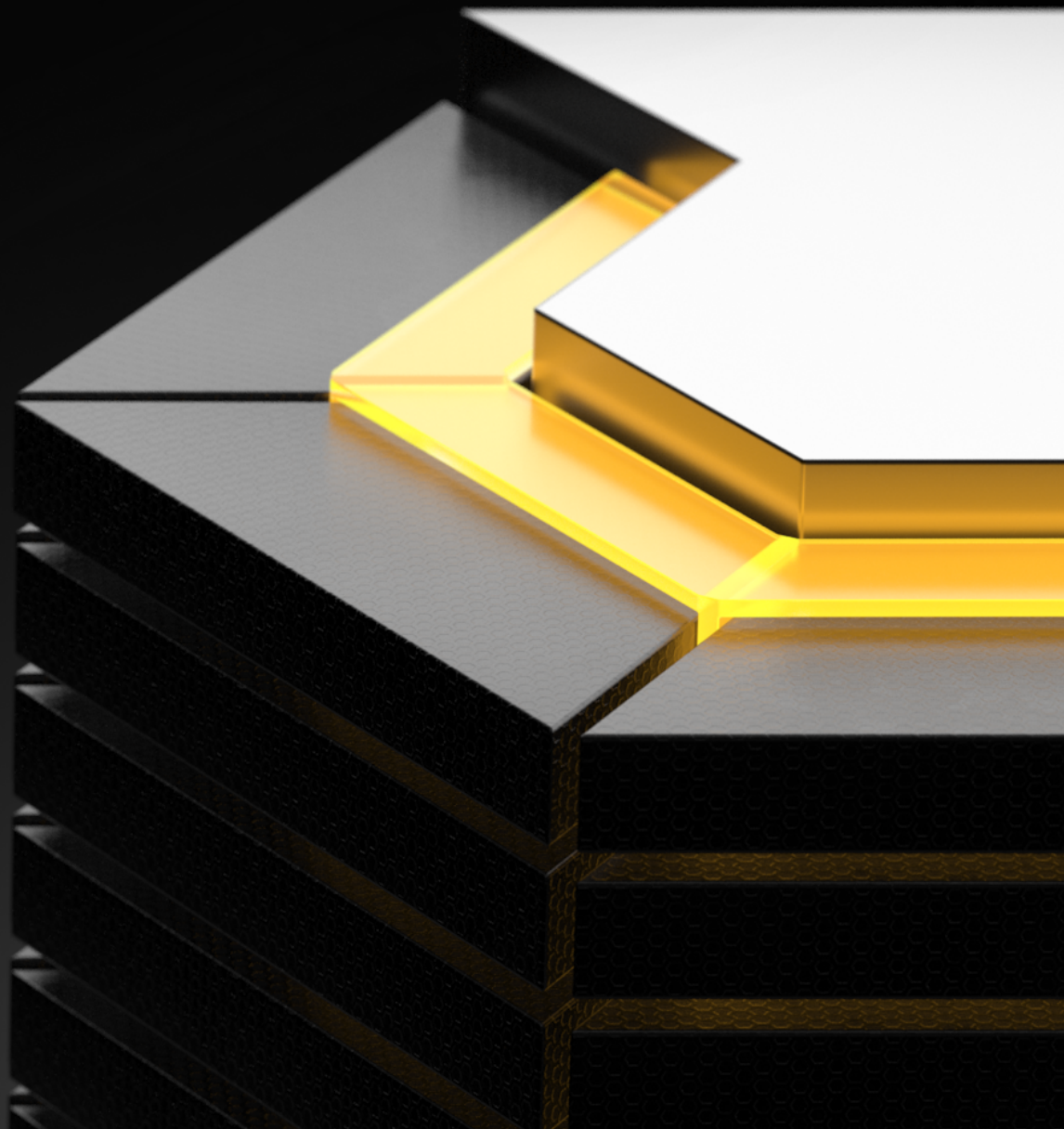


***Altium***<sup>®</sup>

# Altium Vault Design and Library Migration Guide





## TABLE OF CONTENTS

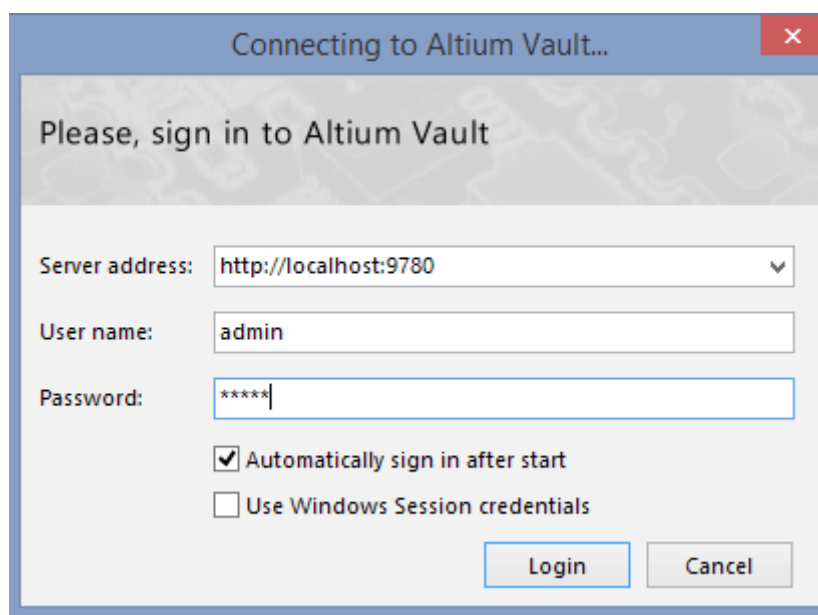
LIBRARIES AND DATA	3
CONNECTING ALTIUM VAULT TO ALTIUM DESIGNER	3
USING THE LIBRARY MIGRATOR TO MIGRATE EXISTING DATA TO ALTIUM VAULT	3
SEARCHING THROUGH MIGRATED DATA	6
APPLYING SUPPLY CHAIN INFORMATION	7
ADDING A COMPONENT TO A DESIGN	9
RELEASING A DESIGN PROJECT	9
UPDATING TO MANAGED COMPONENTS WITH THE ITEM MANAGER	12
CONCLUSION	12

## LIBRARIES AND DATA

This guide provides insight into design project migration into **Altium Vault. Library Management** in **Altium Vault** is inspired by modern data management systems. Rather than simply lists of components, their designs are enriched by **revisions**, **lifecycle management** and **traceability** of relationships between **models**, **components**, and **projects**. Centralization is also improved with newer technologies, along with **usability** and **flexibility**. Rather than being simply a general purpose database, **Altium Vault** is specifically designed for PCB design projects, and is customizable to fit your ECAD needs. Download the example project [here](#).

## CONNECTING ALTIUM VAULT TO ALTIUM DESIGNER

1. Open **Altium Designer**.



2. Go to **DXP >> Sign into Altium Vault**.
3. Fill in your **Server Address**, **User name**, **Password**.

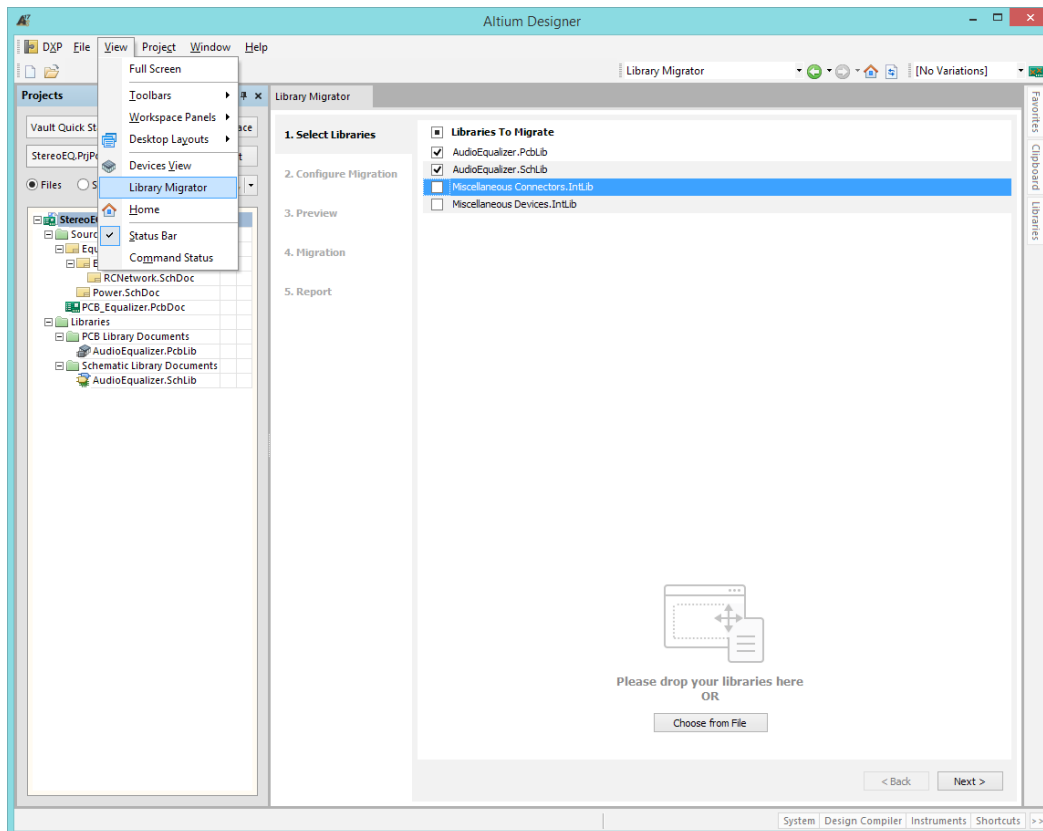
**Note:** If your **Altium Vault** login credentials have been linked to Windows credentials, check the **Use Windows Session credentials** and click **Login**.

4. Click **Login**.

## USING THE LIBRARY MIGRATOR TO MIGRATE EXISTING DATA TO ALTIUM VAULT

You will be working with example project data so that you can get the feel for how to migrate your data from an existing project into the **Altium Vault**. It is recommended that you only migrate libraries from one project at a time using the [Library Migrator](#).

1. Open the provided project **StereoEQ** in **Altium Designer**.



2. Go to **View >> Library Migrator** to open the **Library Migrator**.

3. Select only the **AudioEqualizer.PcbLib** and **AudioEqualizer.SchLib** check boxes.

**Note:** The **PcbLib** will bring over all model information and the **SchLib** will bring over the linked component information.

4. Click **Next**.

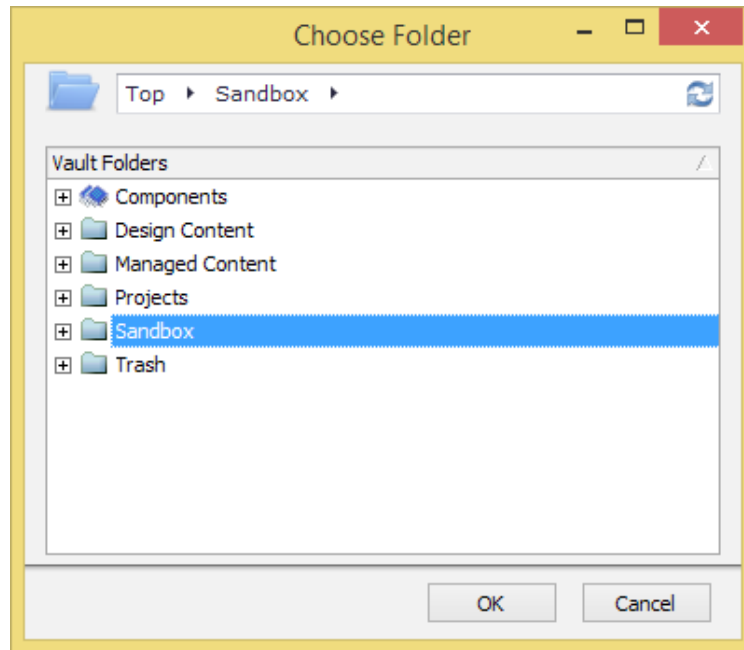
5. Select **AudioEqualizer.PcbLib** under **Source Libraries**.



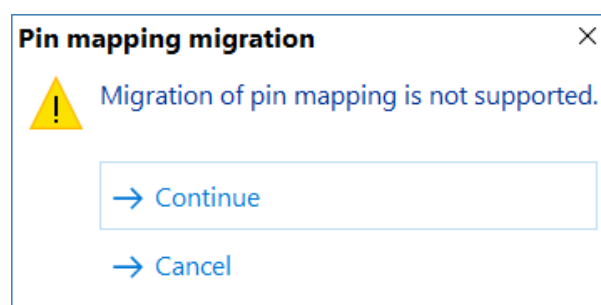
6. In the **Component Settings**, **Folders** section, click the ... button.

7. **Right-click** on the **Sandbox** folder in the **Choose Folder** window.

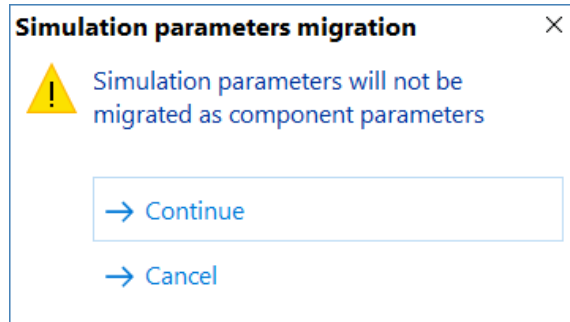
**Note:** If you did not install the sample data during installation, create a generic folder and name it **Sandbox**.



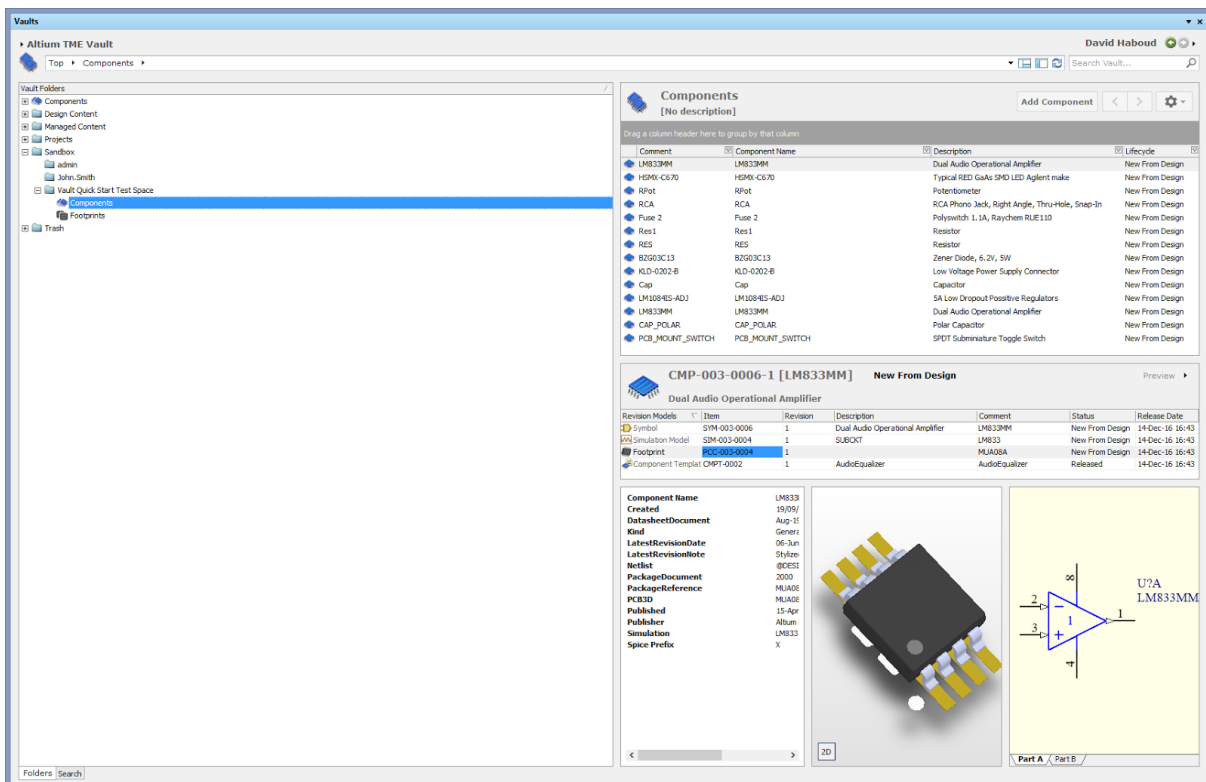
8. Select **Add Subfolder >> Generic Folder**.
9. Set **Folder Name** to **Altium Vault Design and Library Migration Test Space** and click **OK**.
10. **Right-click** on the **Altium Vault Design and Library Migration Test Space** folder in the **Choose Folder** window.
11. Select **Add Subfolder >> Other Folder Type** and name the folder **Footprints**.
12. Select **Footprints** from the dropdown menu of **Folder Type** and click **OK**.
13. Select the **Footprints** folder from the **Choose Folder** menu and click **OK**.
14. Select **AudioEqualizer.SchLib** under **Source Libraries**.
15. In the **Component Settings, Folders** section, click the ... button.
16. **Right-click** on the **Altium Vault Design and Library Migration Test Space** folder in the **Choose Folder** window.
17. Select **Add Subfolder >> Other Folder Type** and name the folder **Components**.
18. Select **Components** from the dropdown menu of **Folder Type** and click **OK**.
19. Select the **Components** folder from the **Choose Folder** menu and click **OK**.
20. Click **Next** to prepare for migration and preview the data you are migrating.
21. Click **Migrate** to begin **Migration**.



23. Click **Continue** from **Simulation Parameters migration warning**.



24. Click **Show in Altium Vault Explorer** once complete to view data.



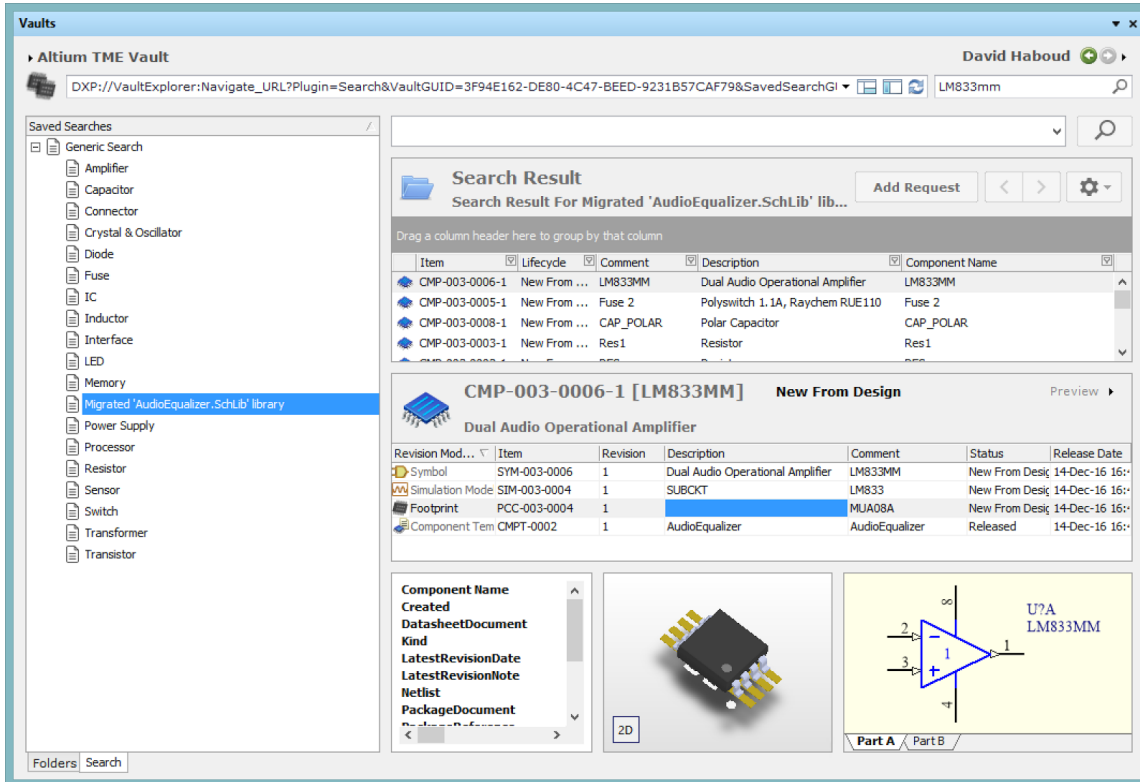
22. Click **Continue** from **Pin mapping migration warning**.

25. **Repeat** for **ANY library!**

## SEARCHING THROUGH MIGRATED DATA

Once you have data in your **Altium Vault**. You can 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. You can now search for a specific component, Dual Audio Operational Amplifier, found in our example design. You can search for items with the traditional search, but migrated libraries have their own saved search query.

1. Go to **DXP >> Altium Vault Explorer**.
2. Click the **Search tab** at the bottom of the **Altium Vault Explorer**.
3. Double click on the saved search named **Migrated 'AudioEqualizer.SchLib' library**.



4. Left click and drag the **Comment** column header to the dark grey header grouping section above it.

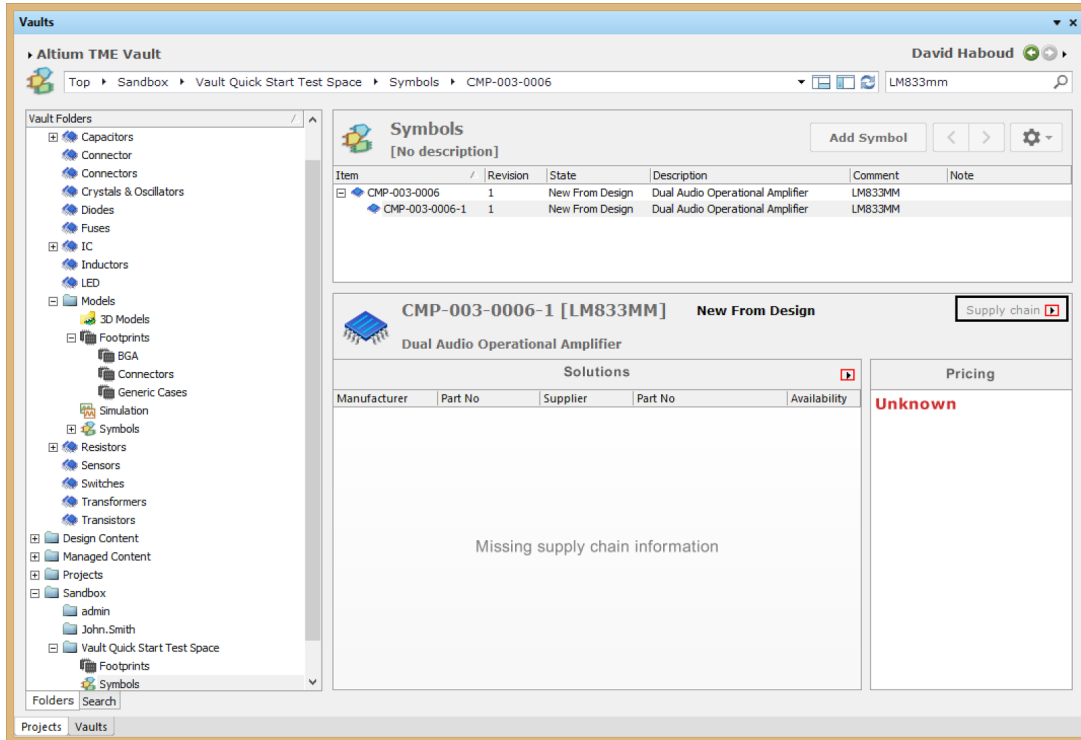
5. Scan the results for **LM833MM**.

## APPLYING SUPPLY CHAIN INFORMATION

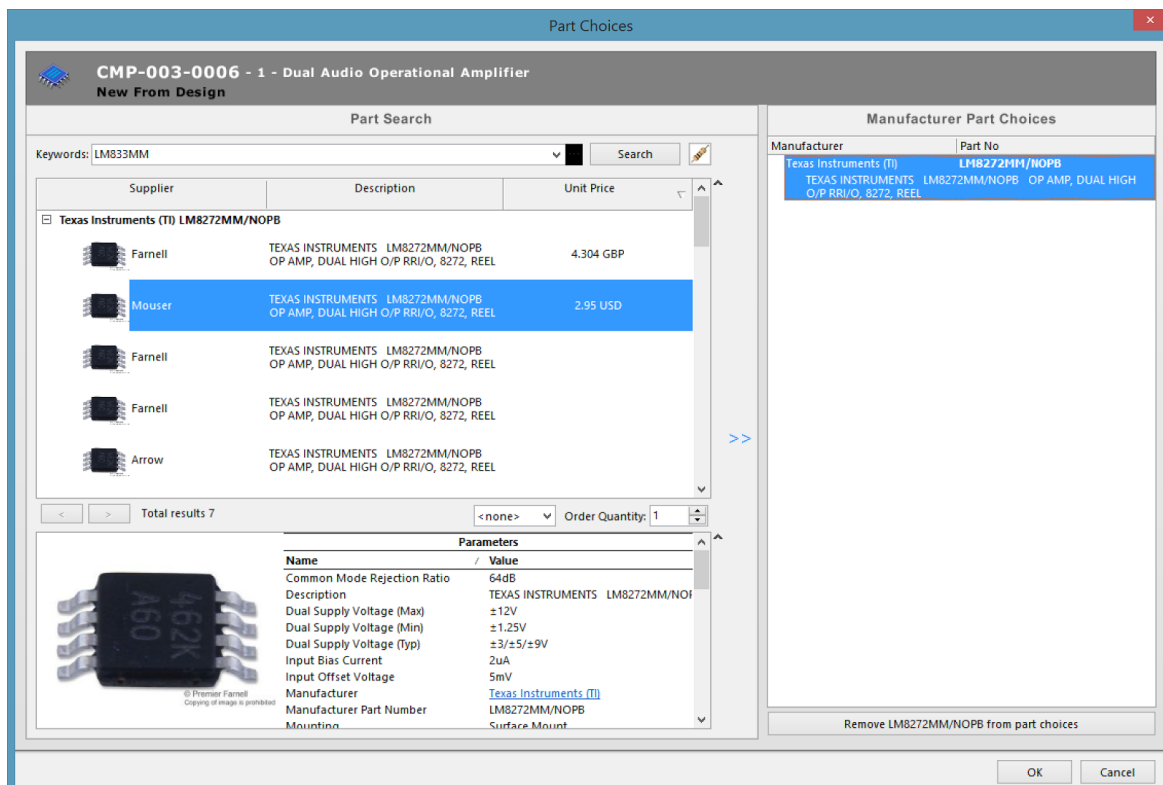
Now that you have identified a component that you can use in our design, you can see some supply chain data. The bottom right corner of **Vault Explorer** shows information on selected design assets. The top right corner of the information sections names the type of information being displayed. If your libraries are linked to supply chain information in **Altium Designer**, the library migrator will stay connected to the proper supply chain information. If your libraries do not contain supply chain information, the supply chain information can be applied with **Altium Vault**.

1. Click the **Black Triangle** at the top right corner of the information section.

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



2. Select **Supply chain**.
3. Click the **Black Triangle >> Create Part Choice List...** at the top right corner of the **Solutions** section.
4. Sort by **Unit Price** by **double clicking** on the column header.
5. Select your desired **part choice** and click >> to add it to the **Manufacturer Part Choices** list.
6. Click **OK**.

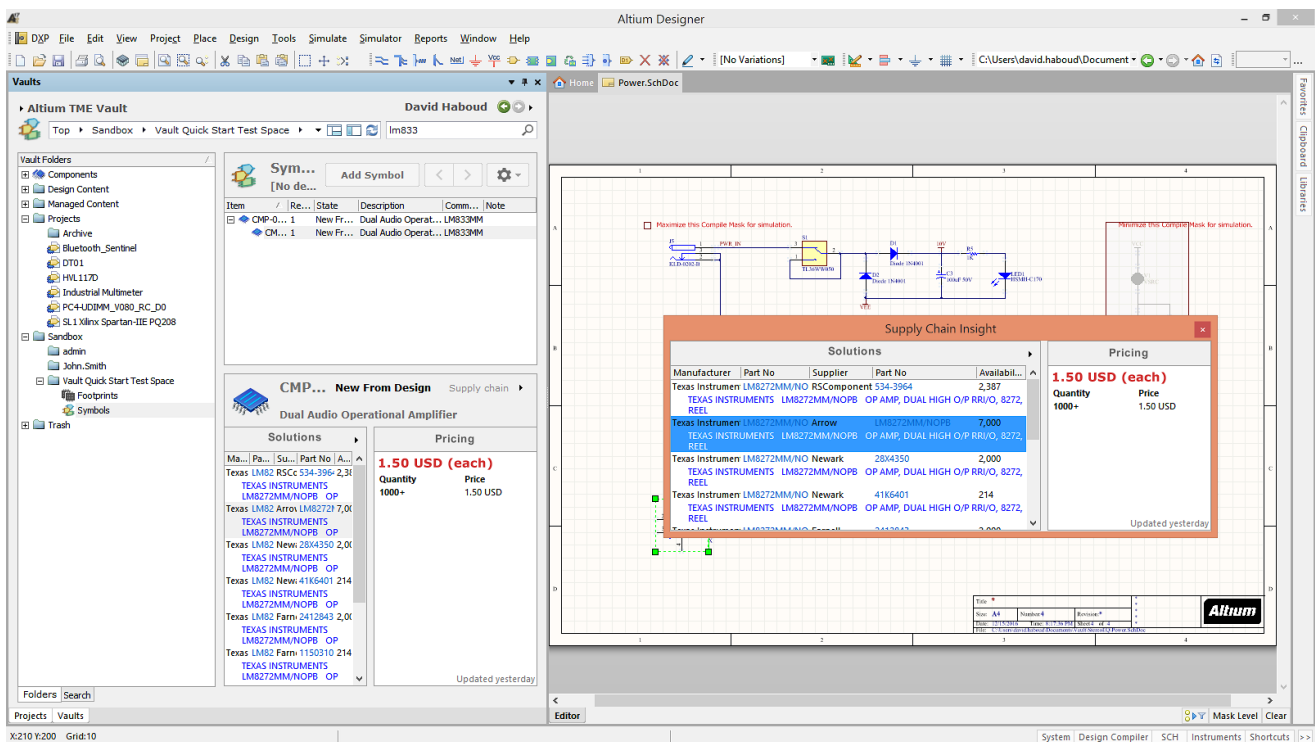




## ADDING A COMPONENT TO A DESIGN

Now that the component has the relevant information, you need to use it in a design. Using an **Altium Vault** component is simple. Dock the **Vault Explorer** panel by dragging it to the desired area of **Altium Designer**. Look at using the component **LM833MM** in a schematic.

1. **Open** a **schematic** or **pcb** document from the example project **StereoEQ**.
2. **Select** the **Altium Vaulted component revision** you would like to use in your design.
3. **Drag** the **component revision** you would like to use for the design to the **open document**.
4. **Click** on the location you would like to **place** the **component**.
5. **Save** the document.

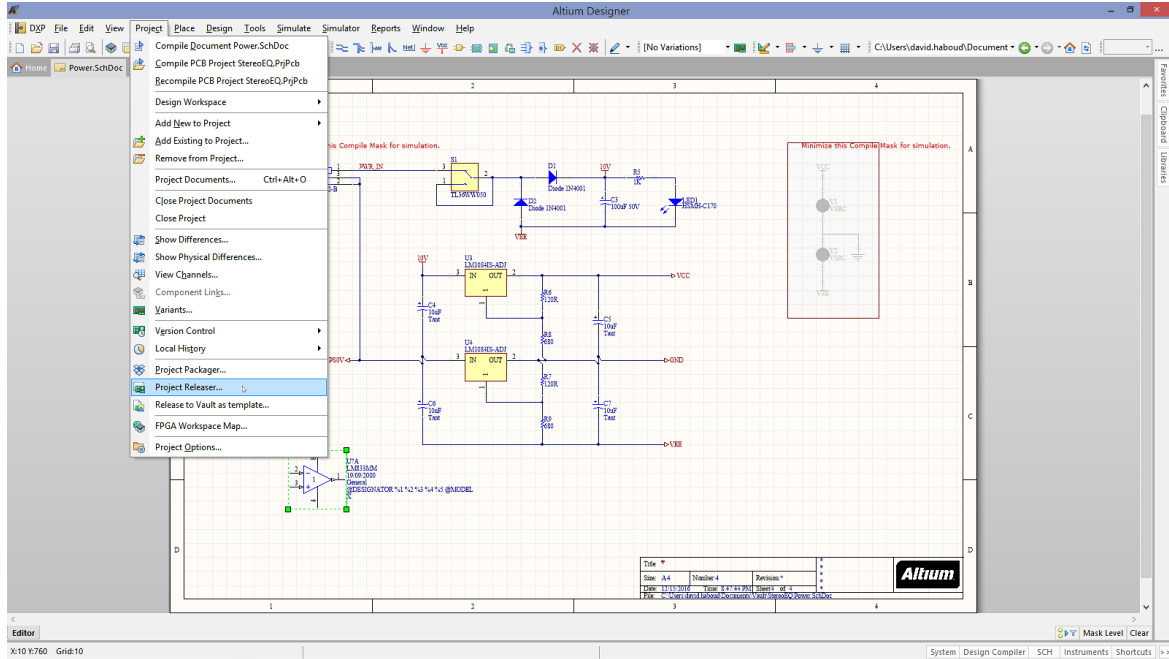


## RELEASING A DESIGN PROJECT

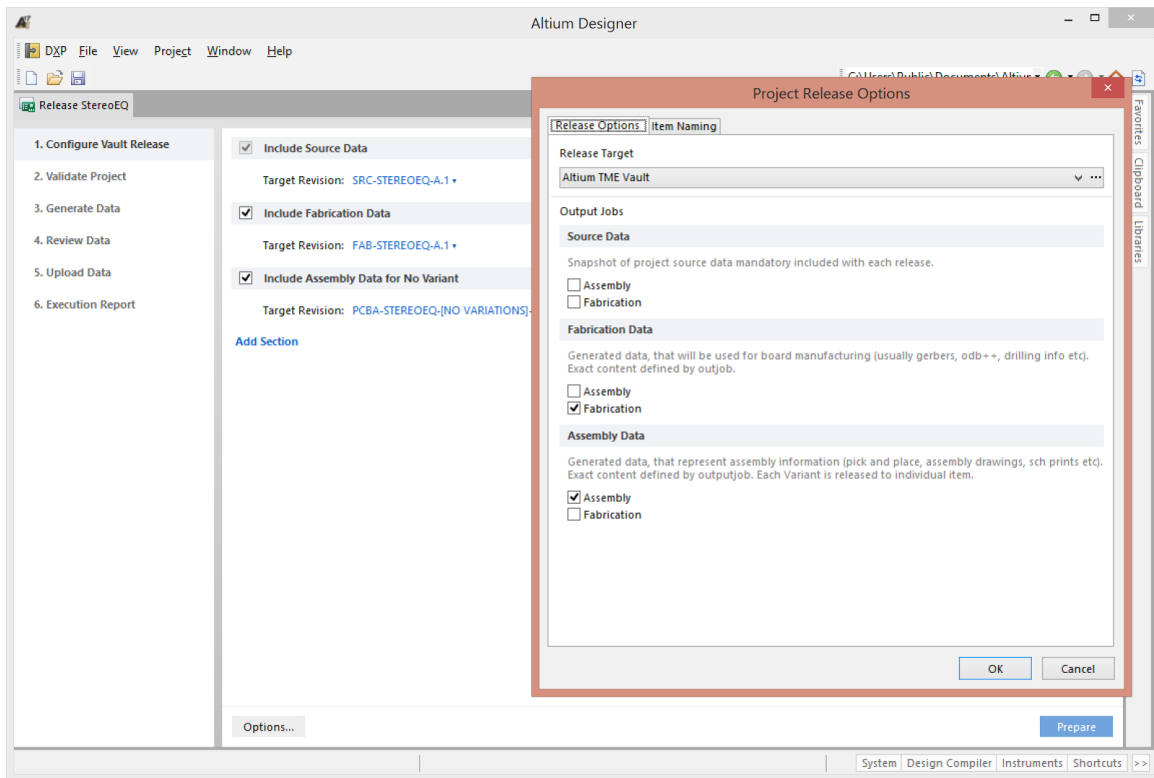
Now take a look at releasing the example design into **Altium Vault**. Release the example project **StereoEQ** into **Altium Vault**.

1. Go to menu **Project >> Project Releaser...**

# ALTIUM VAULT DESIGN AND LIBRARY MIGRATION GUIDE



2. Click **Options**.



3. In the **Project Release Options**, you can decide which **Outjob Files** to link to the different **Output Data types**.

**Note:** Follow the following link for a more in-depth understanding of **Outjob Files** and **Altium Vault**.

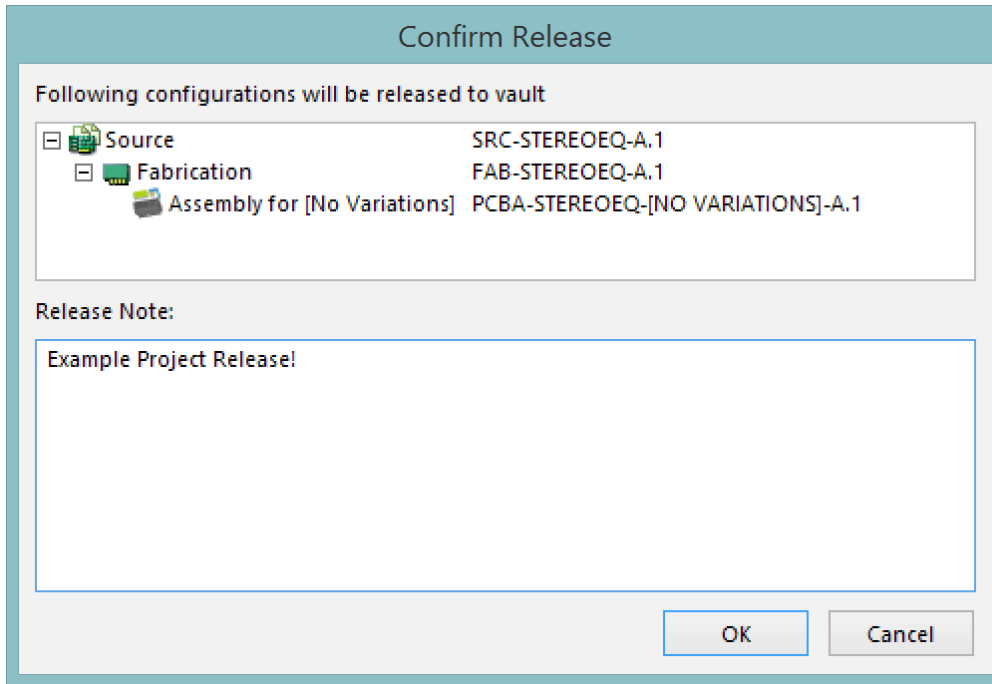
4. Click **OK**.

5. Click **Prepare**.

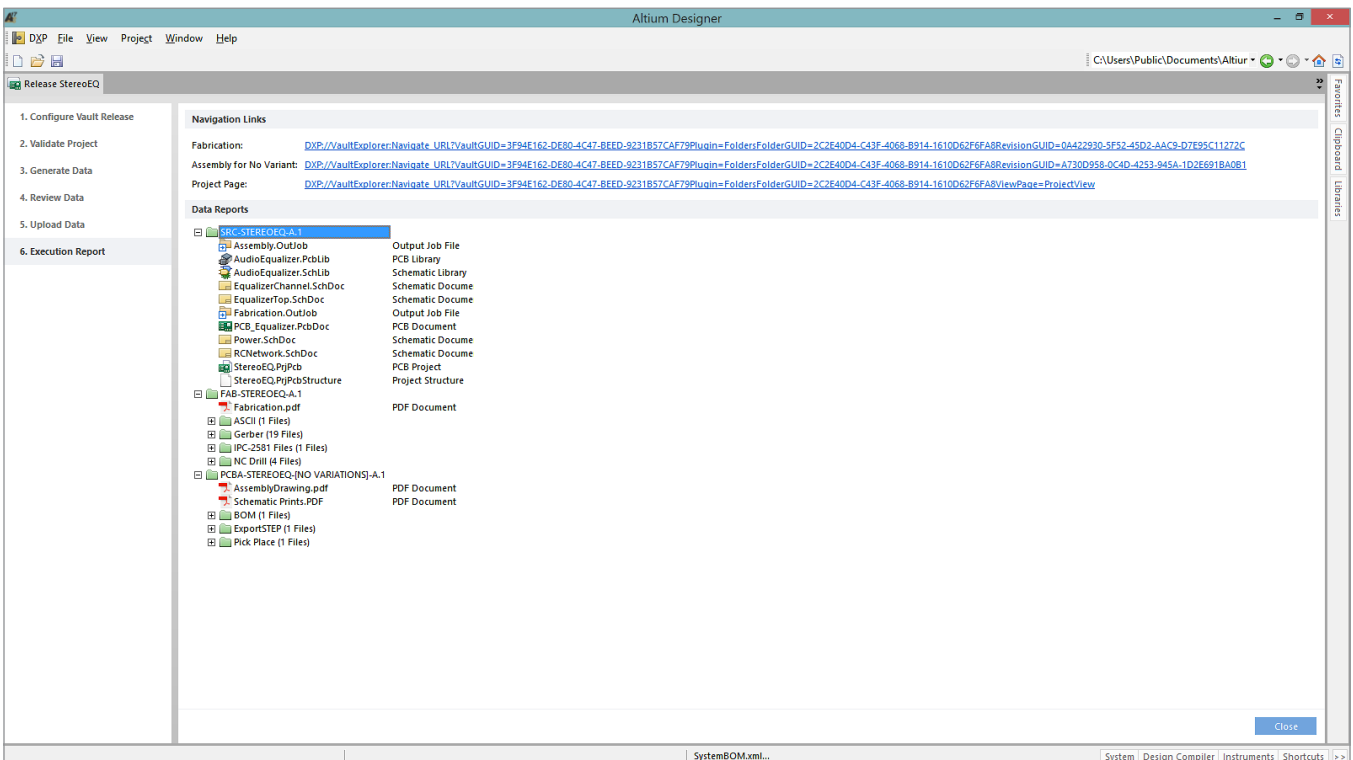
**Note:** The time for preparation is dependant on the number and size of files being generated.

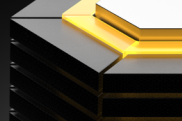


6. Click **Release**.
7. Set **Release Note**: to **Example Project Release!**



8. Click **OK**.
9. Click on the **Fabrication** link to view the project in **Altium Vault**.





## UPDATING TO MANAGED COMPONENTS WITH THE ITEM MANAGER

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 untenable 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**. Update the example project, **StereoEQ**, components with the **Altium Vaulted Managed Components**.

1. Go to the **Tools** menu from an active schematic document and click **Tools » Item Manager**.
2. 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.

3. Click **Options...** to open the **Item Manager Options** dialog.

**Note:** The configurable **Rules** determine which component **Parameters** in the **active** (local) design are matched to the **Parameters** of all **Managed Components** in the selected **Source Altium Vault**.

4. Click **OK**.
5. Click **Automatch**.

**Note:** The subsequent **Automatching Items dialog** will indicate successful matches in **green text** and include a **reference** to the name of the **successful Rule**.

6. Select the **range** of listed components you wish to change and then the **Apply ECO** option from the **ECO...** button menu.

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.

## CONCLUSION

This **Altium Vault Design and Library Migration Guide** is an introduction to existing design project migration to **Altium Vault**. Please refer to the **Altium Vault User Guide** for a more in depth view of what **Altium Vault** has to offer. Visit [Altium.com/documentation](https://www.altium.com/documentation) for more information on all **Altium** solutions.