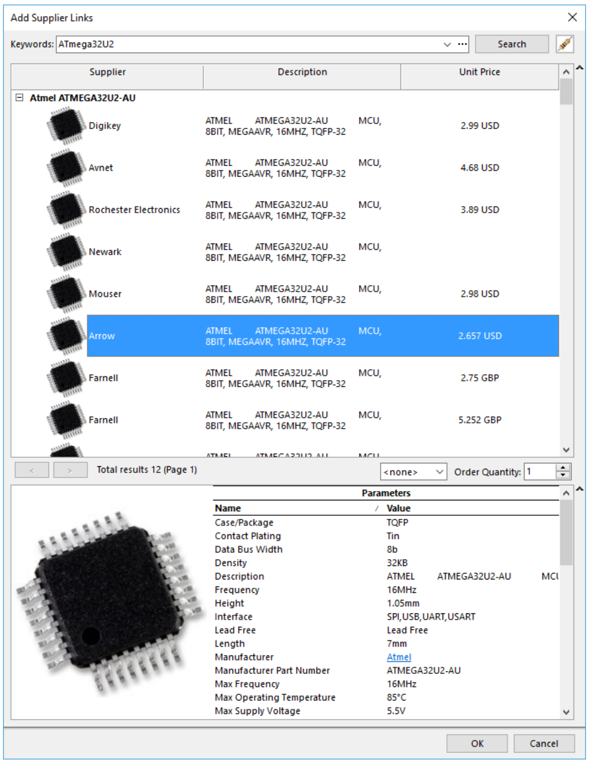
Part Requests can be created and maintained within **Altium Designer**, through the **Vault Explorer** panel. If your part search doesn't give you the results you need, you can use the **Add Request** button in the **Search** tab of the **Vault Explorer** panel.

1. **Open Vault Explorer**.
2. **Perform** a **Search** for **ATmega32U2**. The search will return "Nothing Found."

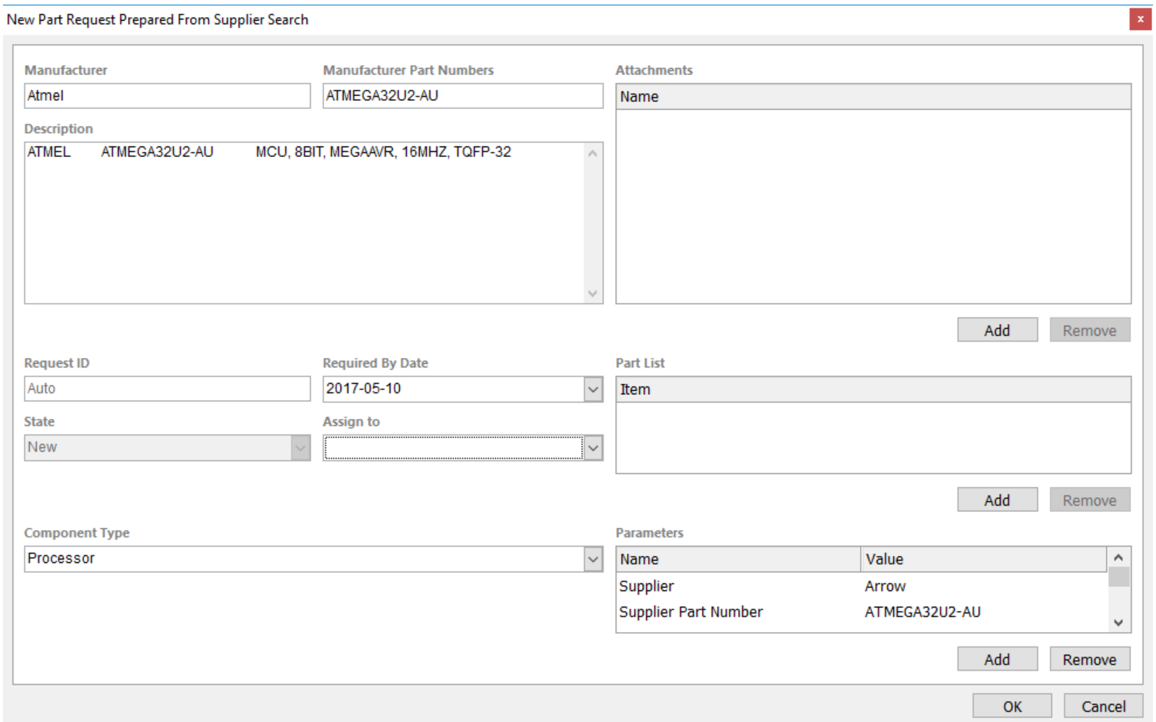
3. **Click** the **Add Request** button.
4. **Select From Supplier Search...**
5. In the **Keywords:** section type **ATmega32U2** and **click Search**.
6. **Double Click** on the **Unit Price column header** to **Filter**.
7. **Select** a **Supplier**.
8. **Click OK**.

Note: In the **New Part Request** dialog, **use** the **options** and **controls** in the dialog to supply as much information as possible on your desired part. It may take a few seconds to populate the dialog box.

9. **Set Assign To** to your account.
10. **Set Component Type** to **Processor**.
11. **Click** the **OK** button to create request.



Add supplier links dialog



New part request dialog

For both the originator of a part request (Requester) and the user(s) defined in roles associated to the Librarian role (Librarians), requests are presented through the Vault Explorer panel using a dedicated Part Requests folder. The number next to the Part Requests folder name indicates how many requests there are. For a designer/engineer, the folder will present entries for only those parts they have actually requested. For a librarian, they will see part requests specifically assigned to them, as well as part requests that have yet to be assigned to a particular librarian.

CREATE COMPONENT FOR RELEASE

Unified components can be created in **Altium Designer** with a **component library** or **CmpLib**. For this example, the **ATmega32U2** component can be created utilizing the existing symbols and footprints used for the **ATmega16U2** component.

1. Open Vault Explorer.
2. Select the Parts Request folder.

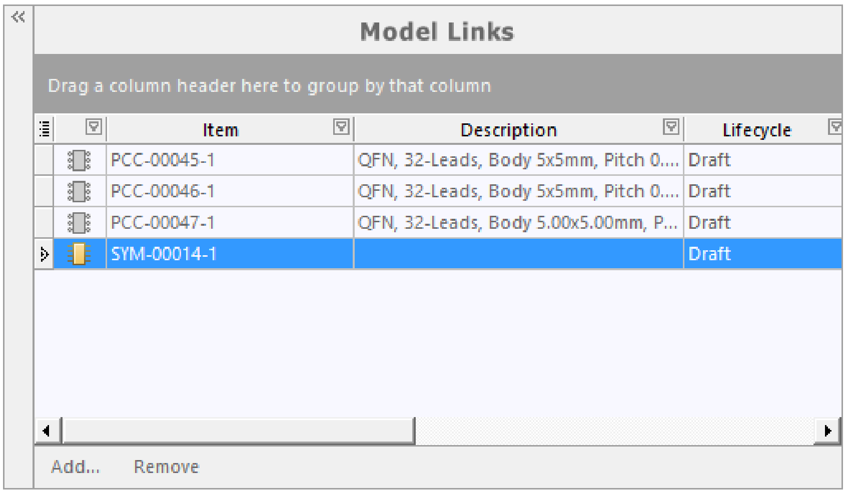
Note: If you do not see the **Parts Request** folder, hit the **refresh key**. If the folder still does not appear, check your **user permissions** and ensure they include **Engineers** and **Librarians**.

3. Right Click on the Part Request created and select Operations >> Create Components...

Note: A CmpLib will open to initiate component creation.

Model Links in CmpLib

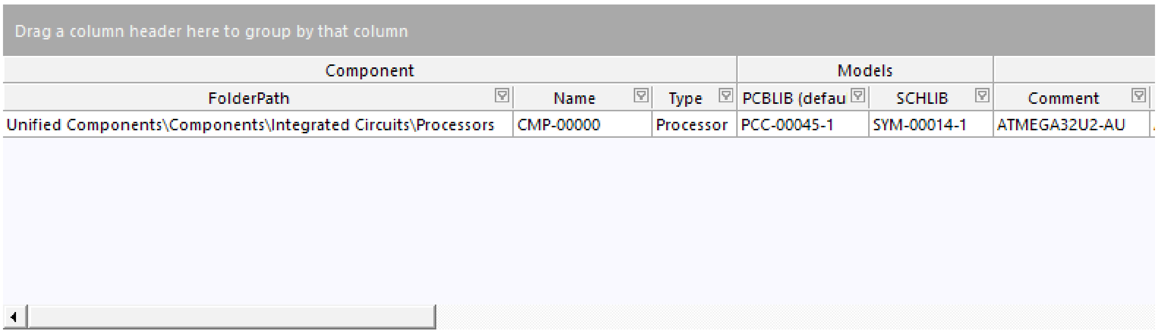
4. **Click Add... >> SCHLIB**.
5. **Search** and **select SYM-00014** and **click OK**.
6. **Click Add... >> PCBLIB**.
7. **Search** and **select PCC-00045** and **click OK**.
8. **Click Add... >> PCBLIB**, **select PCC-00046**, and **click OK**.
9. **Click Add... >> PCBLIB**, **select PCC-00047**, and **click OK**.



Model links in CmpLib

Component Parameters in CmpLib

10. **Select** the first **cell** beneath the **FolderPath Column Header** for the **component** by **clicking** the **...**’ **button** to **navigate** to the following path**.: Unified Components >> Components >> Integrated Circuits >> Processors**.
11. **Select** the first **cell** beneath the **PCBLIB Column Header** and **link PCC-00045-1**.
12. **Select** the first **cell** beneath the **SCHLIB Column Header** and **link SYM-00014-1**.
13. **Save** the **CMPLIB**.



Component			Models		
FolderPath	Name	Type	PCBLIB (defau	SCHLIB	Comment
Unified Components\Components\Integrated Circuits\Processors	CMP-00000	Processor	PCC-00045-1	SYM-00014-1	ATMEGA32U2-AU

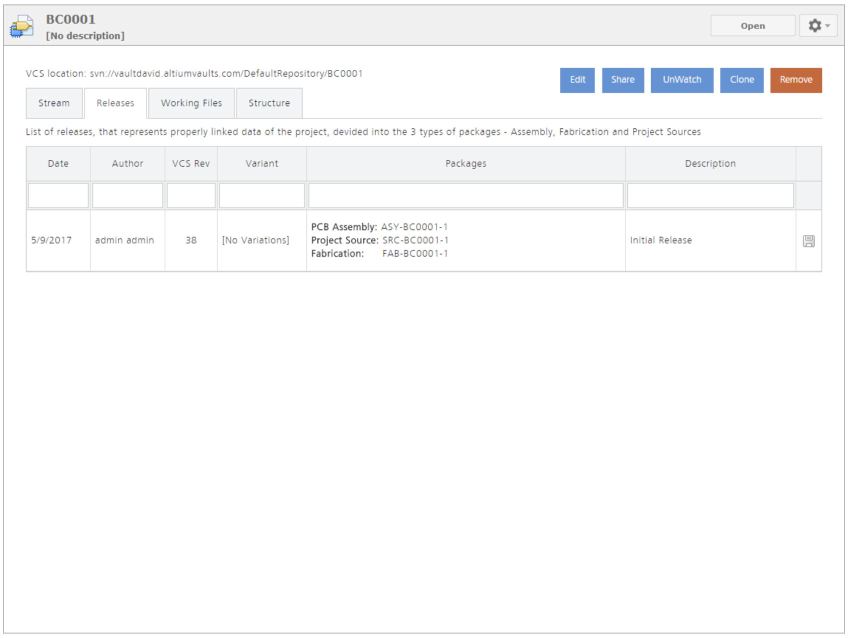
Column headers in CmpLib

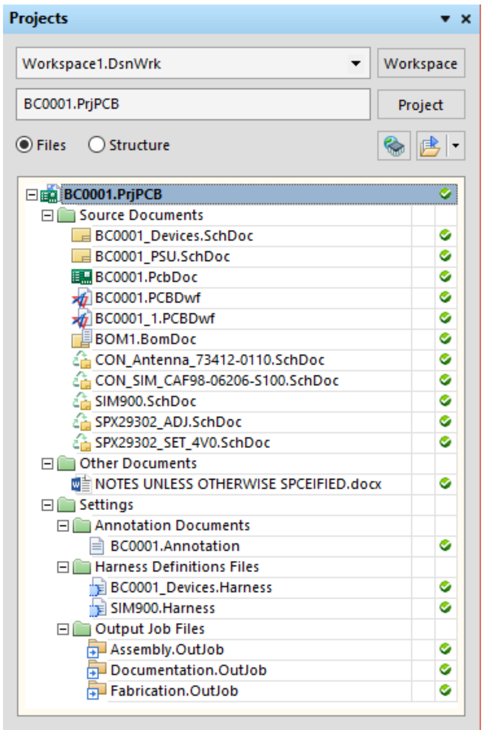
14. **Go** to **File >> Release to Altium Vault...**
15. **Click Release Item**.
16. **Click OK**.
17. **Click Close**.

ACCESSING THE REFERENCE PROJECT

Now you can take a look at releasing other ECAD data into **Altium Vault**. Using the released reference project **BC0001** found in **Altium Vault** you will perform a design review and prepare the reference project for release to **Altium Vault**. Open up the project and learn more about it.

1. **Open** the **Vault Explorer**.
2. **Expand** the **Projects** folder in the **Vault Explorer**.
3. **Select** the project **BC0001**.
4. **Click** the **Open** button.





DESIGN REVIEW FOR RELEASE

One of the major benefits of the release process is the automated design checks. Design integrity is ensured because validation must occur for release. Using ***Output Job*** files, you can produce all relevant design outputs that are necessary for fabrication and assembly. The following Output Job structures are basic recommendations for design outputs. It is important to contact your manufacturer to ensure you are including all of the necessary outputs necessary for manufacturing.

Assembly Output Job File

Draftsman
Assembly Drawing
Generates pick and place files
Test Point Report
IPC-D-356 Netlist
Bill of Materials
Copy of Bill of Materials

Documentation Output Job File

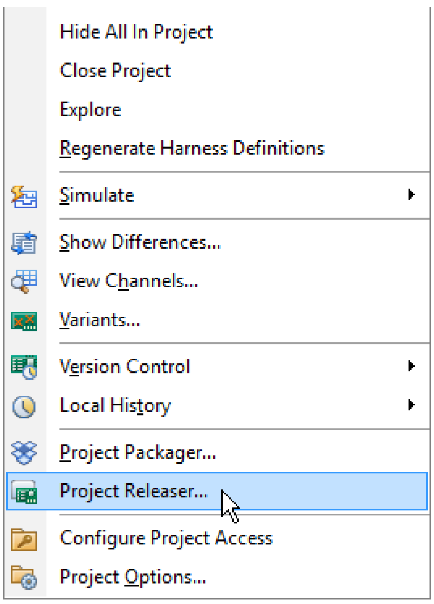
Schematic Prints
Design Rules Check
Differences Report
Electrical Rules Check

Fabrication Output Job File

Draftsman
Fabrication Drawings
Gerber Files
NC Drill Files
ODB++ Files
TestPoint Report

RELEASING A DESIGN PROJECT

1. **Right Click BC0001.PRJPCB** in the **Projects Panel** and **select Project Releaser...**

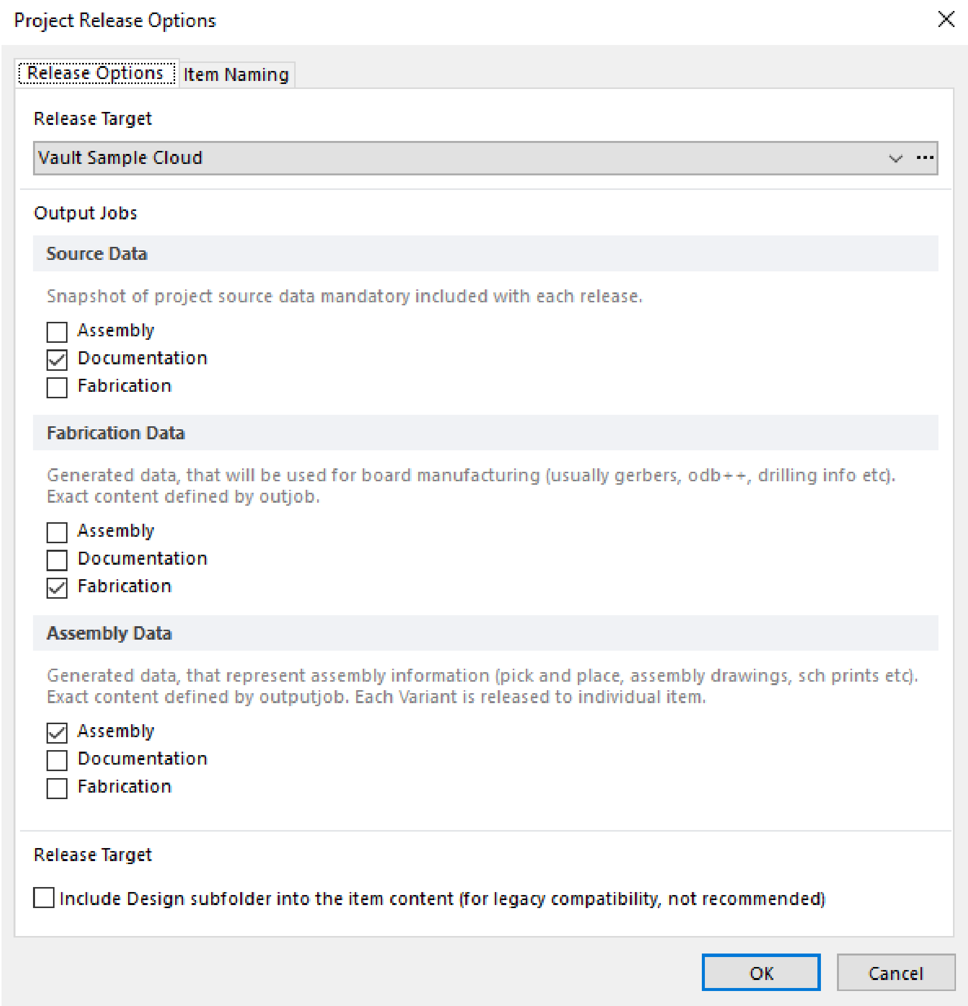


Column headers in CmpLib

2. Click **Options**.

Note: In the **Project Release Options**, you can decide which ***Output Job*** files to link to the different **Output Data** types.

3. **Match** the **Project Release Options** to the image below.



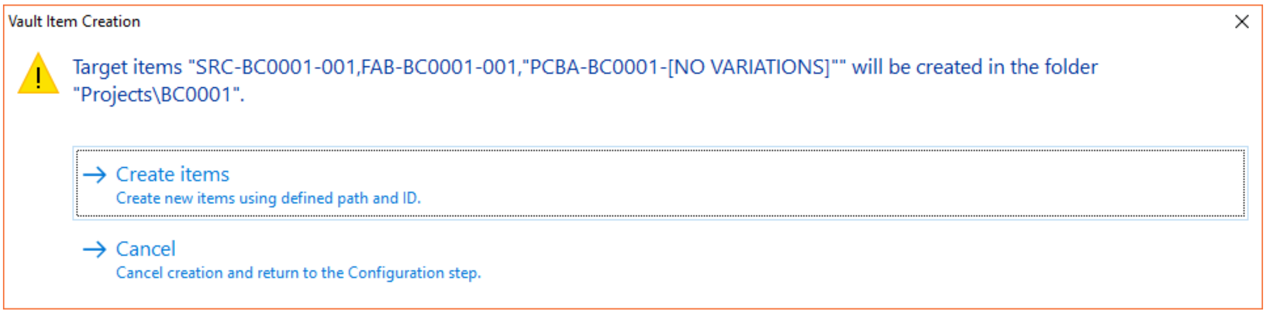
Project release options

4. Click **OK**.

5. Click **Prepare**.

6. Click **Create items**.

Note: The process can take a while, so you might want to grab a cup of coffee or maybe some tea. If preparation of an output fails, click the **Details** button of section to access the produced files. **Double Click** on the failed file to open it to view **errors**. Open the **Messages** panel in **Altium Designer** for more information.



7. Click **Release**.

8. Set **Release Note:** to **Reference Project Release!**

9. Click **OK**.

11. Click on the **Fabrication** link to view the project in **Altium Vault**.

CORRECT DESIGN MISTAKES

DEALING WITH OBSOLETE PARTS AND DESIGN ERRORS

What if during the component check—or in any other stage of the process for that matter—an error is detected? You should also take into account that an error in a component probably means there is an error in the component's parent items as well, such as managed sheets, sheet templates, or even entire projects, depending on the nature of the error. You need to check the ***Where-used*** view of **Vault Explorer** for these parent items.

FINDING IN USE ITEMS WITH WHERE-USED VIEW

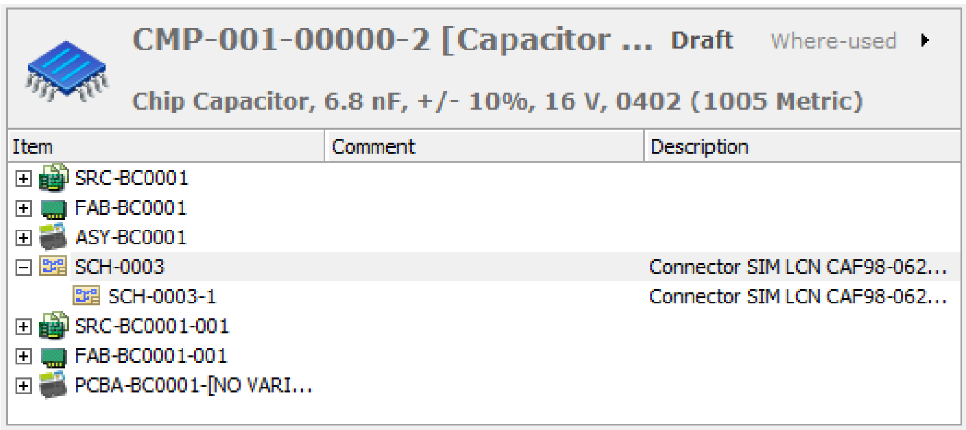
From the **Where-used view**, you can see all places where a selected item is used. In the example case, you are viewing a component and the projects containing the component. You can use the same view for any item type, including symbols, footprints, templates, and managed sheets.

1. **Go** to **DXP >> Vault Explorer**.

2. **Search** for **CMP-001-00000**.

3. **Select** the **latest revision** of **CMP-001-00000**. In this case, it should be **CMP-001-00000-2**.

4. Click the **Black Triangle** at the top right corner of the information section and select **Where-used view**.



Component where-used functionality

CREATE AN ITEM REVISION

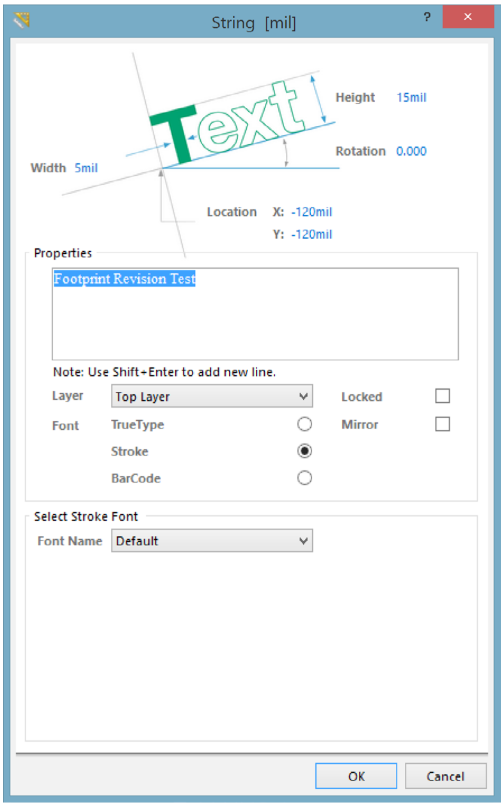
Say you have identified component **CMP-002-00046-1** as a component with a footprint error. You can create a footprint and component revision to demonstrate to fix the issue.

FOOTPRINT REVISION

1. **Go** to **DXP >> Vault Explorer**.
2. **Search** for **CMP-002-00046-1**.
3. **Select CMP-002-00046-1**.
4. **Switch** to **Preview** view.
5. **Right Click** on **Footprint PCC-00468** and select **Edit PCC-00468-1**.

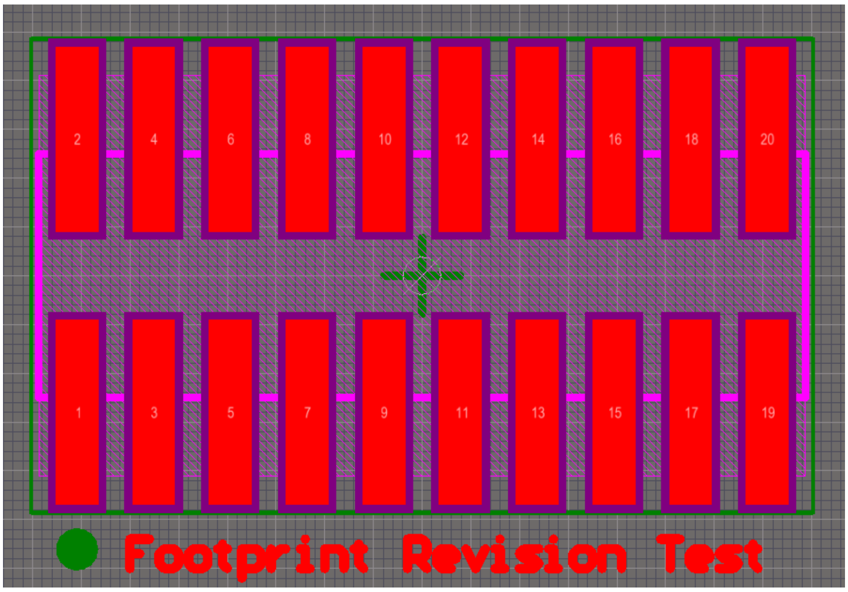
Note: A **PCB Library** will open in **Altium Designer** to begin modifications.

6. **Place** a **string** with **Right Click** and **select Place**, **String**.
7. **Press** the **Tab** key to bring up the properties
8. **Set Height**: to **15 mil** and **Width**: to **5 mil**.
9. Under **Properties**, **set** the text to **Footprint Revision Test**.



String dialog

10. **Click OK**.
11. **Place** the string under the footprint by **clicking** to place.



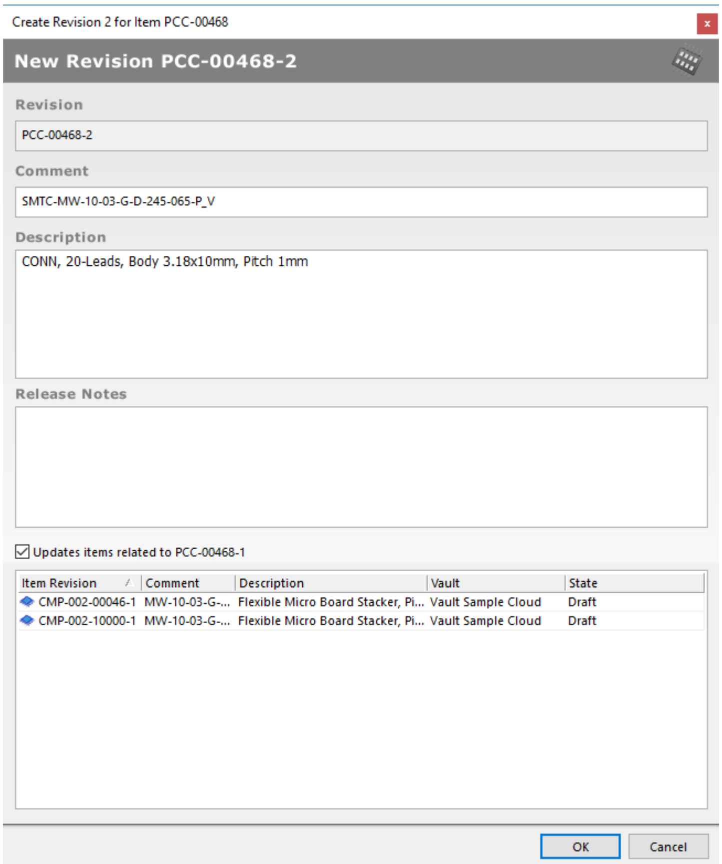
PCB footprint in 2D view

12. **Press Control + S** to **Save PCC-00468-1**.

13. **Press Control + Alt + S** to **Release to Vault...**

Note: You can **Save** and **Release to Vault...** with a **right click** on the **item** in the **Projects** panel.

14. **Click OK** on the **Create Revision dialog** that appears.



Create component revision dialog

15. **Press Control + S** to **Save CMP-002-00046-1**.

Note: The change in linked footprint **PCC-00468-1** opens a CmpLib to allow revision for **CMP-002-00046** and **CMP-002-10000** because they are both using the footprint.

16. **Go** to **File >> Release to Vault...**
17. **Click Release Items** and on the resulting **Confirm Release** dialog click **OK**.
18. **Close** the **Release Manager**.

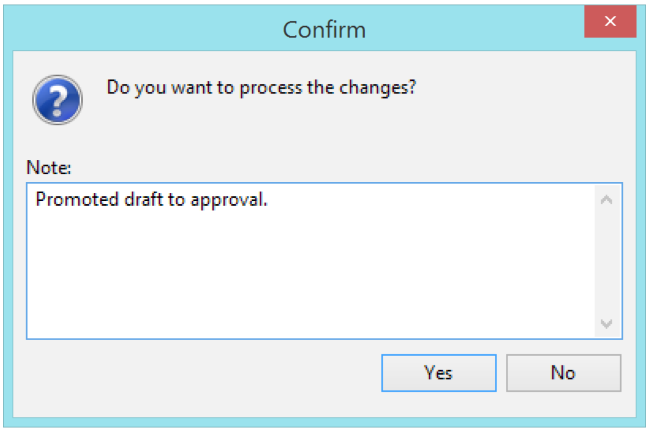
ITEM LIFECYCLE CHANGE

When you **update** or **release** a component it will go to the **default state**. The **default state** for components is **Draft**. You will update the **state** to **Prototype**.

1. **Open Vault Explorer**.
2. **Search** for **CMP-002-00046-2**.
3. **Right Click** on **CMP-002-00046-2** and **select Operations >> Change states...**

Note: The columns give you information about the item and its current **lifecycle state**.

4. Under the column header **Next State select Promote 2 to Prototype**.
5. **Click Process**.
6. **Type "Promoted Draft to Prototype."** in the **Note:** section.



Item lifecycle change dialog

7. **Click Yes**.

UPDATING TO LATEST REVISION WITH THE ITEM MANAGER

You can use the ***Item Manager*** to correct design errors and update components. Now that you know how to identify where components are being used, you can see what projects would need to be updated when component changes are made. Updating the components individually would take a long time, since this method would be insupportable for updating all components in even a small scale board design. The **Item Manager** simplifies and automates that process by using advanced parameter matching (***Automatching***) and a **bulk update** approach. A typical application of the **Item Manager** would be to update an existing board design to use **Managed Components** that have been migrated to **Altium Vault** from **Libraries**.

1. In **Vault Explorer**, **navigate** to the **reference project** found in **Projects >> BC0001**.
2. **Click Open**.

Note: If you do not see the **Open** button, **click** the ⚙ **>> View >> Project View**.

3. **Open** the schematic document **BC0001_Devices.SchDoc**.
4. Go to the **Tools** menu from an active schematic document and click **Tools » Item Manager**.
5. Select the lower **Components tab** to populate the list of components in the current design.

Note: The left section of the dialog shows component settings of the active project (**Current Settings**), while the right section lists how they will change (**New Settings**) when suitable **Managed Components** in **Altium Vault** have been assigned.

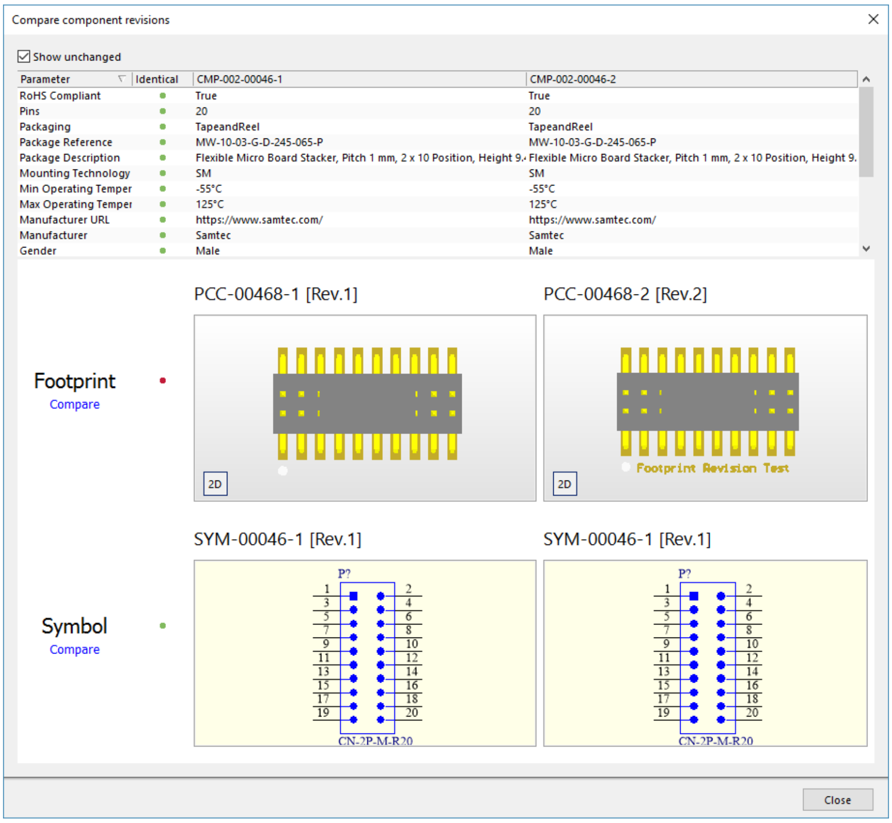
6. **Select all** components.
7. **Right Click** in the window and **select Update to latest revision**.
8. **Select** the **Apply ECO** option from the **ECO...** button menu.
9. **Click Close** once **ECO** completes.
10. **Save** the schematic document **BC0001_Devices.SchDoc**.

The executed **ECO process** will update the project components accordingly, which will then be listed in the **Item Manager dialog** as currently up-to-date **Managed Components**. In the **Schematic Editor**, the updated components are linked to their matched **Managed Components** in **Altium Vault** – the **active link information** will detect a change in the **Managed Component's Revision state** when it is subsequently updated. It is recommended you release the example project again to keep all of your ECAD data in sync and organized.

COMPONENT REVISION COMPARISON

If you want to verify **item** differences or changes, you can compare two revisions to verify differences. The comparison can be made on two **items** of the **same type**, for example to compare similar components with different parameters. We will review the changes you made earlier to component **CMP-002-00046**.

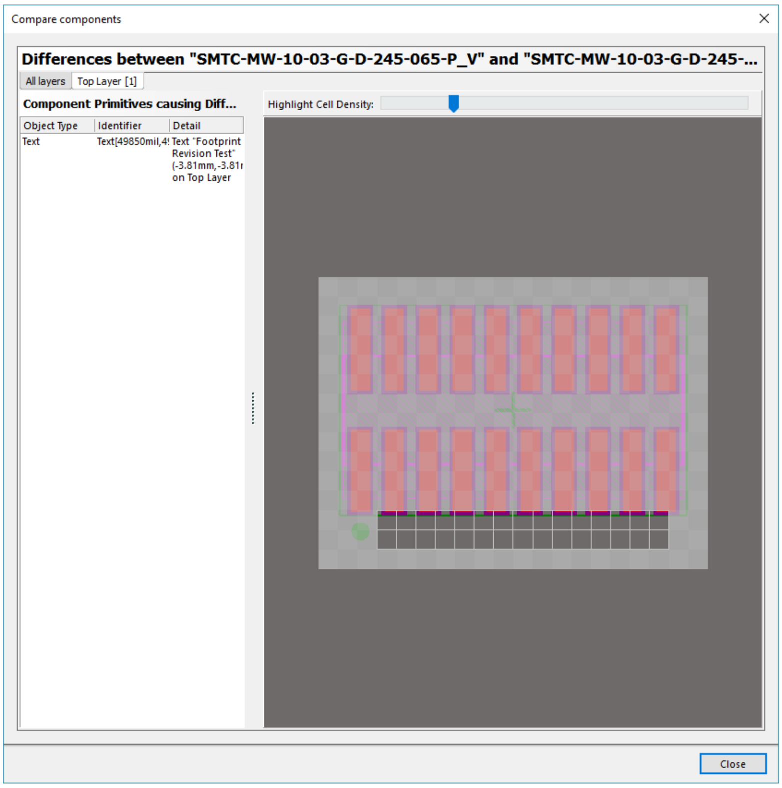
1. **Open Vault Explorer**.
2. **Search** for **CMP-002-00046**.
3. **Click & deselect** ⚙ **>> Show only latest** to enable view of **all** revisions.
4. Select both **CMP-002-00046-1** and **CMP-002-00046-2**.
5. **Right Click** on **CMP-002-00046-1** and **select Operations >> Compare...**



Component revision comparison

Note: All identical parameters are **green** and differences **red**.

6. **Click** the **Compare** link under **Footprint** for more information on differences.



Component footprint comparison

7. **Close** when you have reviewed the differences.

Note: The comparison system works with selected comparison cell density at the top of the and displays component primitives causing difference on the left hand side.

DESIGN CHANGE COMPARISON

Version Control has been used in the **software design** realm for decades, but it hasn't reached the same levels of use in the **hardware design** realm. **Version Control** allows the comparison of design documents to help track and pinpoint design decisions to determine their effectiveness. The ability to compare design documents facilitates identification of when design errors were introduced into your design. You can make some changes to the reference project and commit the project to version control for comparison.

1. **Open** the **reference project**, **BC0001**, from **Altium Vault**.
2. **Open** the schematic document **BC0001_Devices.SchDoc**.
3. **Search** for **CMP-002-00046-2** in **Vault Explorer**.
4. **Place CMP-002-00046-2** anywhere in the **schematic document**.
5. **Save** the **schematic document**.
6. **Right Click** on the **schematic document** in the **Projects** panel and **select Version Control >> Commit...**
7. **Type "Testing Version Control."** in the **Comment** section.
8. **Click OK**.
9. **Right Click** on schematic document **BC0001_Devices.SchDoc** and **select Local History >> Storage Manager**.
10. **Select** the **latest two** revisions.
11. **Right Click** on the **latest** revision and **select Compare**.
12. **View** the design differences in the **Differences** panel.

Note: Open the **Differences** panel along the **bottom menu Design Compiler >> Differences**.

CONCLUSION

This **Altium Vault Evaluation Guide** is an introduction to the key concepts of the **Altium Vault**. It focuses on the use of **Altium Vault** from an everyday work in process perspective. The **Altium Vault Implementation Guide** provides all the details you need for initial installation and configuration of **Altium Vault**. Please refer to the **Altium Vault Design and Library Migration Guide** to learn more about bringing **YOUR** ECAD data into **Altium Vault**. Visit Altium.com/documentation for more information on all **Altium** solutions.