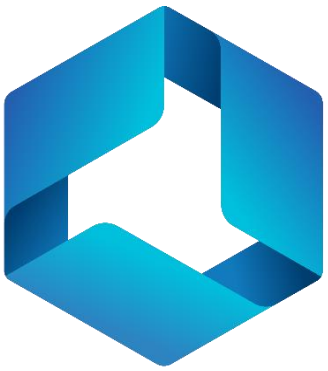

ALTIUM 365

Altium Designer

Essentials Course - Altium 365

Module 24: PCB Global Editing and List Panel

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.
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

Module 24: PCB Global Editing and List Panel 3

1.1 Purpose 3

1.1 Shortcuts 3

1.2 Preparation 4

1.3 Properties Panel 5

1.3.1 Modify PCB Traces 5

1.4 Filter Panel 9

1.4.1 Object Checkboxes 9

1.5 Find Similar Objects 10

1.5.1 PCB Find Similar Objects Dialog 10

1.5.1.1 Selecting Vias 10

1.5.1.2 Changing Via Size 12

1.6 List Panel 14

1.6.1 Selecting Components 14

1.6.2 Modifying Properties 15

Module 24: PCB Global Editing and List Panel

1.1 Purpose

In this exercise, we'll explore the Find Similar Objects function and common panels used in the PCB environment such as the *Properties* panel, *Filter* panel, *List* panel and *PCB* panel.

Global Editing allows us to make a change to multiple objects at the same time. Global Editing tools are available in the schematic, schematic symbol editor, PCB editor, and footprint editors.

1.1 Shortcuts

Shortcuts when working with Module 24: PCB Global Editing and List Panel

S » P or **Ctrl+H:**	Select Connected Copper
Select Track, Vias, PAD » TAB:	Toggle Connected Copper
Shift+C:	Clear
Shift+F:	Find Similar Objects
Shift+Ctrl+X:	Cross Select Mode
Ctrl+S:	Save Document

1.2 Preparation

1. **Close all existing projects and documents.**
2. Next, create a Copy / Clone of the Training Project `Module 24 PCB Global Editing and List Panel`.
3. Select **File » Open Project…** to open the *Open Project* dialog.
4. Navigate to the predefined Training Project `Module 24 PCB Global Editing and List Panel (Top\Projects\Altium Designer Essentials Training Course\...)`.
5. Select **Open Project as Copy…**

.
6. At the new dialog *Create Project Copy.*
   a) Add your name to the project name: `Module 24 PCB Global Editing and List Panel – [Your Name]`.
   b) Add a description: `Altium Essential Training - Module 24 – [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.
7. Wait until Altium Designer created the copy of the project and opened the project for you at the *Projects* panel, this may take up to 1 minute.

For details how to Copy / Clone the predefined training project see Module 8 Making the Connection, Step 1.3 Preparation.

1.3 Properties Panel

The *Properties* panel is used to modify properties for multiple objects easily. In this section, we will use the **Edit » Select** commands to select items, then use the *Properties* panel to make changes in the PCB editor.

1.3.1 Modify PCB Traces

8. Open the `Module 24 PCB Global Editing and List Panel.PcbDoc` document and zoom in on `U1` as shown in Figure 1 below.

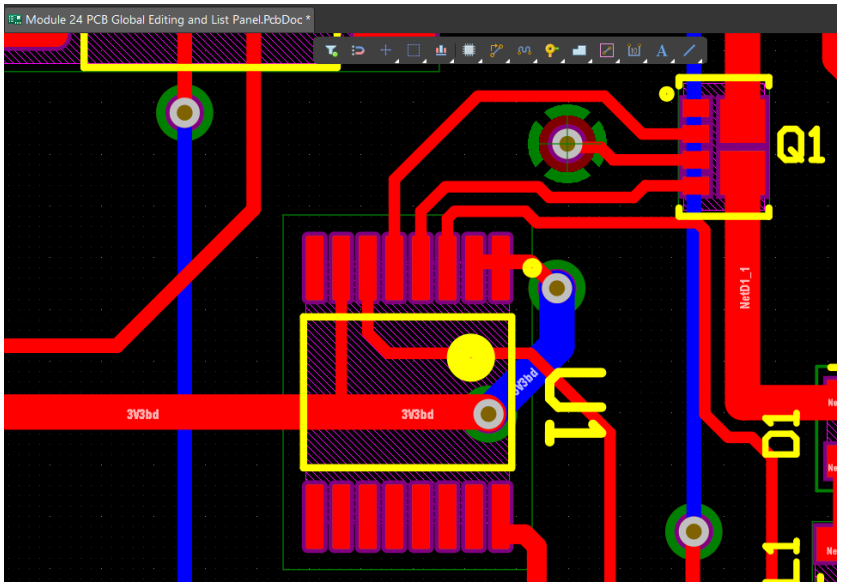

Figure 1. WCToppingGlobalEditing.PcbDoc near component U1

9. Ensure nothing is selected on the board.
   Then, open the *Properties* panel from the **Panels** button.
10. Ensure that the Selection Filter is set as shown in Figure 2 below. The same pre-selection filter can also be accessed from the *Active Bar*.

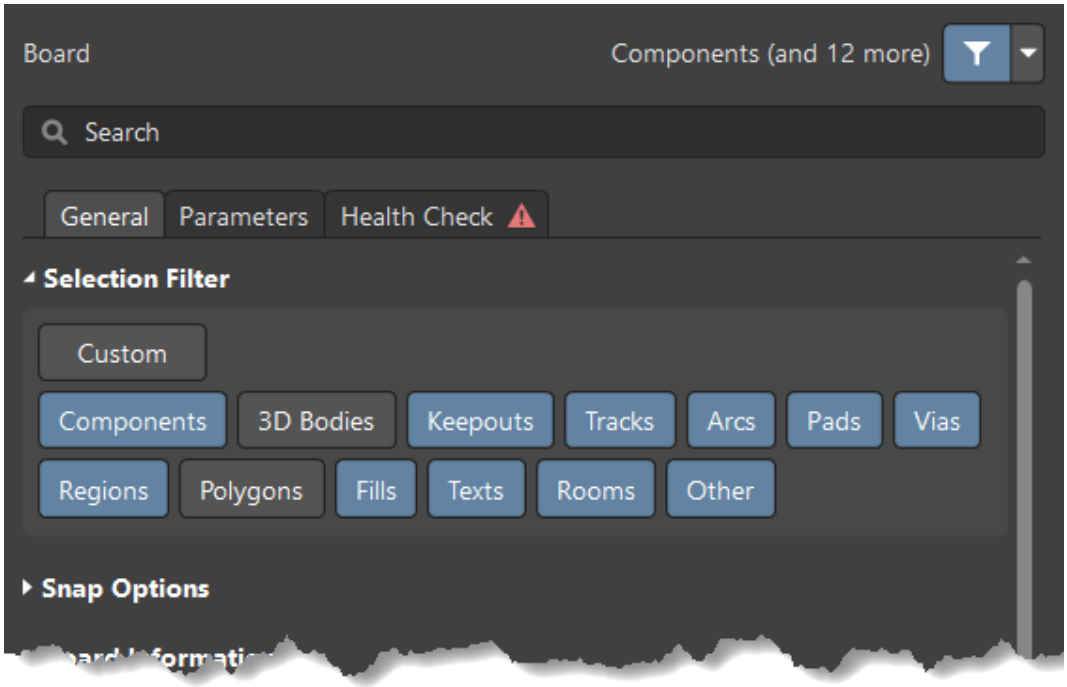

Figure 2. Configuration for the Selection Filter

11. You can select multiple traces quickly by using the **Edit » Select » Connected Copper** command.
    a) While holding the **Shift key**, select the multiple traces on the 4 pads similar to Figure 3 below. You could also select multiple traces and hit the **TAB** key to select the connected copper.
    b) Right-click or hit **Esc** to terminate the +selection command.

If you press **S** you open the Selection menu. The Selection menu shows you a variety of options for a selection. Altium offers e.g., Select Touching Line.

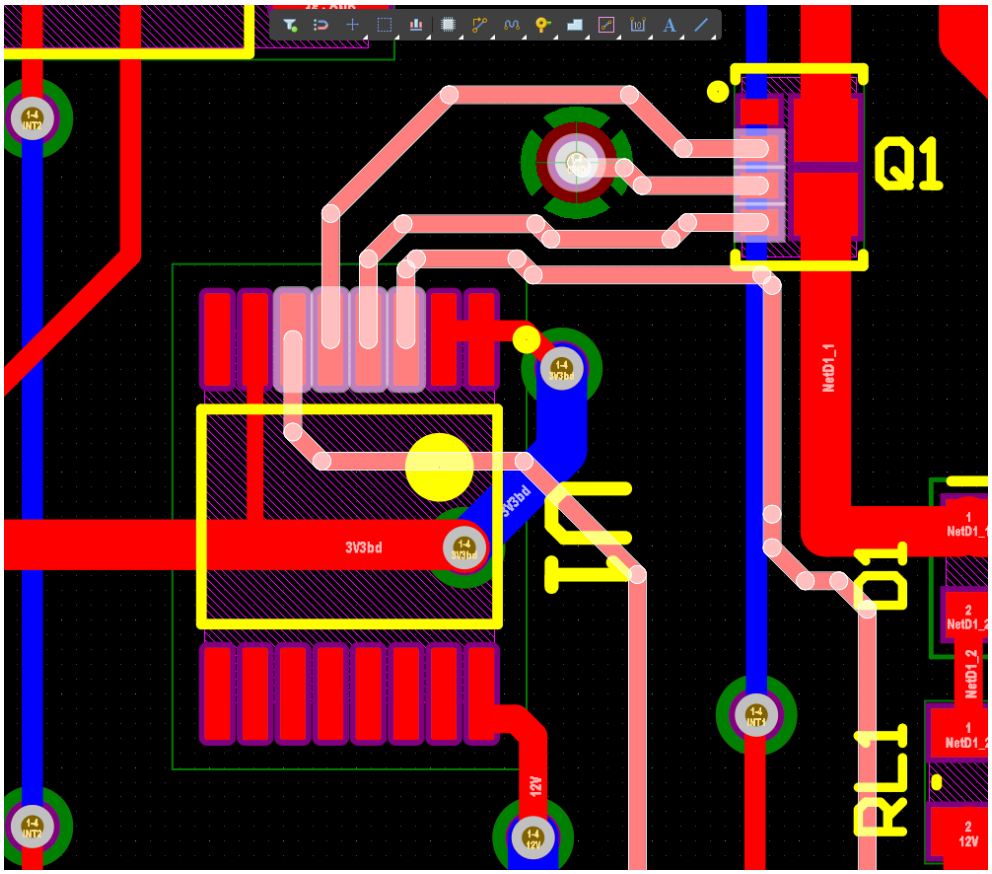

Figure 3. Selecting multiple traces with the Shift key and "Connected Copper" command

12. We would like to change the widths of the selected tracks to `10mil`. The *Properties* panel contains the parameters and values of the selected objects. However, the visible values are represented by '`*`' since there are different values for the selected objects.
    a) As of right now, both tracks and pads are currently selected. Let's change the *Post-Selection* filter to only select tracks by clicking on the drop-down arrow beside the *Post-Selection Filter* button, in the top-right of the *Properties* panel as shown in Figure 4.
    b) Click on **All-Objects**
    c) Ensure to only click **Tracks** from the *Post-Selection* filter (refer to Figure 4).

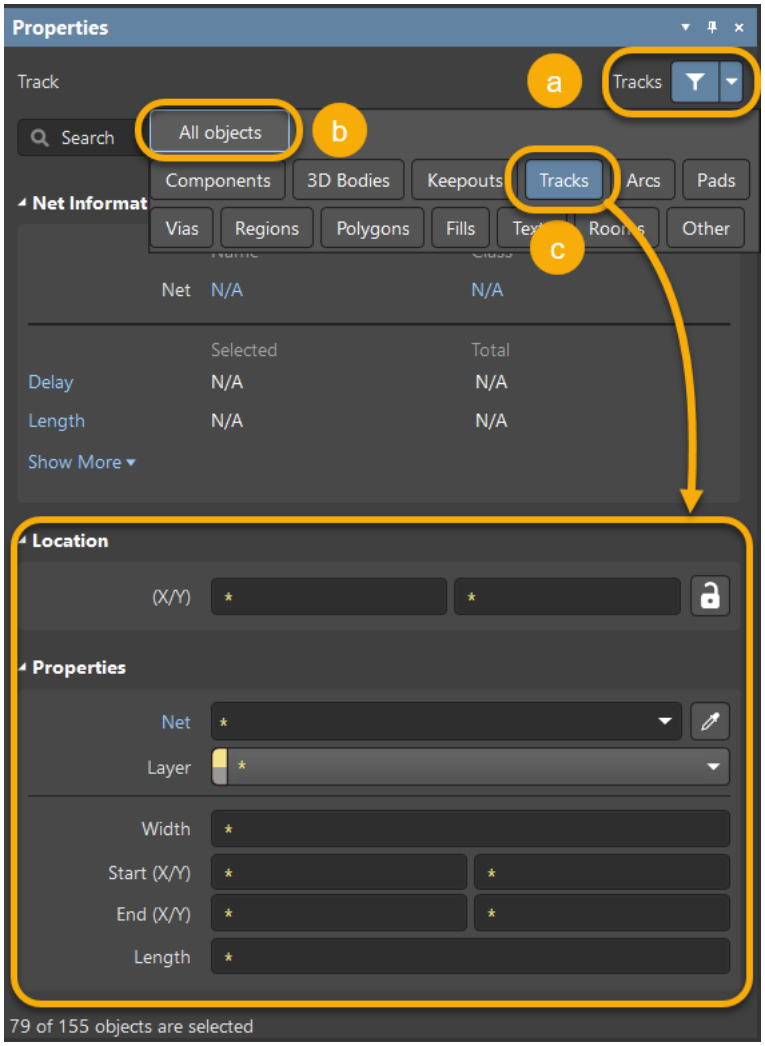

Figure 4. Properties panel with the post-selection filter set to Tracks only

13. In the *Properties* panel, locate the *Width* field and change the value from `12mil` to `10mil` and hit **Enter**. The changes will be reflected as shown in Figure 5 below.
14. Hit **Shift+C** anywhere in the PCB to clear the selection, alternatively left mouse click in free space to clear selction.

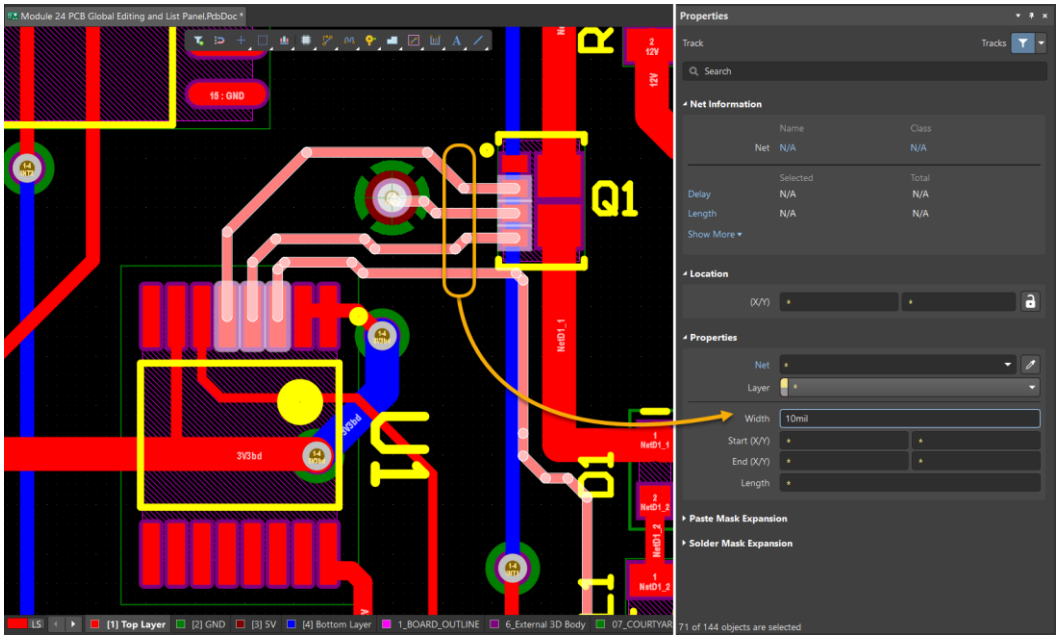

Figure 5. Board Layout with modified traces

In this exercise, we use the Global Editing workflow to modify the trace width. An excellent alternative is to re-apply the preferred width and clearance rules of an existing route using the command **Route » Retrace Selected.**
The **Retrace Selected** feature is covered in our Advanced Training.

1.4 Filter Panel

The *PCB Filter* panel is another method to select items based on a query language. In the examples in this section, we will once again use the *Properties* panel to make changes after the appropriate elements have been selected using the *PCB Filter* panel.

1.4.1 Object Checkboxes

15. We will first familiarize ourselves with the options within the *PCB Filter* panel. Open the *PCB Filter* panel from the **Panels** button.
16. From the topmost pane, enable the **Track** checkboxes associated with **Net** and **Component** as shown in Figure 6.
17. Do the same for the **Via** checkbox for **Net** and **Component**. Notice that the *Filter* field is populated with a query based on the combination of checkboxes you have selected.

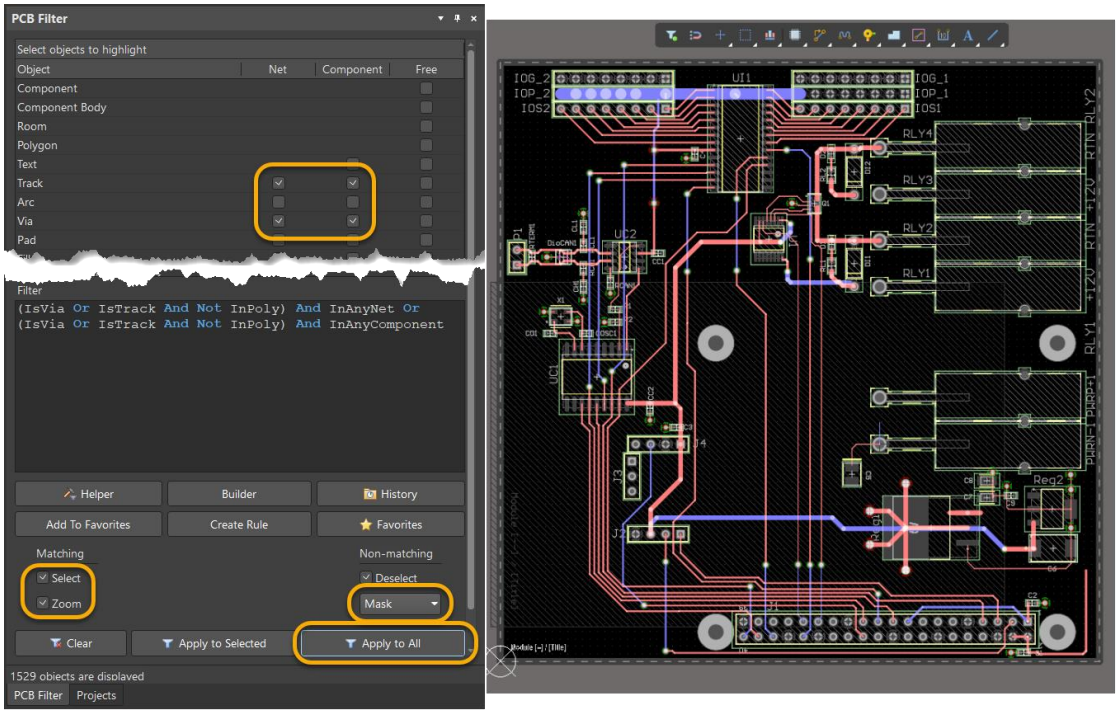

Figure 6. PCB Filter panel selections reflected in the PCB

18. Make sure that at the bottom of the *PCB Filter* panel, you have **Select** enabled under *Matching*, **Deselect** enabled under *Non-matching* and the mode set to **Mask**.
19. From the *PCB filter* panel, click the **Apply to All** button in the bottom right corner of the panel to see the effect of the query to the design. You should have tracks highlighted as shown in Figure 6.
20. Disable the **Comp** checkbox for **Track** and hit **Apply to All** again. Notice that the component outlines have now been excluded from the selection on the board.
21. Enable the **Top Layer** checkbox in the *Layer* pane then click **Apply to All**. Only tracks on the Top Layer will be highlighted.
22. Click the **Clear** button to reset the selection on the board.
23. Try enabling different combinations of the checkboxes then click **Apply to All** to see which objects are selected.
24. Right-click anywhere in the top pane of the *PCB Filter* panel and select **Uncheck All** to reset the query back to `Not All`. Uncheck any selections in the *Layer* pane as well.

1.5 Find Similar Objects

The *Find Similar Object (FSO)* dialog is another powerful way to select multiple items in either the schematic or PCB environments. It allows you to choose a particular element in the design and then identify one or more common parameters that will be used as the basis for global selection.

1.5.1 PCB Find Similar Objects Dialog

1.5.1.1 Selecting Vias

25. Let's first make the tracks and vias visible again by going to the *View Configuration* panel, **View » Panels » View Configuration**.
26. Open the *View Options* tab and enable the icon for **Tracks** and **Vias**. We will now select vias of a specific size and change their sizes.
27. Hit **Shift+C** in the PCB to ensure that there are no objects are currently selected or masked.
28. Select the via located near `UI1-14` and right-click to *Find Similar Objects* as shown in Figure 7 below.

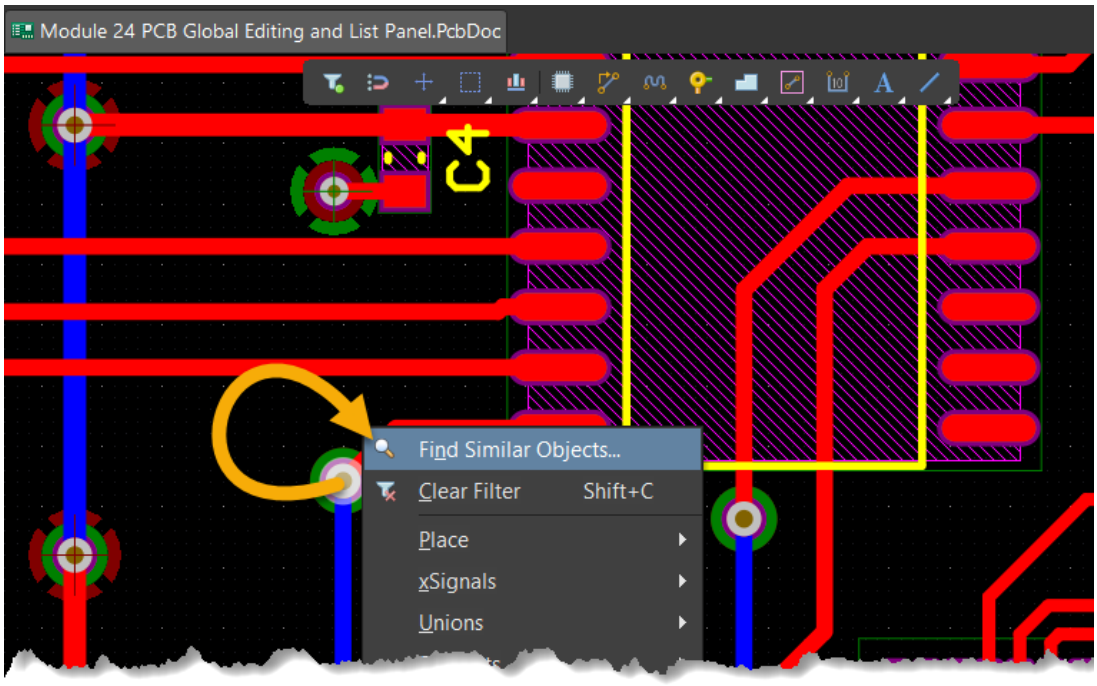

Figure 7. Right-click and select Find Similar Objects

29. The *FSO* dialog will appear, displaying via parameters as shown in Figure 8 below. Note that by default the *Object Kind* is set to a scope of **Same**.
30. Enable the **Mask** setting at the bottom of the panel to mask the rest of the board.

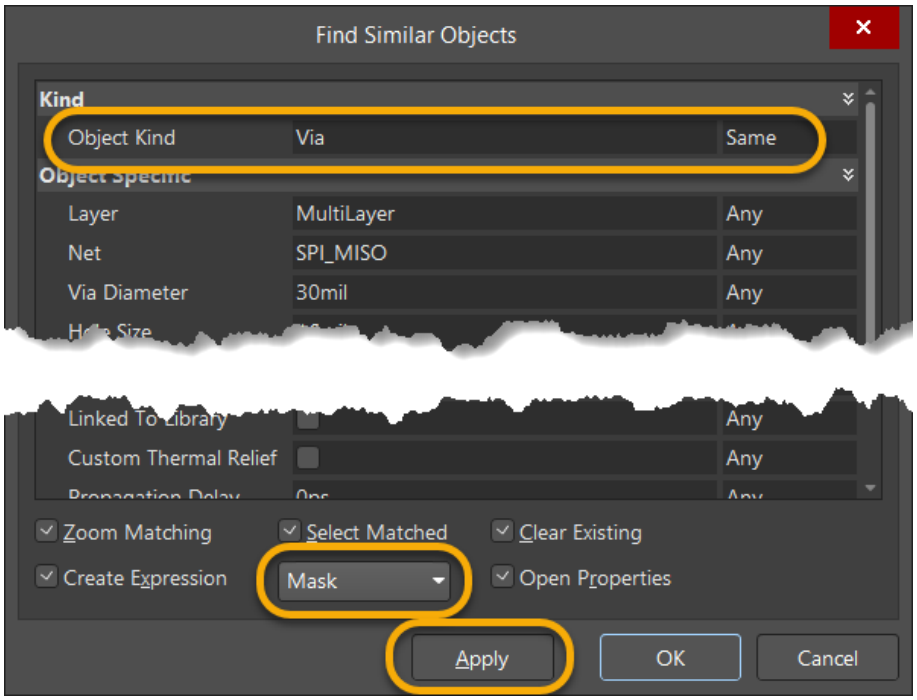

Figure 8. FSO for the selected Via

31. Click the **Apply** button to select all vias in the design as shown in Figure 9 below.

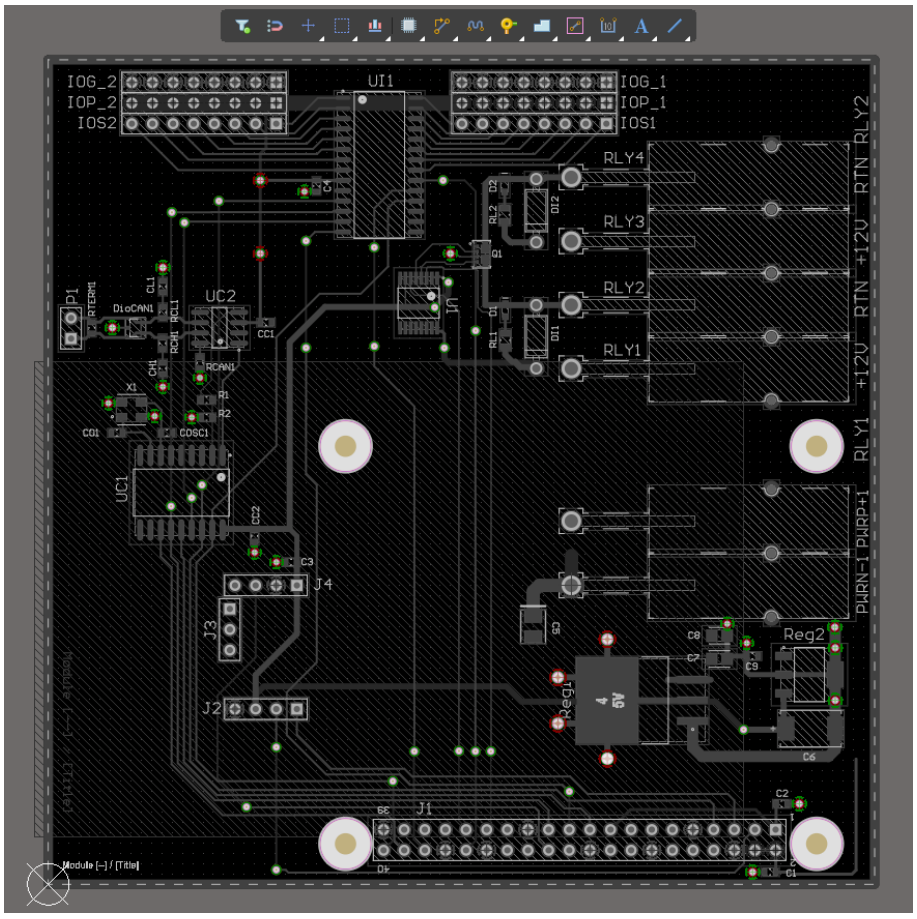

Figure 9. All Vias selected using the Find Similar Objects dialog

1.5.1.2 Changing Via Size

32. Instead of all the vias, we only want to change the via diameter of the 30mil vias. In the *Find Similar Objects* dialog, set the *Via Diameter* to **Same** as shown in Figure 10 below.
33. Click the **Apply** button. Now only the vias of 30mil diameter are selected.

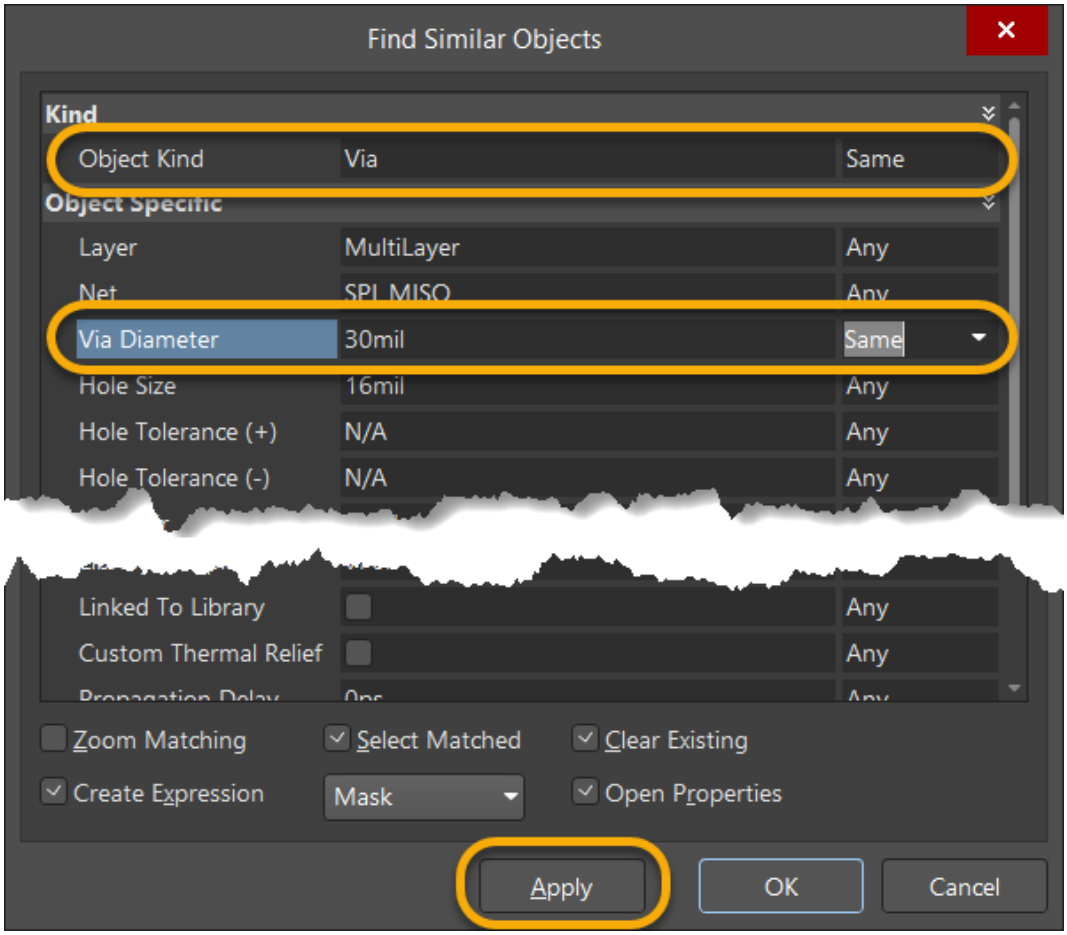

Figure 10. Find Similar Objects dialog to target specific Vias

34. Click the **OK** button to close the dialog and move to the *Properties* panel.

35. In the *Properties* panel, as shown in Figure 11 below:
    a) Locate the *Via Diameter* field under the *Via Stack* pane and change the value from `30mil` to `20mil`.
    b) Change *Hole Size* from `16mil` to `10mil`.
    c) Hit **Enter** and see the changes made on the selected vias.

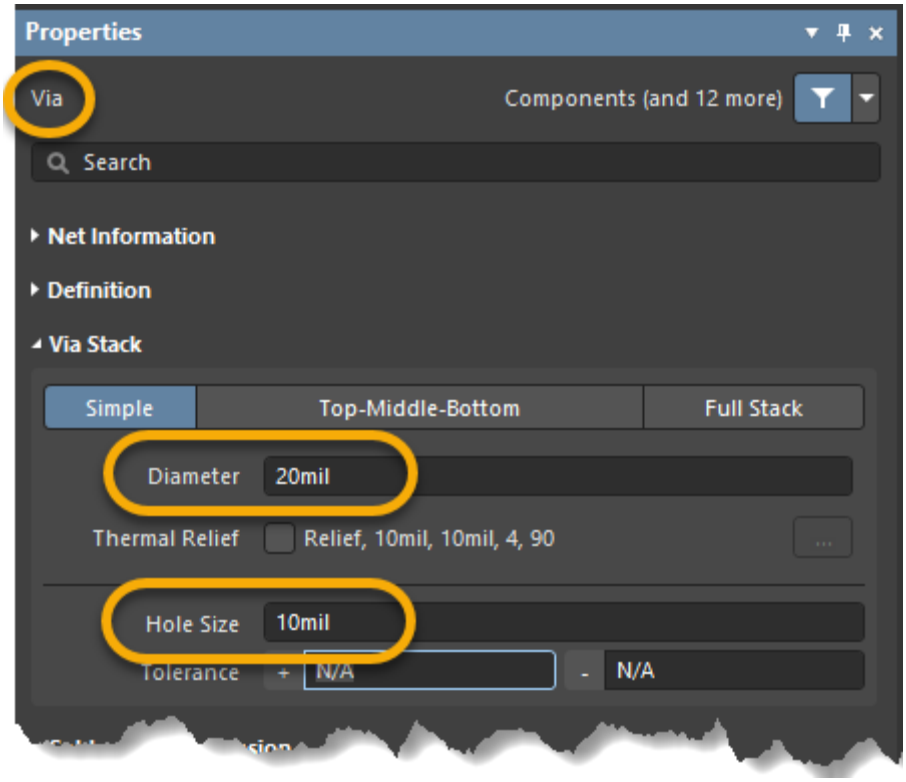

Figure 11. Properties panel with the new values for Via Diameter and Hole Size

36. Hit **Shift-C** in the PCB to clear any selections.

1.6 List Panel

The *PCB List* panel displays items from your design in a tabular format and is used for both selecting and editing.

1.6.1 Selecting Components

37. We will now view attributes of selected components in a tabular form and lock their positions, using the *PCB List* panel.
38. Open the *PCB List* panel through the **Panels** button. If this is your first time using it, you may need to make it larger. Keeping the panel in Floating mode will make it easier to use.
39. Change the list settings in the top-left corner of the panel to match those shown in Figure 12, by clicking on the words with the hyperlink.

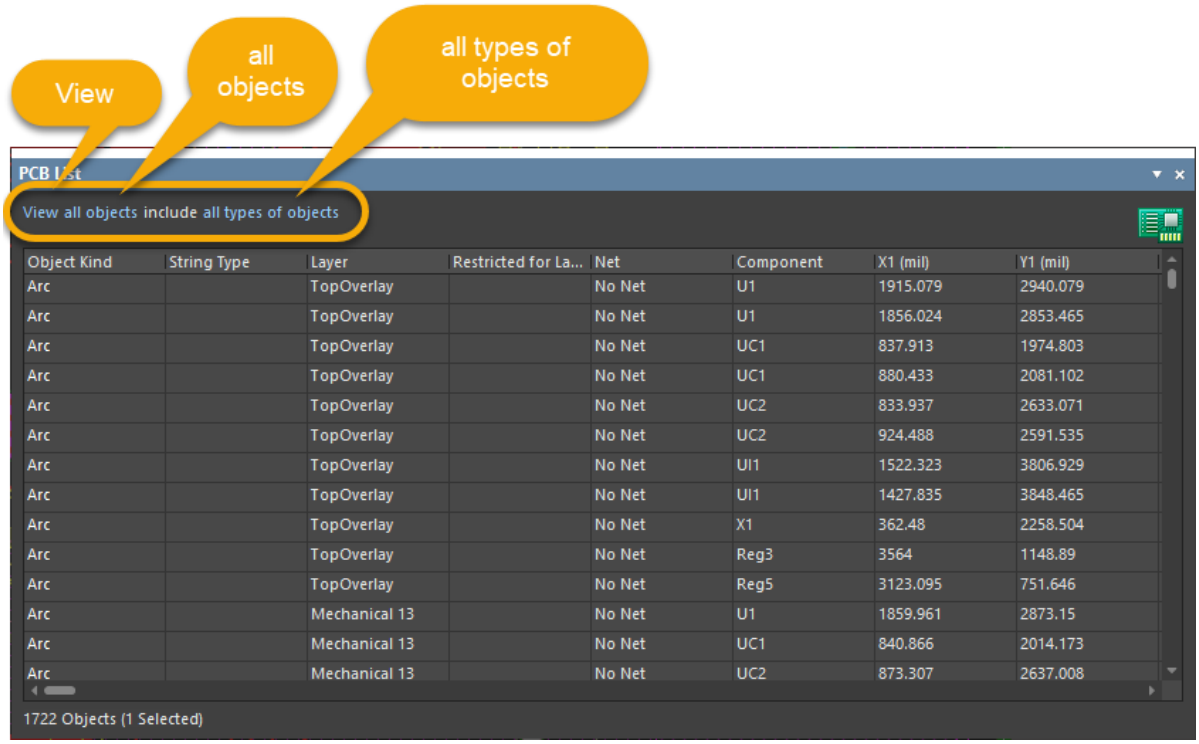

Figure 12. PCB List panel showing All Objects in the design

40. Open the `Power Supply.SchDoc`.
41. Ensure that **Cross Select Mode** is enabled, indicated by its blue highlight in **Tools » Cross Select Mode** (as shown in Figure 13 below). If it's not enabled, click the icon to enable it.

Figure 13. Cross Select mode enabled

42. Hold the **Shift** key to select all of the capacitors on the `Power Supply.SchDoc`.
43. Return to the PCB document and the five components should now be selected in the bottom right corner of the PCB.

44. In the *PCB List* panel, click on the **View** name hyperlink**,** and change it to **Edit.**
45. Change the **all objects** hyperlink to **selected objects.** The *PCB List* panel will now display only the selected components as shown in Figure 14 below.

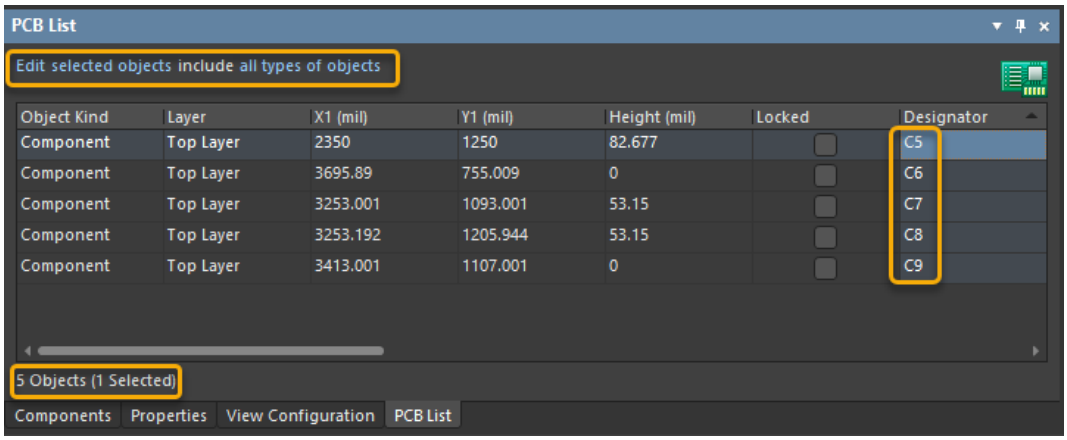

Figure 14. PCB List panel showing 5 components selected in the PCB.

1.6.2 **Modifying Properties**

Our objective is to lock all of these selected components. We will do this by changing the *Locked* column in the *PCB List* panel.

46. Enable the **Locked** checkbox for the first component in the *PCB List* panel, as seen Figure 15

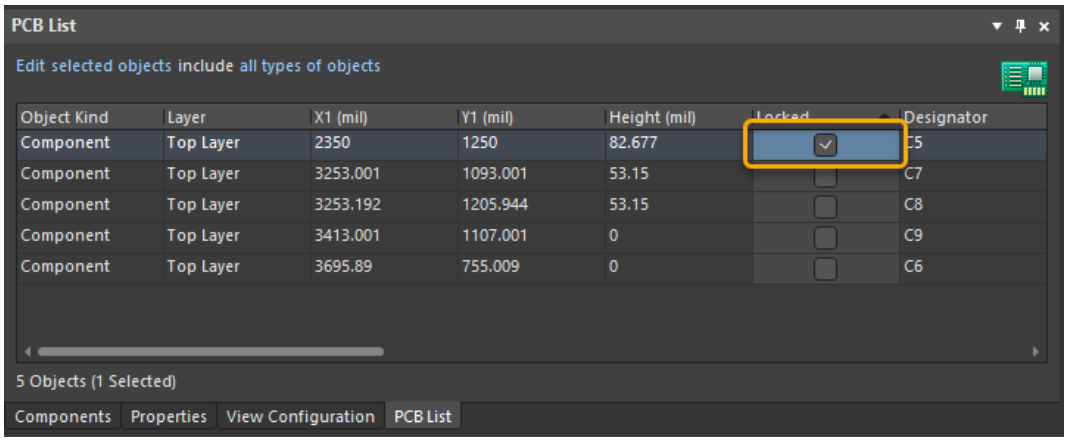

Figure 15. Enable Locked

47. Click on the cell that you just locked and copy it with the **Ctrl+C** keys.
48. Select the locked cells for the remaining four components by clicking on the first unchecked cell, hold the **Shift** key, then click on the last cell as shown in Figure 16 below.

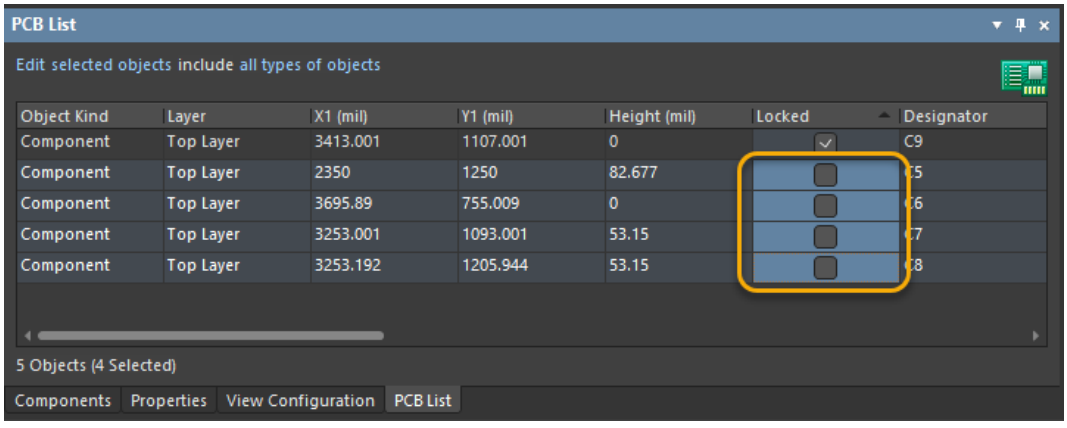

Figure 16. Select Cells

49. Use the **Ctrl+V** keys to paste the locked property as shown in Figure 17 above.

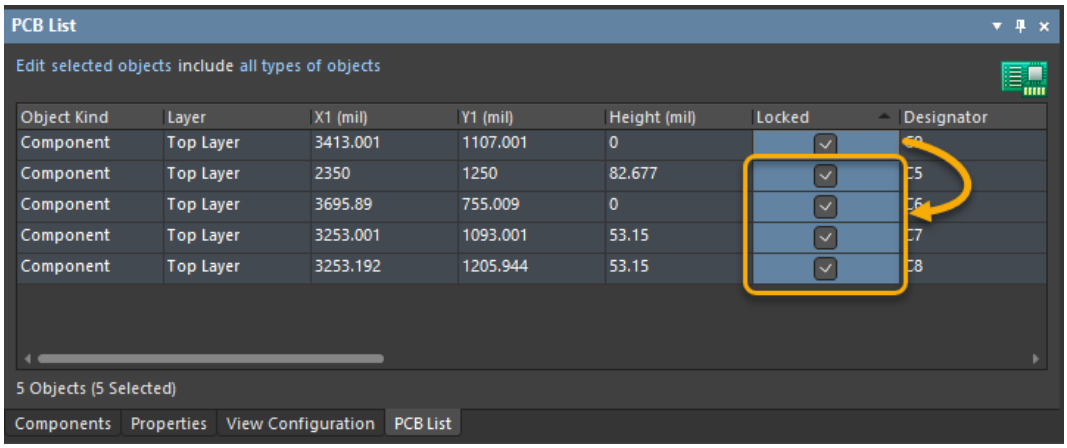

Figure 17. Copy & Paste to quickly change multiple rows in the PCB List panel at once

The same method can be used to copy and paste multiple parameters in the *PCB List* panel at the same time.

50. Try to move any of the five components you just locked to confirm they will not move.
51. Save all documents using **File » Save All**.
52. Save the modifications to the server:
    a) At the *Project* panel, next to the Project name you find the command **Save to Server**

.
    b) Select **Save to Server.**
    c) At the dialog *Save [Project Name],*
        i) Activate the checkboxes for the files that are not under version control.
        ii) Add the comment `Module 24: PCB Global Editing and List Panel - [Add Your Name].`
        iii) Select **OK.**
53. **When ready, close the project and any open documents, Window » Close All.**

Consider the circumstances where these batch replacement methods would help you with your designs.

Congratulations on completing the Module!

Module 24: PCB Global Editing and List Panel

from the

Altium Designer Essential Course
with Altium 365

Thank you for choosing Altium Designer