



Altium Designer

Advanced Training with Altium 365

ActiveBOM

Altium
TRAINING





Software, documentation and related materials:

Copyright © 2024 Altium LLC

All rights reserved. You are permitted to use this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document are made. Unauthorized duplication, in the whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium LLC. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties.

TRADEMARKS

ACTIVEBOM®, ActiveRoute®, A365™, Altium 365®, Altium Concord™, Altium Concord Pro™, Altium Designer®, AD™, Altium NEXUS®, Altium OnTrack™, Altium Vault®, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Common Parts Library™, Concord™, Concord Pro®, Draftsman®, Dream, Design, Deliver®, DXP™, Easytrax®, EE Concierge®, Fearless HDI™, Geppetto®, Gumstix®, Learn, Connect, Get Inspired™, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, OnTrack™, Overo®, P-CAD®, PCBWORKS®, PDN Analyzer™, Protel®, Situs®, SmartParts™, Upverter®, X2®, XSignals® and their respective logos are trademarks or registered trademarks of Altium LLC or its affiliated companies. All other registered or unregistered trademarks referenced herein are the property of their respective owners and no trademark rights to the same are claimed.





Table of Contents

ActiveBOM	3
1 Purpose	3
2 Shortcuts	3
3 Preparation	4
4 New ActiveBOM Document	5
5 Modifying the ActiveBOM View	6
5.1 General	6
5.2 ActiveBOM and Properties Panel	9
5.3 BOM Set	11
6 Managed Components	12
6.1 Update Revision Status	12
6.2 Solutions	13
6.3 Ranking for Solutions	15
7 Unmanaged Components – Update Example	16
8 Obsolete Components	17





ActiveBOM

1 Purpose

In this exercise, you will learn how to work with the ActiveBOM, a powerful Bill of Materials management editor. It brings comprehensive BOM management tools together with Altium powerful part-information aggregation technologies, helping you manage the component selection challenge.

Good component selection underlies the success of every electronic product. Components must meet technical, cost, availability, and lead time requirements, as well as considerations for assembly and testing. Choosing the wrong part can affect the product cost, delivery, and market success. With today's fast-paced design cycles, accurate component information is vital. Altium tools provide designers with detailed, real-time component data, supporting better decisions.

The highly compressed design » test » manufacture product development cycle means the designers have very little room for error while selecting the components. Altium recognizes the critical role that component selection plays in the design process and develops software technologies, products, and teams that help deliver easy-to-use, detailed, and accurate component data and component selection systems, directly into the designer's workspace.

The BOM is a critical document in the design process, used early for cost estimation and finalized for manufacturing. It includes not just electronic components but also all other items needed for assembly, making it essential to the entire production process. Consequently, the BOM is much more than a list of the electronic components fitted to the board - it must also detail every screw, standoff, sticker, heatsink, and tube of glue required to finish the sub-assembly that the PCB becomes a part of.

2 Shortcuts


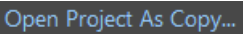

Shortcuts used when working with ActiveBOM

Crtl+S	Save Document
--------	---------------





3 Preparation

1. Close all existing projects and documents.
2. Next, create a copy of the Training Project: `ActiveBOM`.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Enable the folder view button .
5. Navigate to the predefined Training Project `ActiveBOM (Top\Projects\Altium Designer Advanced Training Course\...)`.
6. Select **Open Project as Copy...** .
7. In the new dialog *Create Project Copy*:
 - a) Add your name to the project name: `ActiveBOM - [Your Name]`.
 - b) Add a description: `Altium Advanced Training - [Your name]`.
 - c) Open the *Advanced* section.
 - d) Select the **Ellipsis Button**  from the *Folder* configuration to open the *Choose Folder* dialog.
 - i) Select the folder with your name: `Project\For Attendees\[Your name]`.
 - ii) Select **OK**.
 - e) Change the **Local Storage** path if needed.
 - f) Select **OK** to create the copy.
8. Wait until Altium Designer creates the copy of the project and opened the Project for you in the *Projects* panel, this may take up to 1 minute.

Hint: For details how to copy the predefined training project, see module *03 Getting started - Opening a Project*.

Note: The FMU Project from this exercise is modified to show specific situations for the training.


Caution: For this module, a connection to the Internet and a SignIn to Altium Live is needed. If, because of Security Reasons, security modules like Proxy or Firewalls block specific Ports or URL, the service of the ActiveBOM may not be possible.





4 New ActiveBOM Document

9. In the *Projects* panel, double click on the file `1_reference.SchDoc` and `Active_BOM.PcbDoc` to open the files in the workspace.
10. You can add the ActiveBOM document using one of the methods below:
 - a) Use the command **File » New » ActiveBOM**, as seen in Figure 1.
 - b) Right-click on the project within the *Projects* panel and select **Add New to Project » ActiveBOM**.

Hint: In the *Preferences*, you can configure if an ActiveBOM document is automatically added to the Altium Designer project without an ActiveBom or not. By selecting the command **+ Create**  **+ Create**, the ActiveBOM Document is added. If you see the suggestion to add an ActiveBOM and you disagree to add the file, select the ActiveBOM, then right-click and remove or delete the File.

Note: The ActiveBOM document has the same name as the project. You can re-name the document if you wish.

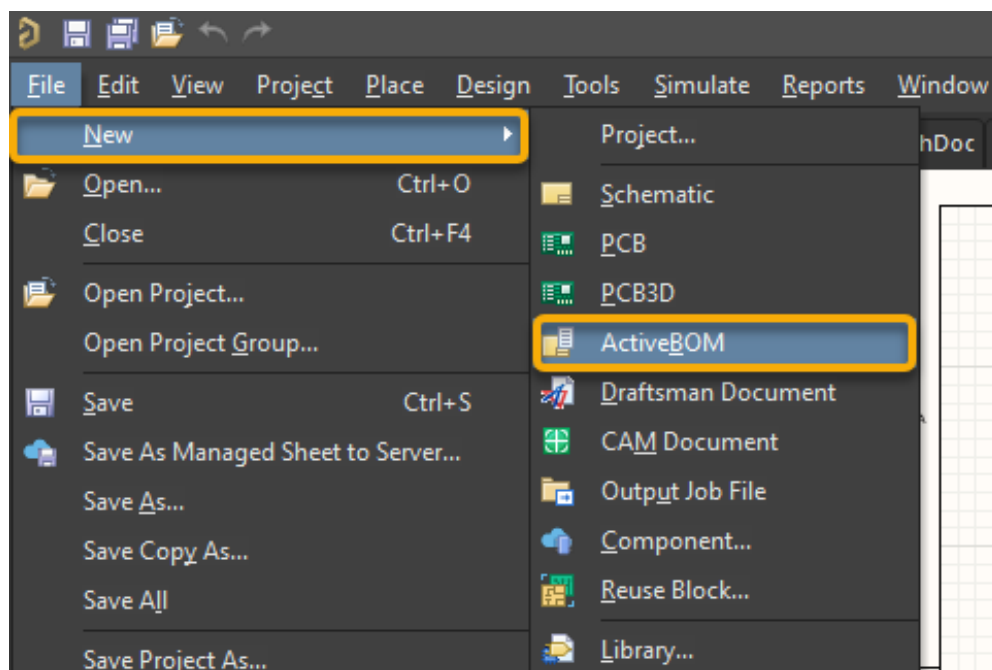


Figure 1. Adding an ActiveBOM document to the project

Note: Unlike other types of documents, an ActiveBOM document can only be created once within a project.

11. The ActiveBOM will list all the components from the project and check the status of the components.

Note: This exercise provides component information that was current at the time the training manual was created. Since the market for components is very dynamic, the component information, such as Revision State, Revision Status, Price, or Lifecycle that you now see when you complete the exercise for the ActiveBOM, may differ from the information in the training documentation.







5 Modifying the ActiveBOM View

5.1 General

12. You can configure the ActiveBOM view to suite specific circumstances, as seen in Figure 3.

- One way you can modify the ActiveBOM view is by using the pre-configured view buttons: Flat, Base, or Consolidated . The Flat view shows individual components listed in each row. The Base view shows each unique part number listed in each row. The consolidated view is similar to the Base view with additional columns showing variant information, if used.
- Another way to modify the ActiveBOM view is to right-click on any column header and click on **Select Columns**. In the new dialog *Select Columns*, use the eye icon  to show or hide specific parameter.
- The ActiveBOM table can also be grouped into different categories using the **Group by** button.

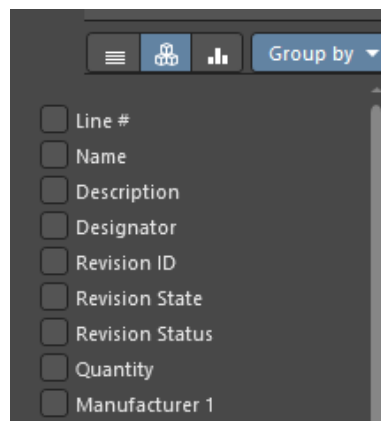
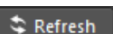


Figure 2. Grouping the ActiveBOM table into different categories

- You can change the column positions using a left-click and drag and drop.
- Click in the Column Header Name cell to change the sorting of a column.
- Click on  **Refresh** to refresh the list information.

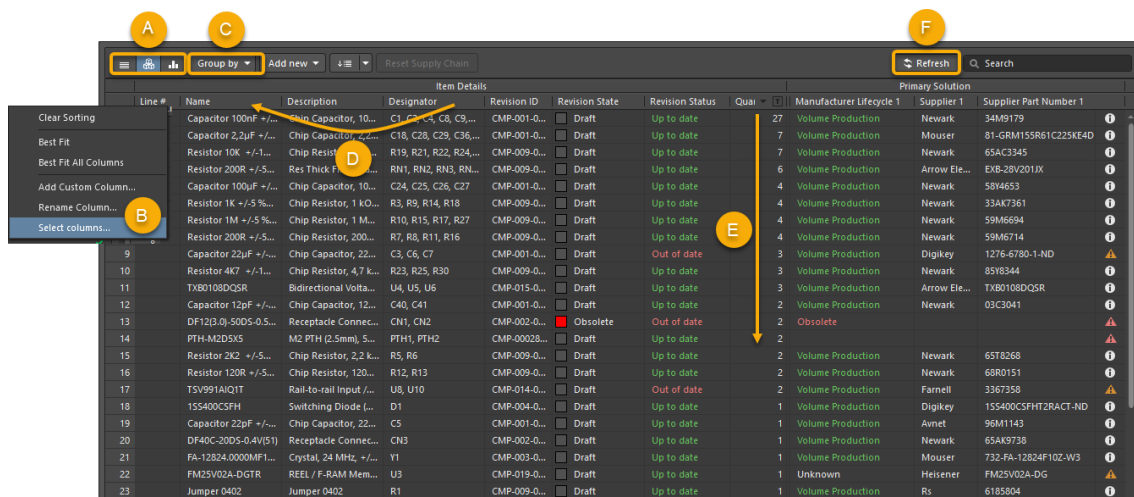


Figure 3. ActiveBOM - GUI Overview





13. Click the consolidated view button.

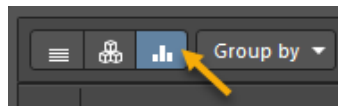


Figure 4. Consolidated view button

14. Right-click on one of the Column Header Names to open the *Select Columns* dialog. The dialog allows you to activate or deactivate columns for your ActiveBOM. Scroll down the list to find the *Fitted* parameter. Activate the **Eye Icon** to add the parameter to the view. Click the up or down button to move heading up in the list order. Click the **OK** button. The Fitted column will only be visible when the Consolidated list is selected, Figure 5 below.

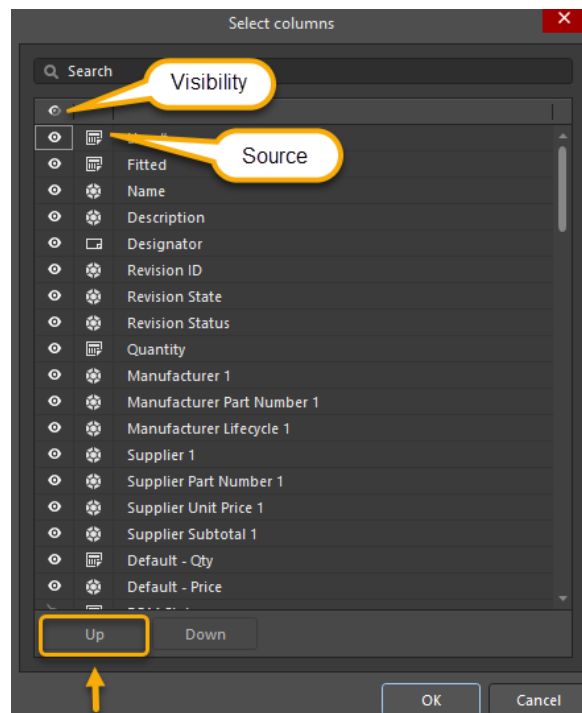

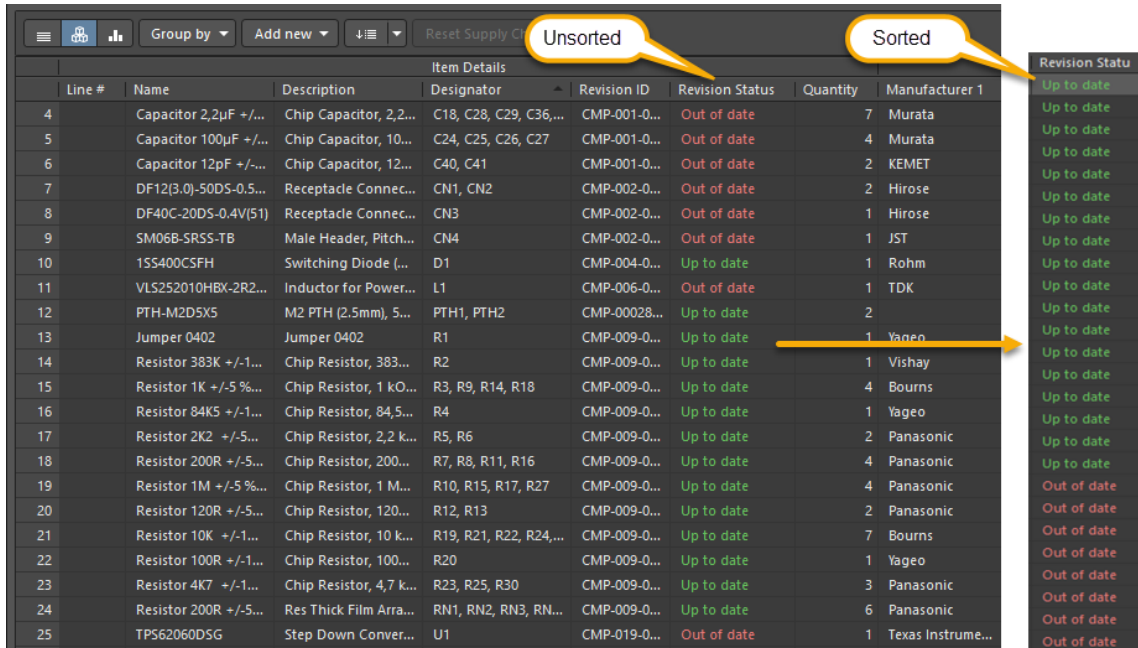


Figure 5. Select a new column for the BOM



15. Click the Base view button .
16. A design could have components from different sources, some of them are managed, others are not. Click on column header **Revision State** to sort the items by Revision State, as seen in Figure 6.






Line #	Name	Description	Designator	Revision ID	Revision Status	Quantity	Manufacturer 1	Revision Status
4	Capacitor 2,2µF +/-...	Chip Capacitor, 2,2...	C18, C28, C29, C36,...	CMP-001-0...	Out of date	7	Murata	Up to date
5	Capacitor 100µF +/-...	Chip Capacitor, 10...	C24, C25, C26, C27	CMP-001-0...	Out of date	4	Murata	Up to date
6	Capacitor 12pF +/-...	Chip Capacitor, 12...	C40, C41	CMP-001-0...	Out of date	2	KEMET	Up to date
7	DF12(3.0)-50DS-0.5...	Receptacle Connec...	CN1, CN2	CMP-002-0...	Out of date	2	Hirose	Up to date
8	DF40C-20DS-0.4V(51)	Receptacle Connec...	CN3	CMP-002-0...	Out of date	1	Hirose	Up to date
9	SM06B-SRSS-TB	Male Header, Pitch...	CN4	CMP-002-0...	Out of date	1	JST	Up to date
10	1SS400CSFH	Switching Diode (...)	D1	CMP-004-0...	Up to date	1	Rohm	Up to date
11	VLS252010HBX-2R2...	Inductor for Power...	L1	CMP-006-0...	Out of date	1	TDK	Up to date
12	PTH-M2DSX5	M2 PTH (2.5mm), 5...	PTH1, PTH2	CMP-00028...	Up to date	2		Up to date
13	Jumper 0402	Jumper 0402	R1	CMP-009-0...	Up to date	1	Yageo	Up to date
14	Resistor 383K +/-1...	Chip Resistor, 383...	R2	CMP-009-0...	Up to date	1	Vishay	Up to date
15	Resistor 1K +/-5 %...	Chip Resistor, 1 kO...	R3, R9, R14, R18	CMP-009-0...	Up to date	4	Bourns	Up to date
16	Resistor 84K5 +/-1...	Chip Resistor, 84,5...	R4	CMP-009-0...	Up to date	1	Yageo	Up to date
17	Resistor 2K2 +/-5...	Chip Resistor, 2,2 k...	R5, R6	CMP-009-0...	Up to date	2	Panasonic	Up to date
18	Resistor 200R +/-5...	Chip Resistor, 200...	R7, R8, R11, R16	CMP-009-0...	Up to date	4	Panasonic	Up to date
19	Resistor 1M +/-5 %...	Chip Resistor, 1 M...	R10, R15, R17, R27	CMP-009-0...	Up to date	4	Panasonic	Out of date
20	Resistor 120R +/-5...	Chip Resistor, 120...	R12, R13	CMP-009-0...	Up to date	2	Panasonic	Out of date
21	Resistor 10K +/-1...	Chip Resistor, 10 k...	R19, R21, R22, R24,...	CMP-009-0...	Up to date	7	Bourns	Out of date
22	Resistor 100R +/-1...	Chip Resistor, 100...	R20	CMP-009-0...	Up to date	1	Yageo	Out of date
23	Resistor 4K7 +/-1...	Chip Resistor, 4,7 k...	R23, R25, R30	CMP-009-0...	Up to date	3	Panasonic	Out of date
24	Resistor 200R +/-5...	Res Thick Film Arra...	RN1, RN2, RN3, RN...	CMP-009-0...	Up to date	6	Panasonic	Out of date
25	TP562060DSG	Step Down Conver...	U1	CMP-019-0...	Out of date	1	Texas Instrume...	Out of date

Figure 6. ActiveBOM with different Revision States

Hint: For more information on managed and unmanaged components, see Sections 6 and 7.



5.2 ActiveBOM and Properties Panel

17. Using the Flat View   : Sort the ActiveBOM by Designator. Select C1, which the first Element in the list.
18. Click the **Panels** button and select *Properties*.
19. In the *Properties* panel, as shown in Figure 7, you can see and change the following parameters:

Properties

Document options

Search

General Columns

General

Production Quantity: 1 (a)

Currency: USD

	Actual	Target
Price per Board	27.15 USD	0 USD
Order Price	27.32 USD	0 USD

Supply Chain

Solutions per Item: 1 (b)

Suppliers per Solution: 1 (c)

Solutions

☒ Exclude Invalid SPNs (d)

Favorite Suppliers List: Edit... (e)

Manufacturer Link: Edit... (e)

BOM Items

☐ Show Not Fitted

Designator Grouping: R1, R2, R3 (f)

Component Grouping: Edit...

BOM Sets

Name

Add Update

BOM Checks




Violation Type	Items
No suppliers (h)	1
Component revision state is restricted from use	2
No solutions	2
Component revision is out of date	16

Figure 7. Properties Panel for ActiveBOM

- a) The production quantity with the Price per Board and Order Price, based on the Components with the price information.
- b) The number of solutions per item. At the moment, the CAP has only one allowed solution. Change the number of allowed solutions for the ActiveBOM to **2**. Select C1 again and check the listed solutions and the hatching that disappear if you increase the allowed solutions.









- c) The Number of Suppliers for a Solution. At the moment, we can see several Suppliers for the CAP but only the first one is active, the others are hatched. Change the Number of allowed Suppliers for the ActiveBOM to **2**. Select C1 again. Now, the suppliers at position 1 and 2 are allowed and listed in the ActiveBOM.
 - d) The Favorite Supplier List. In the dialog that opens, you can activate or deactivate suppliers for the ActiveBOM.
 - e) Manufacturer Link. The dialog is used to define which component parameters hold the Manufacturer Name and the Manufacturer Part Number. The ActiveBOM can use this information to query the Altium Parts Provider to identify suitable supply chain solutions. This option is available if you have Not Managed components.
 - f) BOM Items. This allows you to define the Grouping for the components.
 - g) BOM Sets. More information regarding BOM Sets is covered later in this document (Section 5.3).
 - h) The result of the BOM Check. Depending on the current Component status, you can see several status levels for the components. Click on the Filter Icon  to filter only components that do not have a Supplier Information. Click on the Filter Icon  again to remove the filter.
20. Use the Gear  at the lower right to open the settings for the BOM Check.
21. Close the BOM Check with **Cancel** to avoid that you changed the settings by accident.





5.3 BOM Set

22. Next, you will add BOM sets. BOM sets allow you to save specific layout configurations for your BOM, as shown in Figure 8 and Figure 9.

- Open the *Properties* panel.
- In the *Properties* panel, in section *BOM Sets*, click the **Add** button twice to add two new BOM Sets.
- Rename the BOM Sets as follows:
BOM set 1 – Base
BOM set 2 – Flat
- Select BOM set 1 – Base, change the configuration to a base View   , select the **Update** button, click on the **Eye Icon**  to make the BOM set 1 – Base the active view.
- Feel free to move some columns and change the visibility of the columns, if so click **Update** to save the new BOM Set Layout.

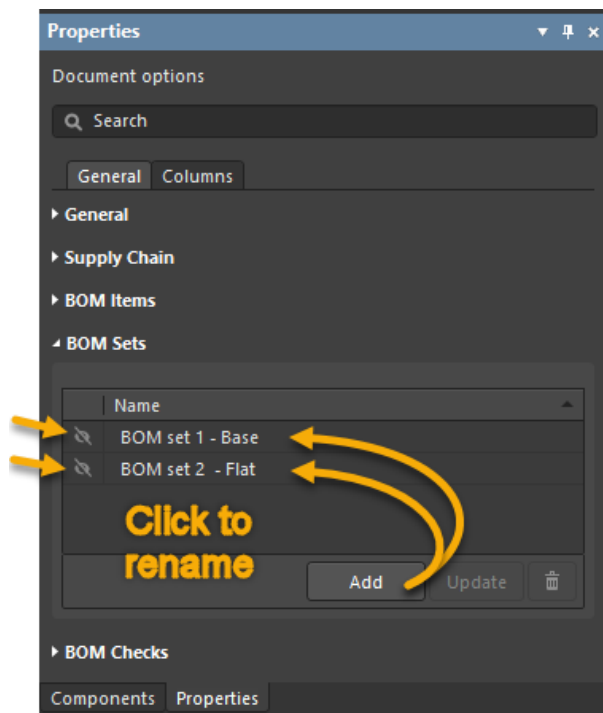


Figure 8. Properties Panel - ActiveBOM with two BOM Sets

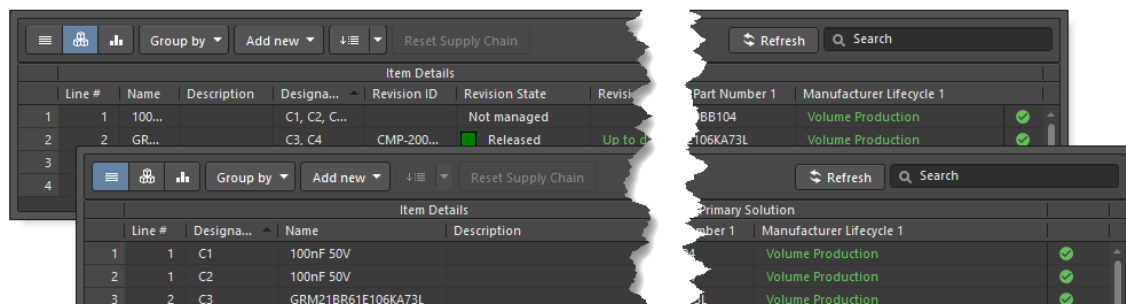


Figure 9. Two BOM Sets with different Layout Configurations





6 Managed Components

Design components stored in an Altium Managed Content Server are called managed components. These components offer revision history, lifecycle management, and real-time connection to the supply chain. A Part Choice links the design component to the manufacturer part, allowing Altium Cloud Services to provide detailed and current supply chain information.

6.1 Update Revision Status

23. Managed Components can have different statuses. Click on column header **Revision Status** to sort the Components by Revision Status. There are various Groups in the design.
24. Based on your Component Information and the Managed Content server, you can see Revision Statuses such as Up to date, Out of date, and so on.
25. Next, you will update C3, C6, and C7 to the latest revision state.
 - a) Click on column header **Designator** to sort the Items by Designator.
 - b) Select C3, C6 and C7.
 - c) **Right click** to select **Operations » Update to the Latest Revision**, as seen in Figure 10 below.
26. You will NOT do the next step during the training because it could modify several Symbols and Footprints in the design and that could be a big redesign step. Feel free to update all components to the latest revision AFTER the training.
This example has several Items that are out of date. You can update them one by one or select all Items and update them with the **Right Click » Operations » Update to the Latest Revision** command.

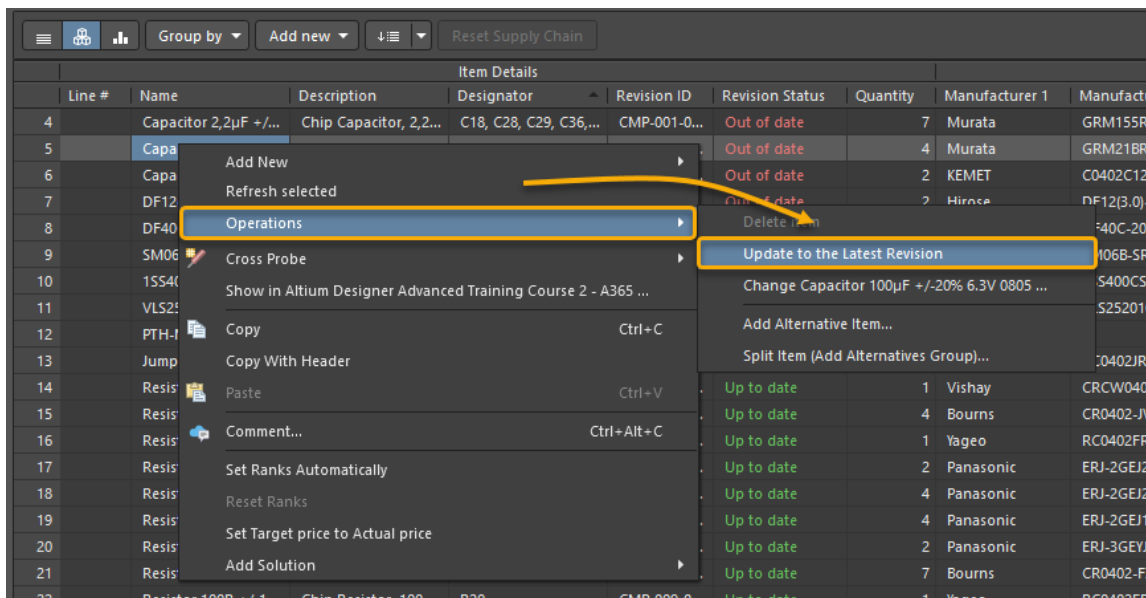


Figure 10. Components out of date - Update to latest revision



6.2 Solutions

27. With the ActiveBOM selected, ensure the view is set to **Flat**

28. Select component C1. At the bottom of the ActiveBOM, you will see the current solutions (Yageo, Kyocera, Kemet...) for the component. Supply Chain Information such as price, stock, or Manufacturer Lifecycle for different suppliers are listed as shown in Figure 11.

- Click on the **Supplier Part Number Field** from the first Supplier to open the Web page for the CAP.
- Click on the **Manufacturer Part Field** to open the web page from Octopart for the CAP. You may be redirected to the manufacturer's website.
- Click on the button **Datasheet** to open the Datasheet.

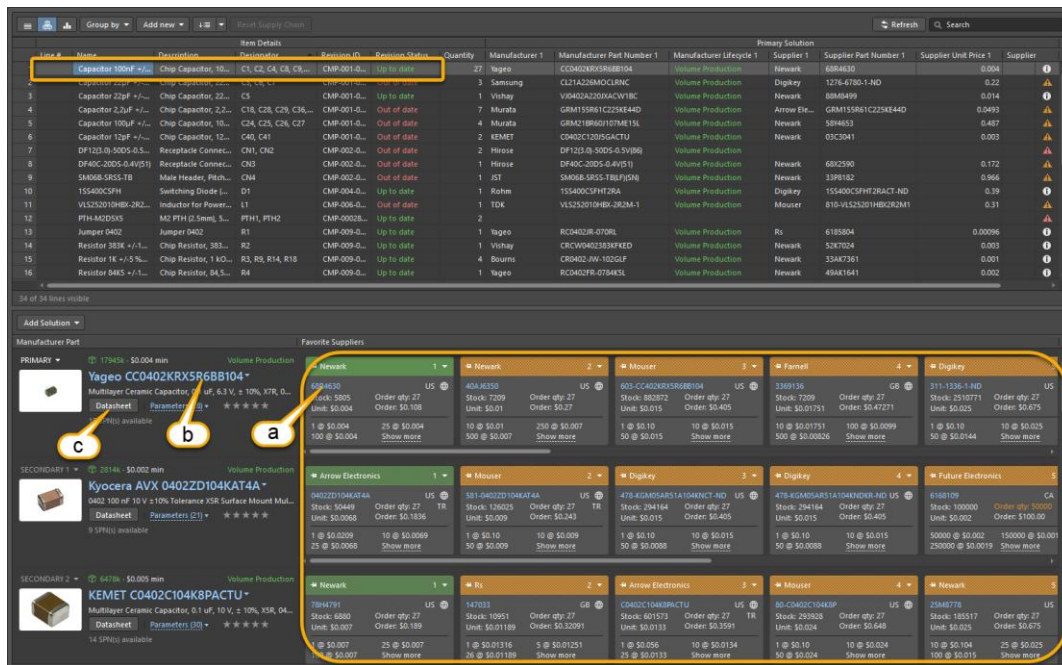


Figure 11. Example Solution



29. If Yageo is not your preferred Supplier for the CAP C1 (as seen in Figure 12a below):
- Scroll down to Kyocera.
 - Select the 5-Stars, this will define the ranking, meaning that a parts supplier with the Kyocera part will always top choice of the solutions.

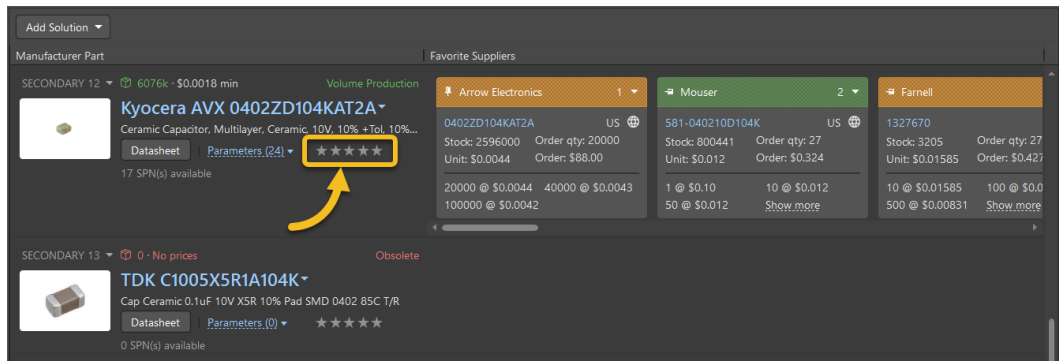


Figure 12a. Selecting the Preferred Supplier

- Click at the top of Arrow Electronics, might be necessary to scroll to the right to see Arrow.
- Click on the **Position Number** and select, for example, **1**, as shown in Figure 12b. By selecting 1, you move Arrow Electronics to Position 1, automatically the first Supplier (for example, Arrow Electronics) is now your preferred Supplier for the CAP.

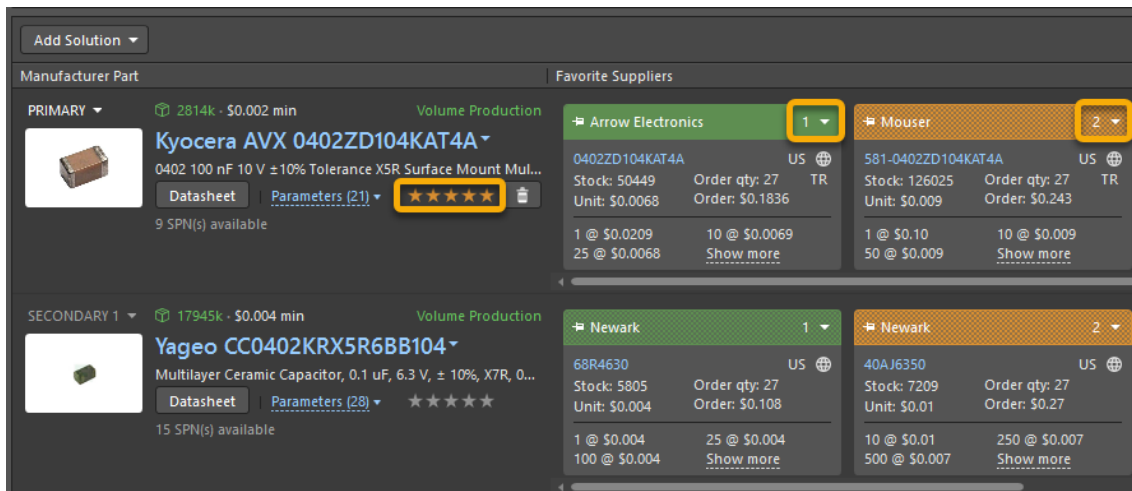


Figure 12b. Ranking solutions for the Preferred Supplier



6.3 Ranking for Solutions

30. There's a large number of components without ranking. The ranking sets a priority for components that have more than one solution. It is represented by up to 5 Stars.
31. Back in the ActiveBOM document, select C1. With Step 29.b), you have set 5 Stars to define the ranking for C1, as seen in Figure 13.

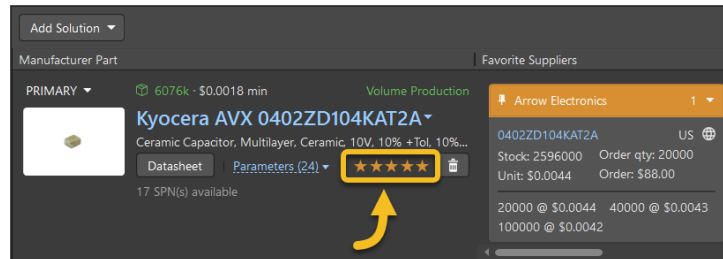


Figure 13. Solution and ranking with 5 stars

32. Altium offers an automatic ranking definition.
- Select all components in the table with **CTRL+A**.
 - Right Click** and select **Set Ranks Automatically**.
 - For C1, the 5-Star rank is still a manual ranking, the other solutions for C1 set to 4-Star, 3-Star, depending on the number of allowed solutions per Item. Solutions from other components are similar set to 5-Stars, 4-Star, 3-Star, and so on.

Hint: The solutions are automatically ranked from highest to lowest, based on the part availability, price and manufacturer lifecycle state.

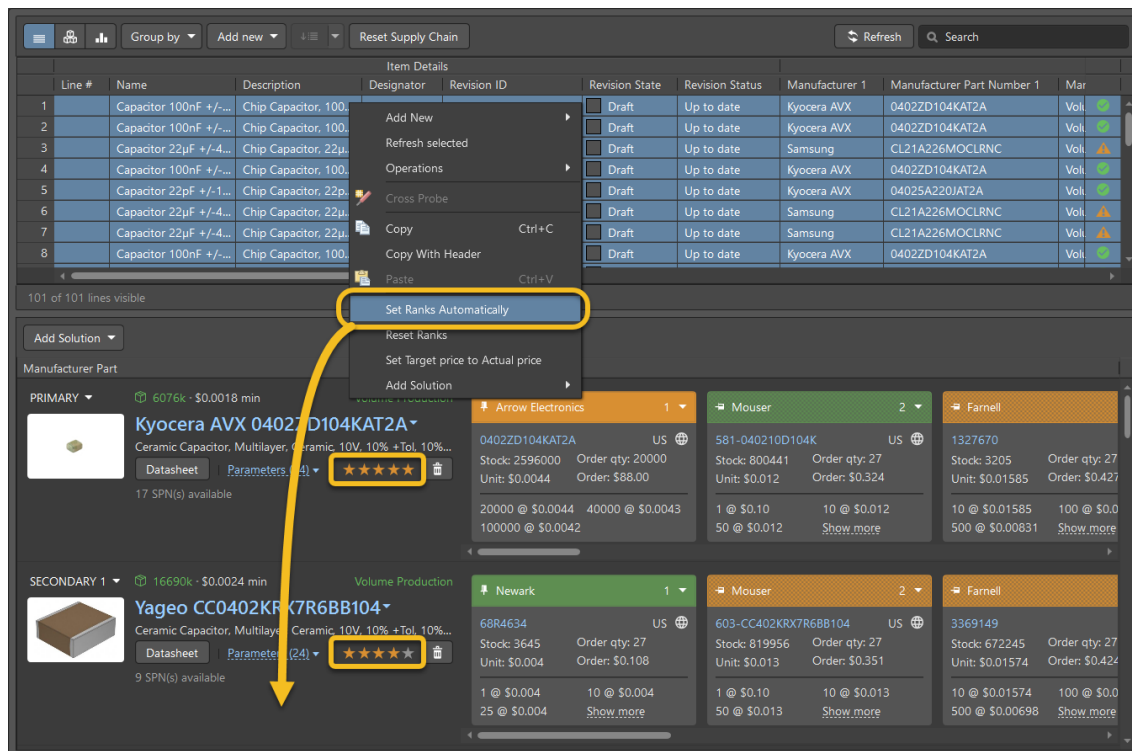


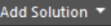
Figure 14. Automatic ranking for ActiveBOM

7 Unmanaged Components – Update Example

Unmanaged components are parts not sourced from a managed content server and don't have pre-defined Part Choices. However, they are still supported in ActiveBOM. Since these components aren't linked to the supply chain, you can add those links directly in ActiveBOM. The tool can search for manufacturer details in the component's existing information, or the designer can manually add manufacturer links if needed.

33. While in the ActiveBOM, select U2. You can use the *BOM Check* filter in the *Properties* panel to reduce the number of objects listed in the *ActiveBOM*. U2 is an unmanaged component, without a Solution, as described in Step 19.h).

34. Next, you will add a solution for U2, as shown in Figure 15.

- Click on the Button **Add Solution** , or do a right click in the list, and select **Add Solution » Create/ Edit Manufacturer Links** to open the dialog *Edit Manufacturer Links*.
- Click on **ADD...** to open the *Add Part Choices* dialog.
- The information from the column *Name* (the Component Comment) will be used as search criteria and the search results will be listed in the *Add Part Choices dialog*. Below of the search field, you can see the current Filter information with `LP5907SNX-3.3`.
- Select one of the solutions: U2 has just 1 solution that is for Volume Production and has a Footprint / Symbol. See Figure 15 as an example.
- Click on **OK** to close the *Add Part Choices* dialog.

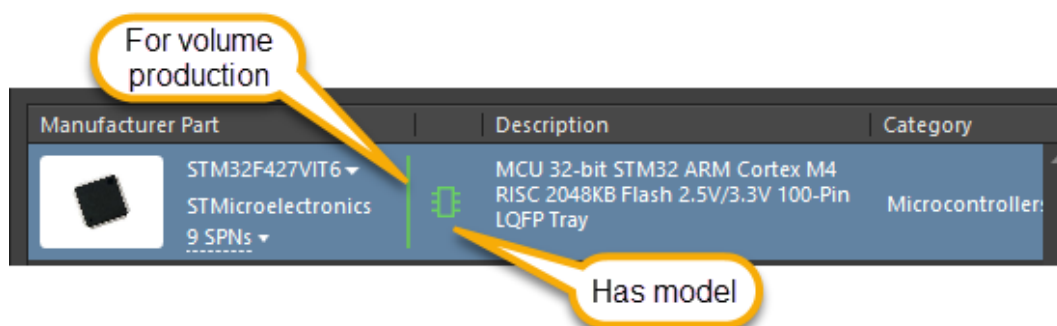


Figure 15. Example for a selected solution

Hint: You can select several search results to add several solutions for one component. You can repeat the search step to search for additional solutions and add them as additional solution. The number of possible solutions depends on the number of solutions you allowed in the *Properties* panel.

- Click on **OK** to close the *Edit Manufacturer Links* dialog.
- Remove the BOM Check filter, if used.
- U2 is now a component with a solution, even if it is an unmanaged Component.

Caution: Depending on the filter and search criteria, the resulting component may have a different footprint than the one currently used in the design.

8 Obsolete Components

Next, you will look for an Obsolete Item.

35. Sort the Table by Lifecycle or use the Grouping for Lifecycle (Manufacturer Lifecycle 1).

36. Some of the Items are reported as obsolete. You will replace one of them.

37. Select one of the connectors: CN1 or CN2.

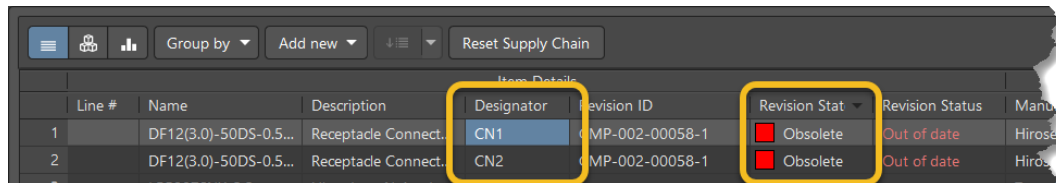


Figure 16. Selecting the items

Note: The lifecycle status of a component will not be updated or changed, the component will still be officially listed as Obsolete.

38. Next, you will add another solution:

- a) Click on **Add Solution** and execute **Create / Edit Manufacturer Links**.
- b) In the *Edit Manufacturer Links* dialog, click on **Add...**
- c) The Part Choice window will auto populate with alternates, if available, for the obsolete part.
 - i) If you find a solution select the part choice and press **OK**.
 - ii) If the component is too old or end-of-life, you may not be able to find a solution, making it necessary to consider alternatives, but that also could mean that a PCB redesign may be necessary.
 - iii) To check availability, feel free to click the Component Name within the details to open Octoparts, Figure 17.

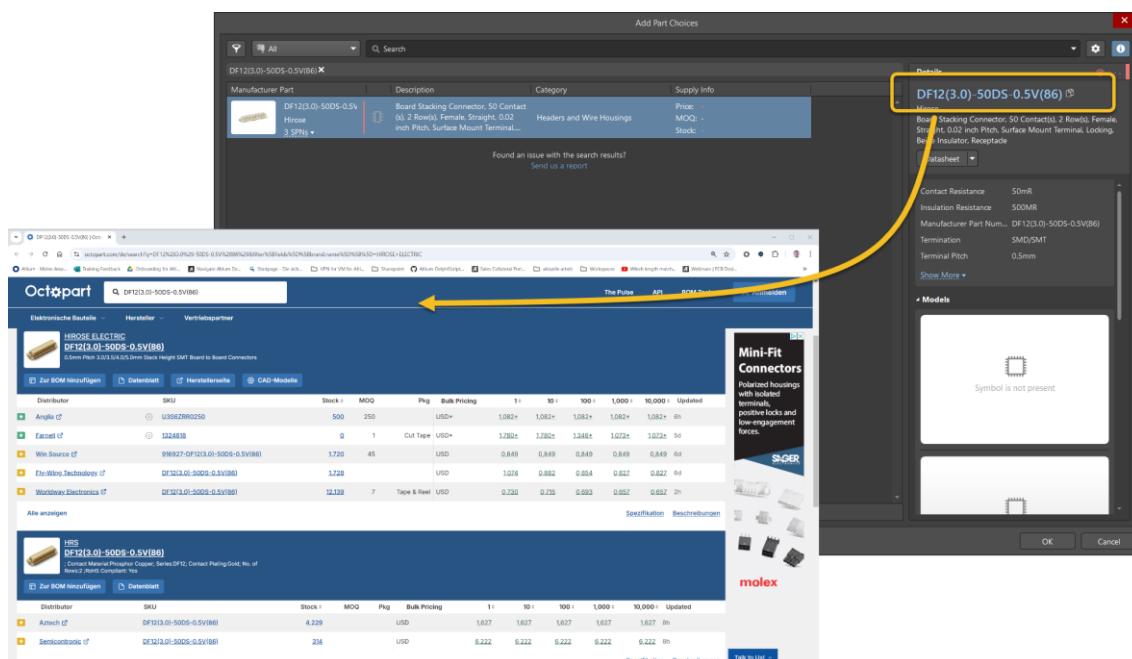


Figure 17. New Solution for CN1 / CN2

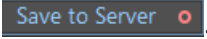
- d) Press **OK** or **Cancel** to close the *Add Part Choices* and the *Edit Manufacturer Links* dialog.



You have seen the process of solving *BOM Checks* violations within ActiveBOM. Typically, all violations should be resolved within a design. However, since this is for training purposes only, you don't need to resolve all violations. Focus on the example violations shown above.

39. Save all documents using **File » Save All**.

40. Save the modifications to the server:

- a) In the *Projects* panel, next to the Project name you find the command **Save to Server** .
- b) Select **Save to Server**.
- c) In the dialog *Save [Project Name]*:
 - i) Add the comment `ActiveBOM - [Add Your Name] - Finished`.
 - ii) Select **OK**.

41. When ready, close the project and any open documents, **Window » Close All**.



ActiveBOM

Altium Designer Advanced Training with Altium 365

Thank you for choosing **Altium Designer**