

Altium Designer

Advanced Course

Module: Using Snippets with Design
Reuse Panel

Software, documentation and related materials:

Copyright © 2022 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®, Altium 365™, Altium Concord Pro™, Altium Designer®, Altium Vault®, Altium NEXUS™, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Codemaker™, Common Parts Library™, Draftsman®, DXP™, Easytrax®, EE Concierge™, xSignals®, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, 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 subsidiaries. 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

Using Snippets	3
1.1 Purpose	3
1.2 Shortcuts	3
1.3 Preparation	3
1.4 Accessing the Design Reuse Panel	4
1.4.1 Creating a Schematic Snippet	4
1.4.2 Creating a PCB Snippet.....	6
1.5 Using Snippets	9
1.5.1 Using the Snippets in a Project	9
1.5.2 Designator Assignment for Snippets	11
1.5.3 Adding Snippets to Project	11
1.5.4 Synchronizing Snippets within the Project	13
1.5.5 Creating Unique Designators.....	14

Using Snippets

1.1 Purpose

Snippets provides a simple and easy way to save and reuse sections of design circuitry. They can be added into any design, without you having to start from scratch each time. The Snippets system lets you save any selection of:

- Circuitry on a single schematic sheet.
- Circuitry in a PCB design, including the components and the routing.
- When connected to an Altium 365 Workspace, you can create a single entity – a Reuse Block – that can contain both schematic circuitry and its physical representation for the PCB. When such a reuse block is placed on a schematic sheet, its physical representation will be placed automatically in the PCB document during the ECO process.

We will use local snippets in our training.

1.2 Shortcuts



Shortcuts when working with Using Snippets

F1:	Help
W+V:	Vertical Split / Vertical Tile
W+H:	Horizontal Split / Horizontal Tile
T-P:	Open Preferences
CTRL+S:	Save Document

PCB active document

C-K:	Component Links
Shift-S	Single Layer Mode

SCH active document

T-A-A:	Annotation Dialog
D-U:	Update PCB document

1.3 Preparation

1. **Close all existing projects and documents.**
2. Open the `Using Snippets.PrjPCB` project found in its respective folder of the Advanced Training.



Any files or folders that point to Altium Designer installation folder will be deleted when a Complete Uninstall is performed. It is preferable to store any files, like the snippets we create during this exercise, outside of the Altium Designer folder.

1.4 Accessing the Design Reuse Panel

1.4.1 Creating a Schematic Snippet

3. Open the *Design Reuse* panel from the **Panels** button.
4. Check that the **Local Snippets** are active and selected, use Figure 1 as reference.

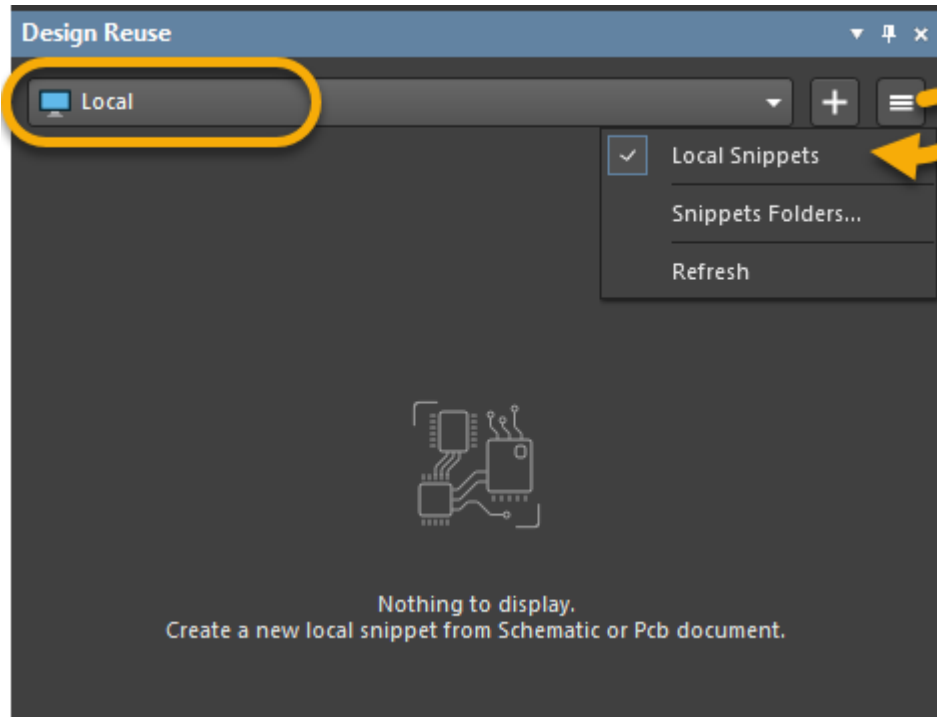


Figure 1. Local Snippets

5. Open the schematic file *LEDS.Schdoc*.
6. Select the of all objects in the schematics as shown in Figure 2.

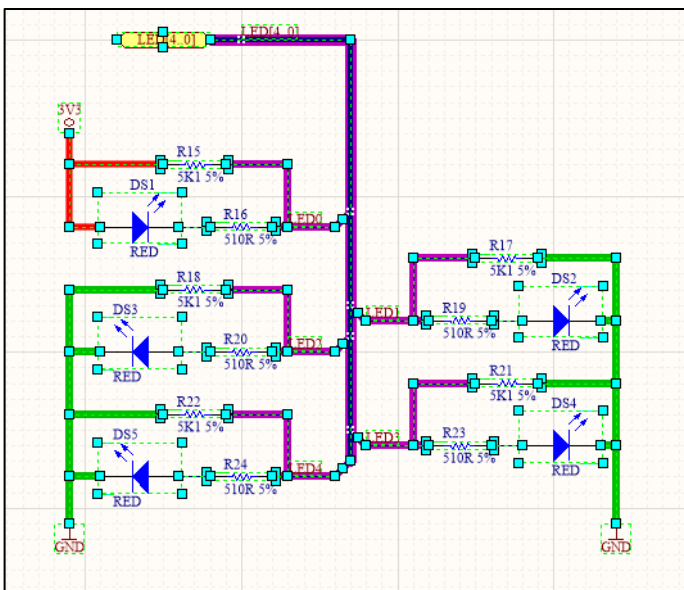


Figure 2. Selecting LEDS.SCHDOC

7. Right-click select **Snippets » Create Snippet from selected objects**.



Right-clicking on the any of the selected objects in the schematic lets you open the *Create a Snippet* command from the Snippet submenu.

8. When the *New Schematic Snippet* dialog opens we need to assign a name to the snippet:
- Change the name to `Schematic LED Design`, as shown in Figure 3.
 - In the *comment* field, add the following comment: `Schematic LED Design from PHY`.
 - Select *Local Folder* from the **Save to** drop down.
 - Click **Create** to continue.



By Clicking on the path information a new dialog opens, *Chose Snippet Folder*, that allows you to configure path and structure of your Snippet locations.

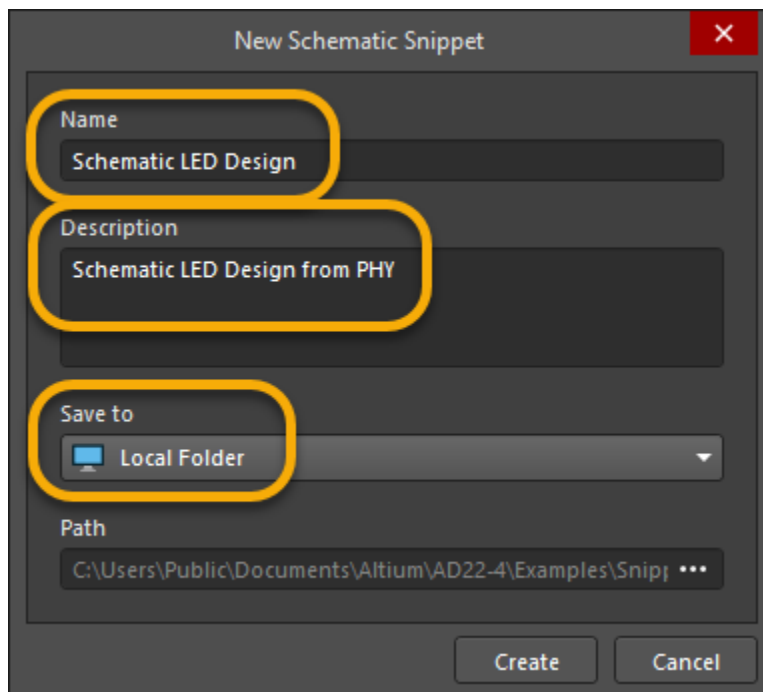


Figure 3. Creating New Snippets

9. Ensure you have created the schematic snippet before proceeding to the next step. You'll see a thumbnail of the snippet in the *Design Reuse* panel as shown in Figure 4.

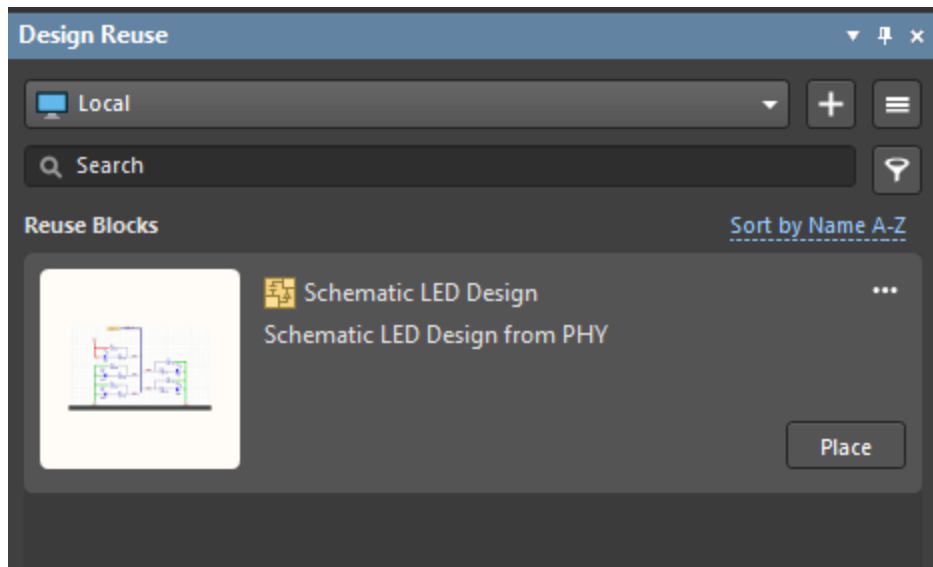


Figure 4. Snippet for Schematic



The Snippets Examples folder is the default folder included in the Altium Designer installation. Additional folders can be added at any time, and stored in the location of your choice by going to **Snippets Folders...** » **Open Folder...** and browse to the desired location.

1.4.2 Creating a PCB Snippet

10. Open the `Phy.PcbDoc` document.
11. To easily locate the components used from the schematic on the PCB, we will split the view between the schematic and PCB documents. From the *Documents Bar*, right-click on one of the documents and select **Split Vertical** as shown in Figure 5.

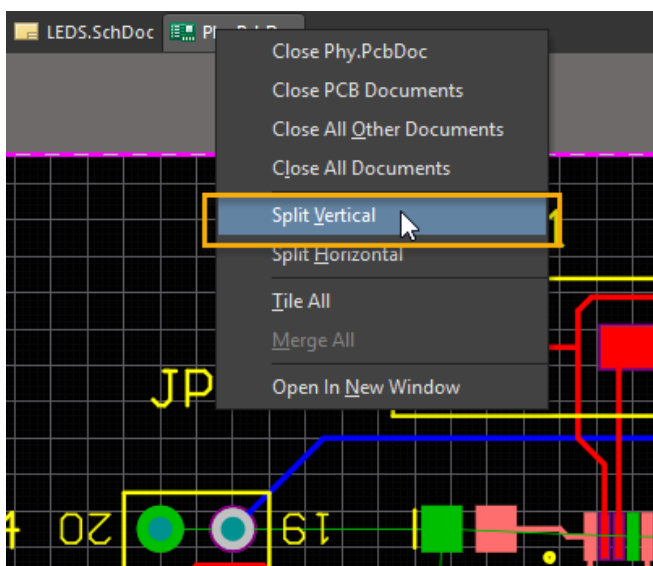


Figure 5. Splitting Vertical Documents on a Single Screen

12. Click on the `LEDS` schematic in the *Documents Bar* to make it the focused document.
13. Then, select everything on the `LEDS.SchDoc` if not already done.

14. From the **Tools** menu, select **Select PCB Components**. Notice that the components in the PCB will be selected as shown in Figure 6.

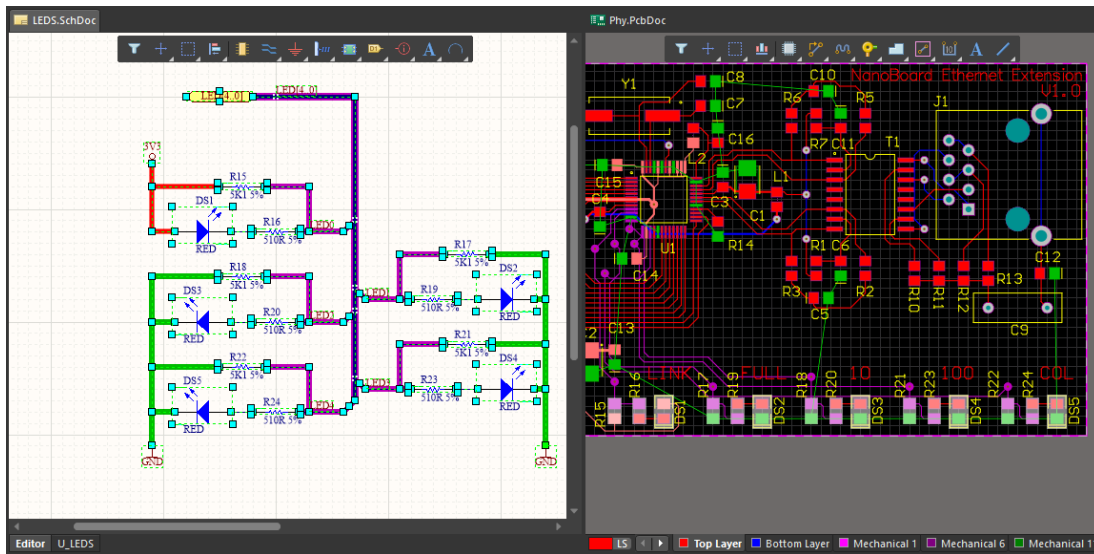


Figure 6. Cross Select Mode Preview

15. Now that the corresponding PCB components have been selected, we can switch back to a single document view by right-clicking on the PCB tab in the *Documents Bar* and select **Merge All**.
16. If not already the focused document, click on the *Phy.PcbDoc* document to make it the focused document.
17. Ensure the Top Layer is the active layer by clicking on the *Top Layer* tab at the bottom of the PCB workspace as shown in Figure 7.
18. If not already active, change the view to single layer mode by pressing **Shift+S** to mask out other layers as shown in Figure 7 as well.

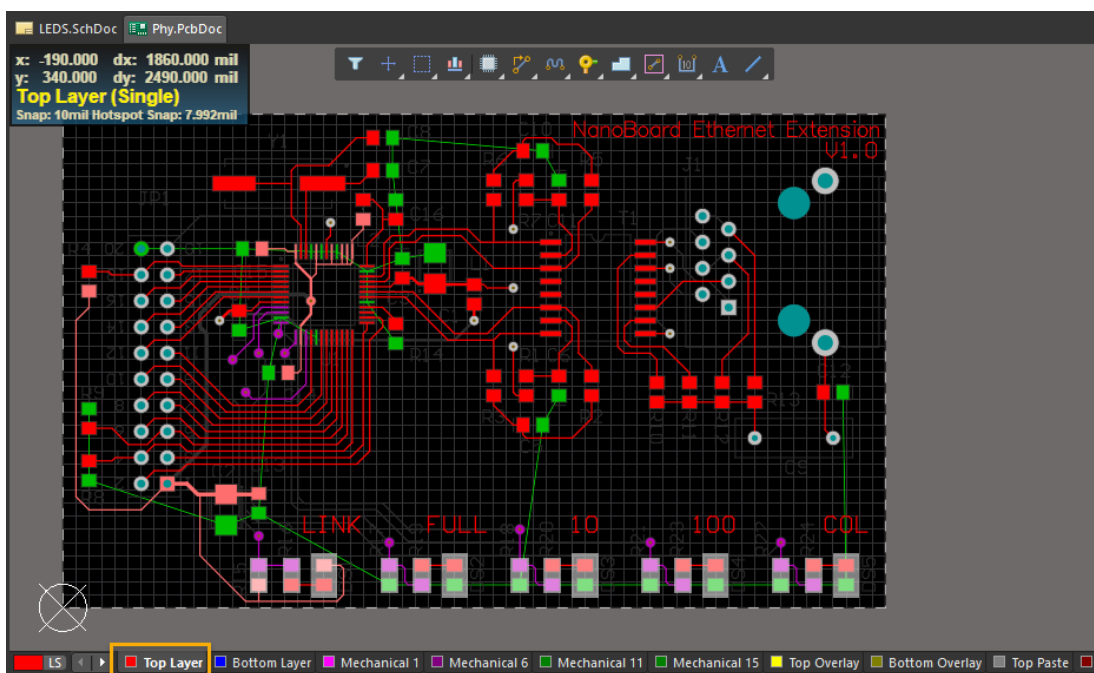


Figure 7. Single Layer Mode View

19. Even though the LED components are already selected, we'd like to add the tracks and vias that are attached to the components as well, as shown in Figure 9. To make this easier, click the **Selection Filter** icon in the *Active Bar* as shown in Figure 8.
20. Ensure that the *Selection Filter* is set to select only **Components, Tracks** and **Vias** as shown in Figure 8. If your filter is set to **All-On**, click on it to change it to **Custom**, then select **Components, Tracks** and **Vias**.

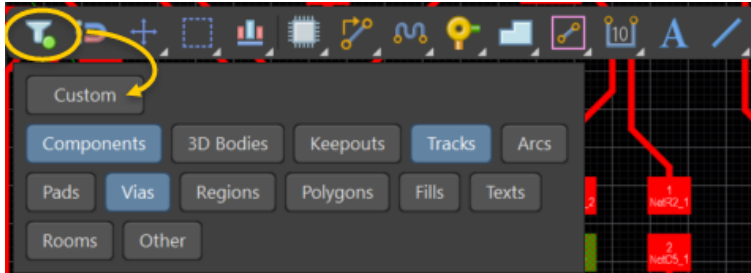


Figure 8. Setting the selection filter scope

21. Create a selection rectangle around all of the tracks and vias connected to the selected components as shown in Figure 9.

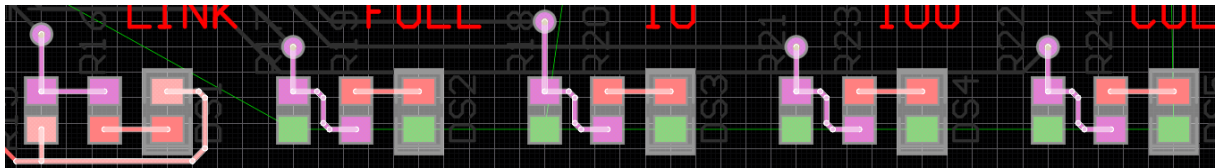


Figure 9. Selected LEDS components, traces and vias

22. When the objects are selected, right-click on one of the selected objects.
23. From the **Snippets** submenu, select **Create Snippet from Selected Objects**.
24. In the *New PCB Snippet* dialog:
 - a) Rename the snippet to `PCB LEDS Design`.
 - b) In the Description section, add a comment of `PCB LED Design from PHY` as shown in Figure 10.
 - c) Select **Local Folder** from the *Save to* drop down.
 - d) Click **Create** to create the snippet and close the dialog.
25. Hit **Shift+S** until you're not longer in Single Layer.

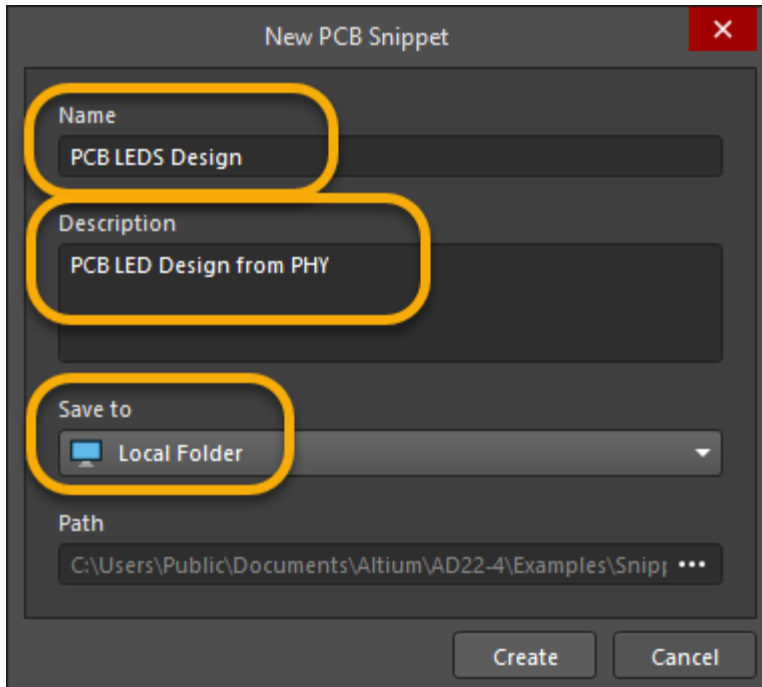


Figure 10. Creating PCB Snippet

26. Ensure that the *Selection Filter* is set to select all kind of objects again.

1.5 Using Snippets

1.5.1 Using the Snippets in a Project

Next, we will create a new project and use the snippets that we created.

27. Create a new project from the **File** menu and select **New » Project....**

28. Use Figure 11 as a reference and set the project values to the following:

- a) In the *Locations* section, select **Local Projects**.
- b) In the *Project Type* section, under the *PCB* category, select **<Empty>**.
- c) Change the *Project Name* to *Snippets*.
- d) Change the *Folder* location to a location of your choice.
- e) Select **Create** to create the project.

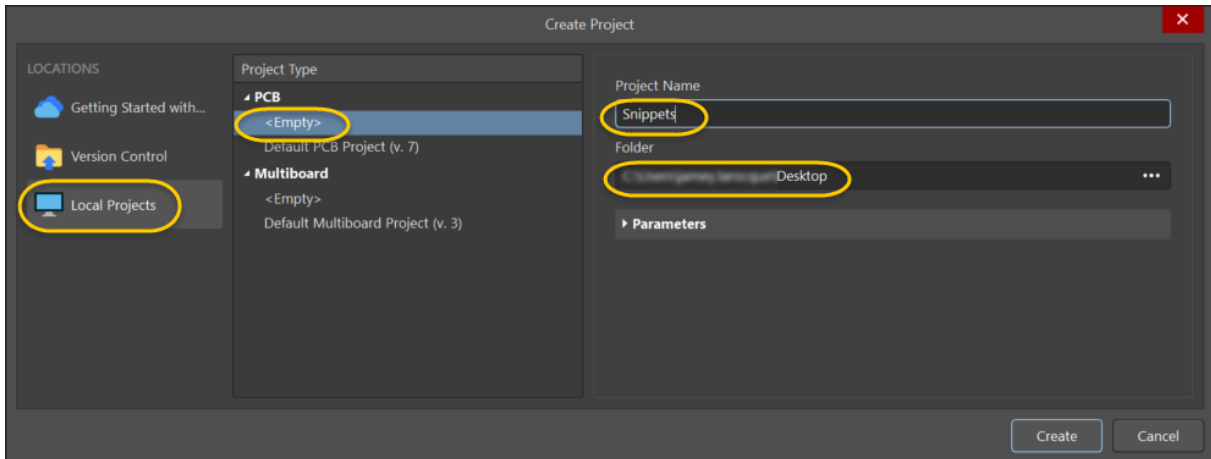


Figure 11. New Project dialog

29. Right click on the new `Snippets` project from the *Projects* panel and select **Add New to Project » Schematic** as shown in Figure 12. This will add a new schematic to the project with a default name of `Sheet1.SchDoc`.

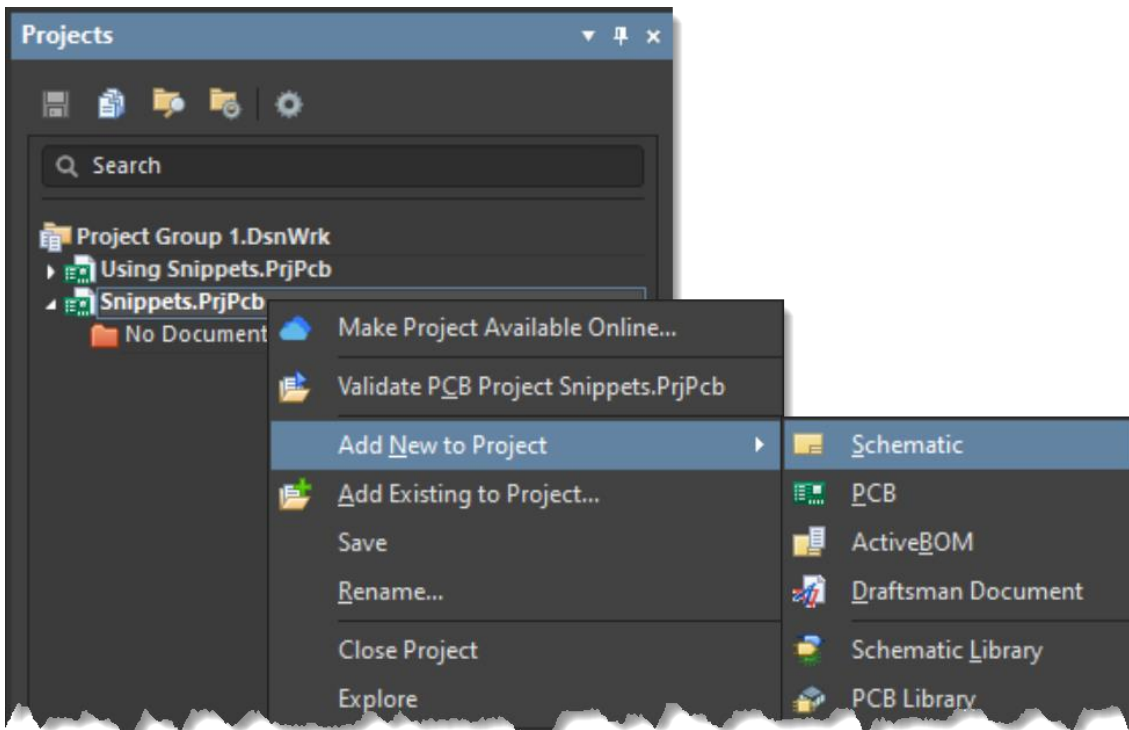



Figure 12. Adding New Schematics to the Project

30. Repeat the previous command and select **Add New to Project » PCB**. The default name will be `PCB1.PcbDoc`.
31. Right-click on the new `Snippets` project from the *Projects* panel, and select **Save**. Save the project to a location of your choice.
32. Rename the 2 files as the following:
 - a) Save the PCB file as `Snippets.PcbDoc`
 - b) Save the SCH file as `Snippets.SchDoc`

1.5.2 Designator Assignment for Snippets

Before placing the snippets into our design, we will need to enable a specific preference to ensure we maintain the synchronization of designators between the schematic and PCB.

33. Open the *Preferences*  from the upper-right corner of Altium Designer.
34. From the *Schematic* branch, in the *Graphical Editing* page, disable the option to **Reset Parts Designators On Paste** as shown in Figure 13. This will maintain the unique designators when we place the snippets into our designs.

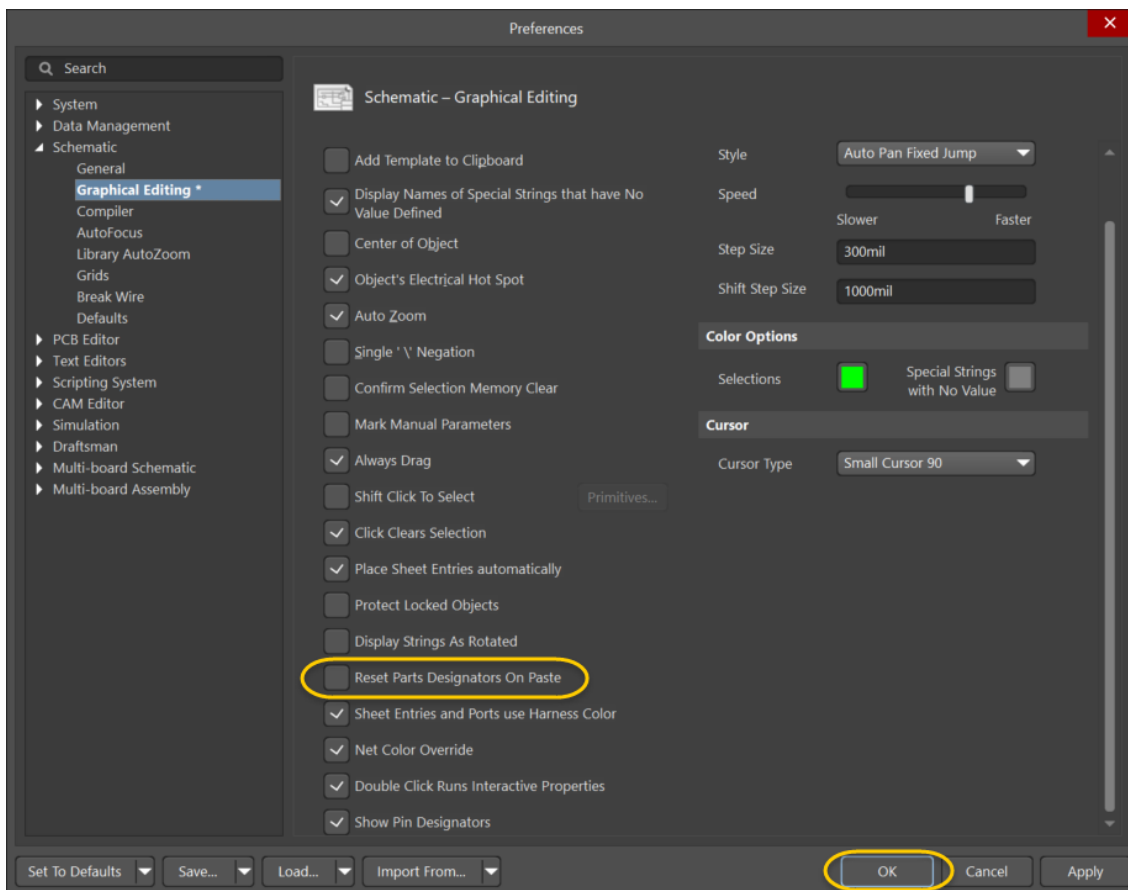


Figure 13. Reset Parts Designators preference

35. Click **OK** to apply these changes.

1.5.3 Adding Snippets to Project

36. Change the focus to the `Snippets.SchDoc` schematic document.
37. If not already opened, open the *Design Reuse* panel from the **Panels** button.
38. Select the schematic snippet in the *Design Reuse* panel.
39. Select **Place Schematic LED Design** button to place the snippet in the schematic as shown in Figure 14.

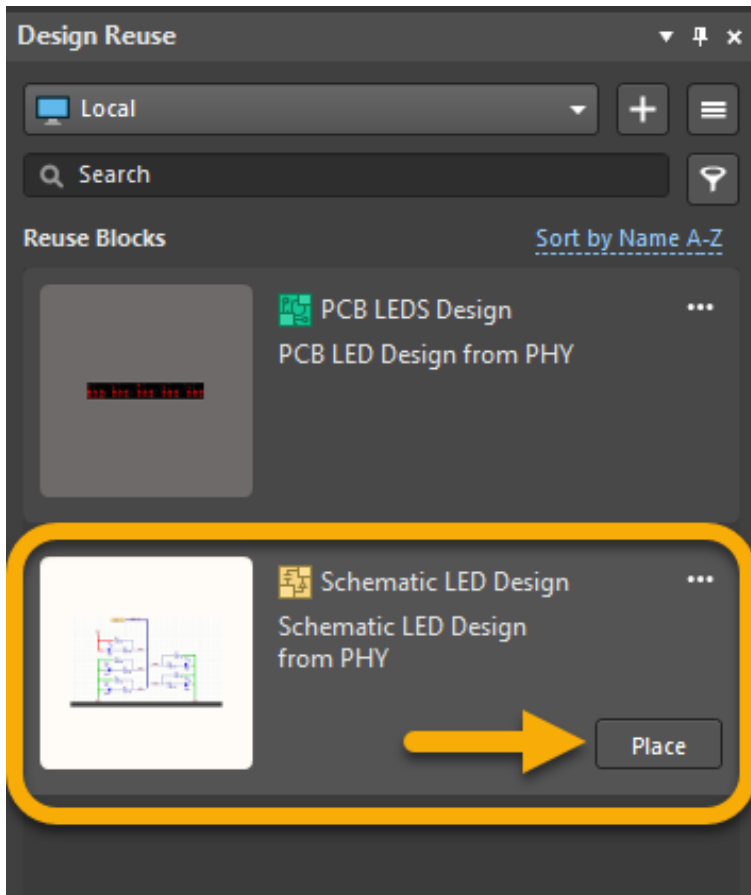


Figure 14. Placing Snippets

40. With the schematic snippet on your cursor, left-click to place it anywhere in the schematic document.
41. Change the focus to the `Snippets.PcbDoc`.
42. In the *Design Reuse* panel:
 - a) Click to select the `PCB LED Design` snippet.
 - b) Click on **Place PCB LEDs Design** to place the snippet into your PCB.
 - c) Left-click to place the snippet into the PCB document, your PCB should look similar to Figure 15.

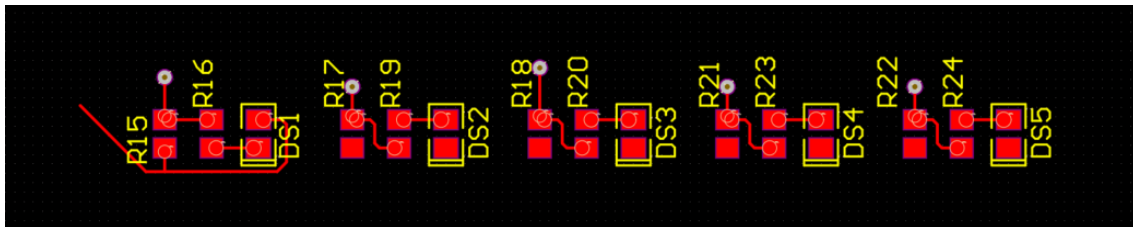


Figure 15. Snippet placed into the PCB

1.5.4 Synchronizing Snippets within the Project



When a component is placed on a schematic sheet, it is given a unique ID automatically. It is imperative to have all components matched using unique IDs, so that the annotation of designators in either the schematic or PCB document can be carried out. The documents can still be synchronized even if components aren't matched by unique IDs, but in this case, you will be prompted to match the components by designators to update the Unique ID.

43. In the PCB document, go to the **Project** menu and select **Component Links...**
 - a) In the *Edit Component Links* dialog menu, make sure both the **Designator** and **Footprint** checkboxes are enabled as shown in Figure 16.
 - b) Click the **Add Pairs Matched By »** button. All of the components should now show as matched in the rightmost section.
 - c) Click the **Perform Update** button.
 - d) Click **OK** when prompted in the *Information* dialog which shows that 15 links were modified.

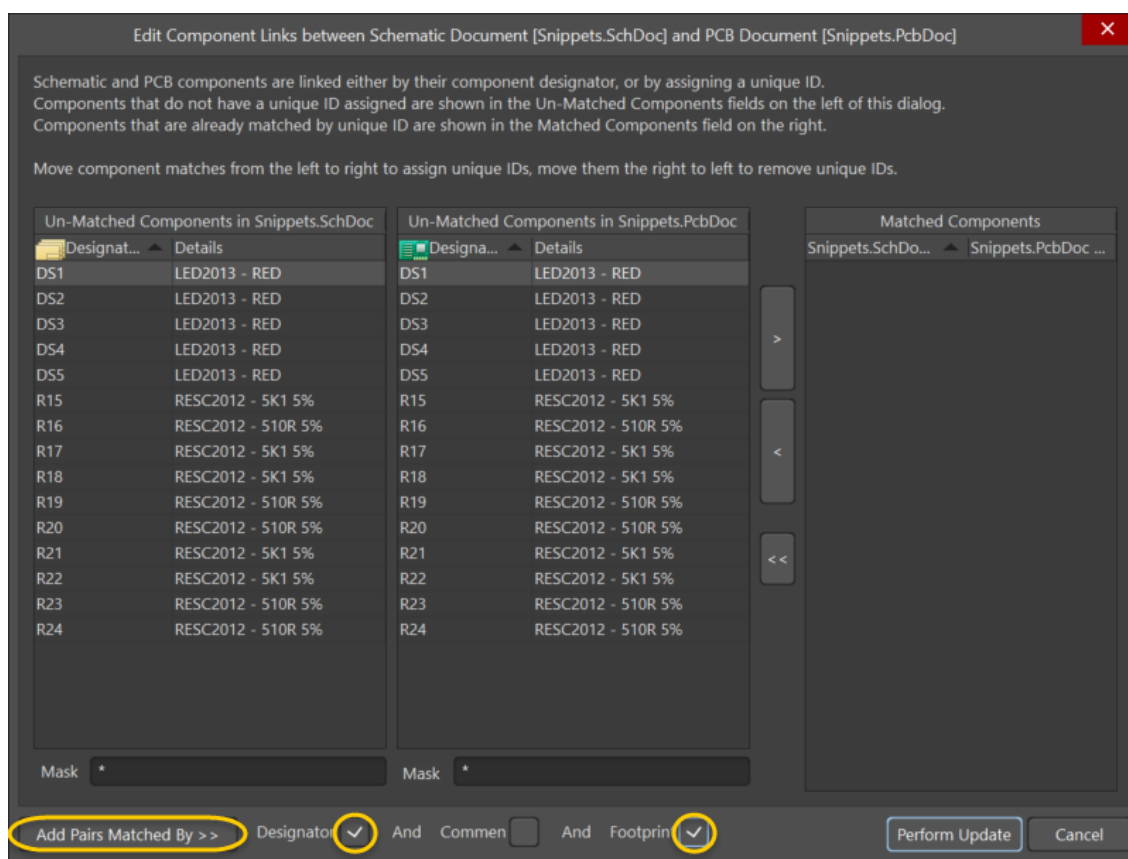


Figure 16. Component Links Menu

44. You have now successfully linked both the schematic and PCB snippets properly.

1.5.5 Creating Unique Designators

Up to this point in the exercise, the designators matched in our original snippets, however, if we had added them to an existing project, the designators may have conflicted. A better approach is to append a suffix to the designators for easy matching in the Component Linking process. If we would have skipped the Component Link matching step in the previous steps, there would be no easy way to match things up if designators change.

45. Open the `Snippets.SchDoc` schematic document.

46. From the **Tools** menu, select **Annotation » Annotate Schematics...**

47. For the following steps, refer to the information below as well as Figure 17:

- In the *Suffix* field, enter a value of `_LEDS`.
- In the *Start Index* cell, enter a value of 100.
- Enable the **Designator Index Control** checkbox as shown in Figure 17. These 3 settings will make it clear as to which components originated from a snippet.
- Click the **Reset All** button and click **OK** when prompted.
- Click the **Update Changes List** button and click **OK** when prompted. You can then see the proposed designators in the *Proposed* column of the dialog.
- After inspecting the proposed changes, click the **Accept Changes (Create ECO)** button.

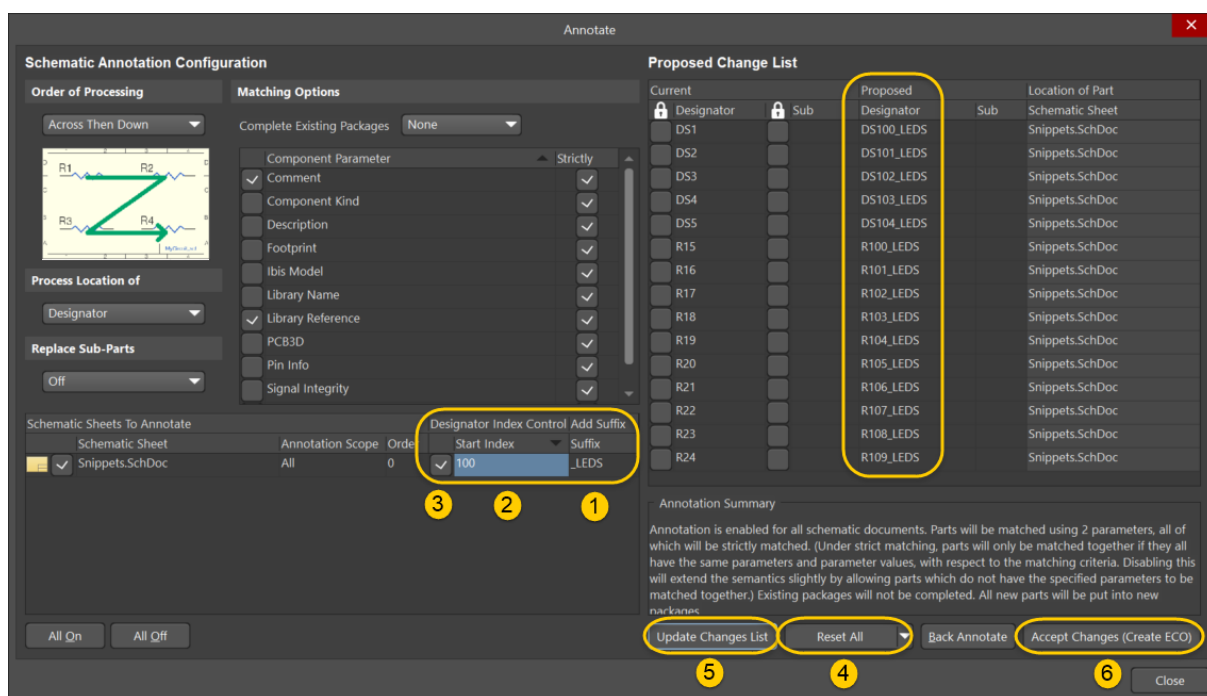


Figure 17. Annotation Options

48. When the *Engineering Change Order* dialog opens, click the **Execute Changes** followed by **Close**.

49. Click **Close** again to close the *Annotate* dialog.

50. We will update the PCB document to match the new designator changes by going to **Design » Update PCB Document Snippets.PcbDoc**.

51. Scroll down and disable the addition of Rooms by unchecking the **Add Rooms** checkbox as shown in Figure 18.

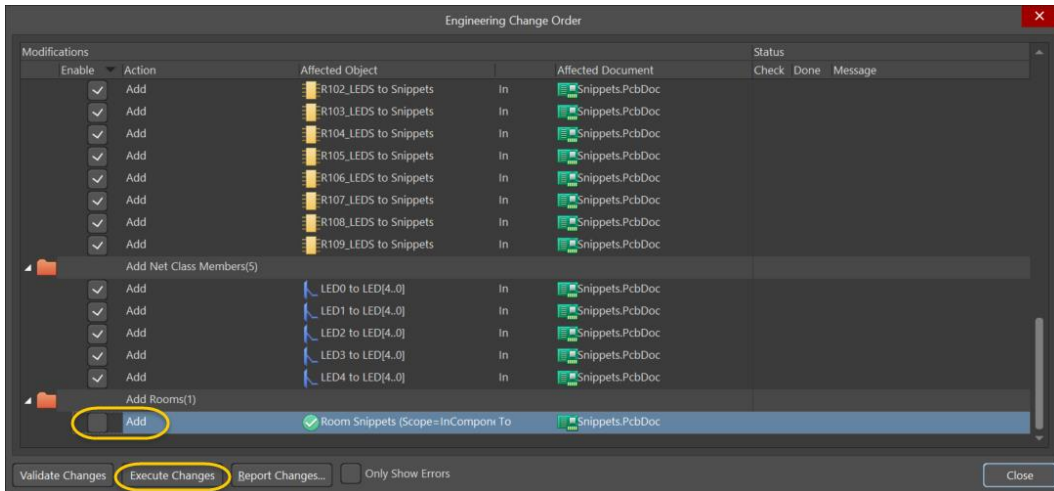


Figure 18. Engineering Change Order for new designators

52. Click **Execute Changes** in the *Engineering Change Order* dialog that appears followed by **Close**.

53. The designators in the SCH and PCB will now look similar to what is shown in Figure 19

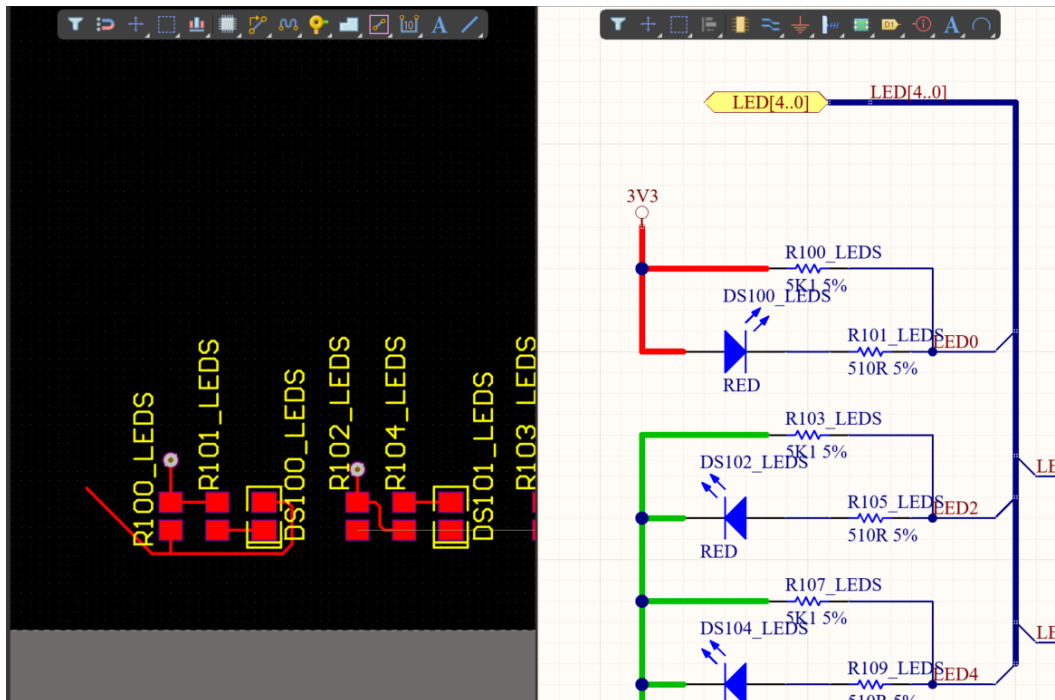


Figure 19. Designator synchronization between the SCH and PCB



If Snippets were created using this designator formatting scheme, the snippets could be inserted into any existing design.

54. Feel free to save your modifications.

55. **Close the project and any open documents.**

Congratulations on completing module

Using Snippets

from the
Altium Designer Advanced Course

Thank you for choosing Altium Designer