





Altium Designer

Advanced Training with Altium 365
PCB Layout Replication









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.





PCB Layout Replication



Table of Contents

P	PCB Layout Replication 3			
1	P	rpose 3		
2	2 Shortcuts		3	
3	Pı	Preparation		
4	Overview		5	
	4.1	Preparation	5	
	4.2	Selecting the PCB items from Schematic	7	
	4.3	Selecting PCB Tracks	7	
	4.4	Replication	9	
5	R	eplicating SFP Circuit	11	
	5.1	Selecting PCB Items	11	
	5.2	Replication	12	







PCB Layout Replication

1 Purpose

With the PCB Layout Replication tool, you can replicate the layout for repetitive blocks of circuitry in a flat PCB design. As an informal reuse feature, this tool is a perfect fit when you need to quickly replicate placement of a group of components to another group of the same, not-yet-placed components with the same connections.

2 Shortcuts

Shortcuts used when working with PCB Layout Replication

Spacebar	Rotate the target block counterclockwise
Shift+Spacebar	Rotate the target block clockwise.
L	Flip Board side for Target Block
S	Open the selection Menu
S»S	Physical Connection Single Layer





3 Preparation

- 1. Close all existing projects and documents.
- 2. Next, create a copy of the Training Project: PCB Layout Replication.
- 3. Select File » Open Project... to open the Open Project dialog.
- 4. Enable the folder view button
- 5. Navigate to the predefined Training Project PCB Layout Replication (Top\Projects\Altium Designer Advanced Training Course\...).
- 6. Select **Open Project as Copy...** Open Project As Copy...
- 7. In the new dialog Create Project Copy:
 - a) Add your name to the project name: PCB Layout Replication [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.







4 Overview

First, you will replicate the circuit Ethernet. The circuits for Ethernet 1 and Ethernet 2 are on two individual schematic pages.

4.1 Preparation

9. Open the Schematic Ethernet 1.schdoc and Ethernet 2.schdoc.

The Circuits for Ethernet 1 and Ethernet 2 are identical, as seen in Figure 1.

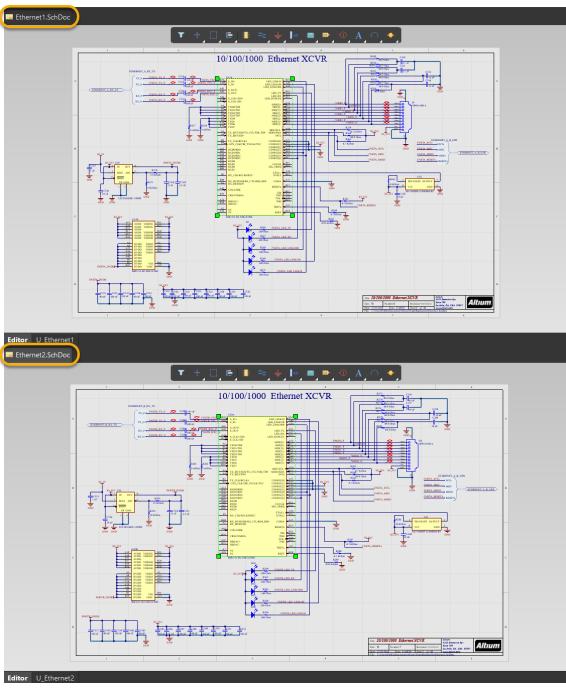


Figure 1. Ethernet 1.schdoc and Ethernet 2.schdoc







- 10. Open the PCB MiniPC. PcbDoc.
- 11. At the lower left side of the PCB, you will see the two Ethernet connectors. The circuit for Ethernet 2 is placed and routed. For Ethernet 1, just the connector is placed, the other components still need to be placed and routed, see Figure 2.

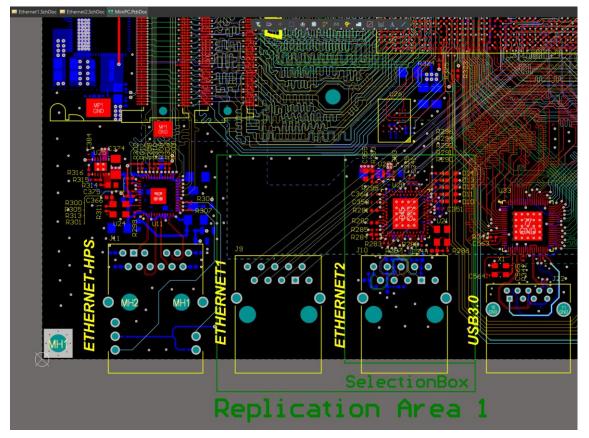


Figure 2. MiniPC - Ethernet1 and Ethernet 2





4.2 Selecting the PCB items from Schematic

- 12. Change focus back to the Schematic Ethernet 2.schdoc.
- 13. Select with Ctrl+A all objects from Ethernet 2.schdoc.
- 14. Select **Tools** » **Select PCB Components**. Altium Designer selects the PCB components in the PCB document for the symbols selected in the schematic.
- 15. Change the focus back to the PCB document.
- 16. You will see that all footprints in PCB are now selected.

4.3 Selecting PCB Tracks

17. With the footprints selected, press **Tab** key to extend the selection to the routed tracks connected with the selected footprints.

Hint: TAB key selects the tracks from the component pad to the first layer change. If you want to include Tracks on Bottom Layer, select these tracks manually.

- 18. To select additional tracks, you need to include the PCB replication command:
 - a) Select S » S for the command Physical Connection Single Layer.
 - b) Press and hold **Shift** key.
 - c) Add the Differential Pairs from the U20 to J1, 0 routed on the Inner Layer. See Figure 3 as reference for the four Differential Pairs you add to the existing selection.

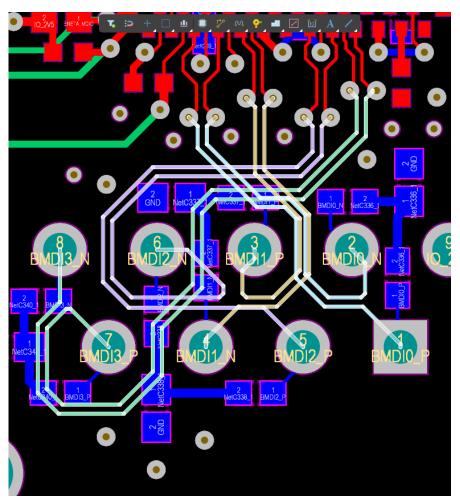


Figure 3. Reference for Differential Pairs selection





d) With the selection tool **Physical Connection Single Layer** still active and **Shift** key selected, add four signals on Bottom Layer to the existing selection. Use Figure 4 as reference for the four Signals to add.

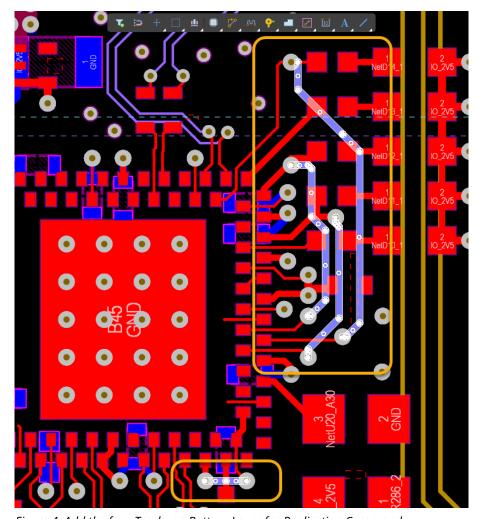


Figure 4. Add the four Tracks on Bottom Layer for Replication Command







4.4 Replication

- 19. Select **Tools » PCB Layout Replication** to start the replication.
- 20. Wait until the tool prepares the PCB Layout Replication dialog, Figure 5.

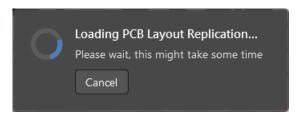


Figure 5. PCB Replication preparation

- 21. In the dialog PCB Layout Replication:
 - a) Select J10 as reference.
 - b) Activate the following options:
 - i) Copy routed nets
 - ii) Copy Designator & Comment formatting
 - iii) Copy unrouted objects
 - c) Select **Replicate** to start the replication process.

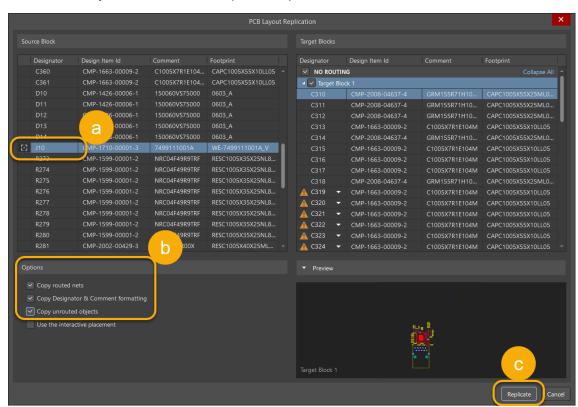


Figure 6. Replication Dialog

Note: The Capacitors at the right side with a symbol _____ are Backup Caps without a unique signal path. This means that for the replication tool, these Capacitors don't have specific position or designator pair information and any Capacitor from that group could be used at any position.

Altıum







- 22. To see the result of the replication command (Figure 7):
 - a) Zoom out to see the complete PCB.
 - b) Select the newly replicated components.
 - c) Move the components to the right position.

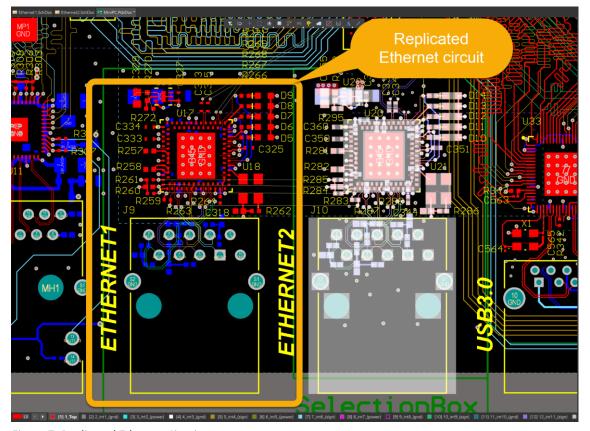


Figure 7. Replicated Ethernet Circuit

Hint: Instead of starting the selection in the SCH with **Select » PCB Components**, you can select the components and routing with a mouse selection box (Left to Right – Fully Include) in the PCB. We added a reference for the selection box on a mechanical layer as reference. Similar to the flow described in the training, you may need to include or exclude objects after the initial selection.

Selecting the footprints and tracks in the PCB is the method for the second circuit SFP of the training.





5 Replicating SFP Circuit

Next you will replicate the circuit SFP. The two circuits for SFP-A and SFP-B are on one schematic page.

23. Open the Schematic SFP_PortA-B.SchDoc to see the two circuits on one page.

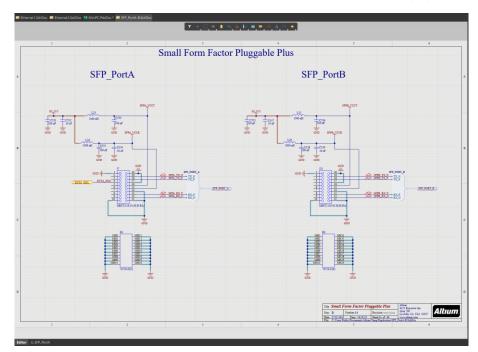


Figure 8. SFP Port A and SFP Port B

5.1 Selecting PCB Items

- 24. Change the focus back to the PCB document and zoom to the lower right corner with the circuits SFP-A and SFP-B.
- 25. Select with a mouse rectangle (left to right fully include selection) all element inside the reference rectangle we added on a mechanical layer, Figure 9.

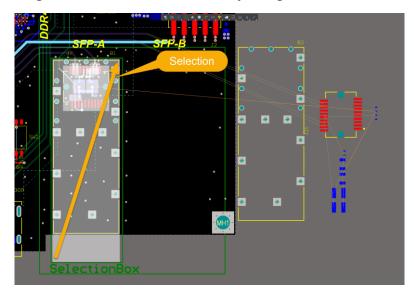


Figure 9. Select SFP-A items





5.2 Replication

- 26. Execute the command Tools » PCB Layout Replication... to start the replication.
- 27. Wait until the tool prepares the PCB Layout Replication dialog, Figure 10.

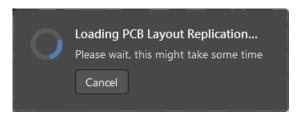


Figure 10. PCB Replication preparation

- 28. In the PCB Layout Replication dialog, Figure 11:
 - a) Change the reference to **B1**. The preview is updated with a 90° rotated placement.

Hint: If B1 isn't available, cancel the process, go back to the PCB, and redo the selection with a slightly larger selection Box.

- b) Activate the following options:
 - i) Copy routed nets
 - ii) Copy Designator & Comment formatting
 - iii) Copy unrouted objects
 - iv) Use the interactive placement
- c) Select **Replicate** to start the replication.

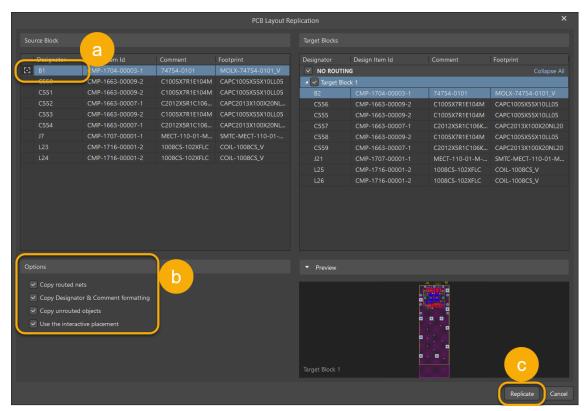


Figure 11. Replication for SFP-B

29. With the option **Use the interactive placement**, the Circuit SFP-A is attached to the cursor for placement.

Altıum





30. After the replication completes, the circuit is attached to the cursor. Press Spacebar to position the circuit SFB-B to its correct orientation, as seen in Figure 12.

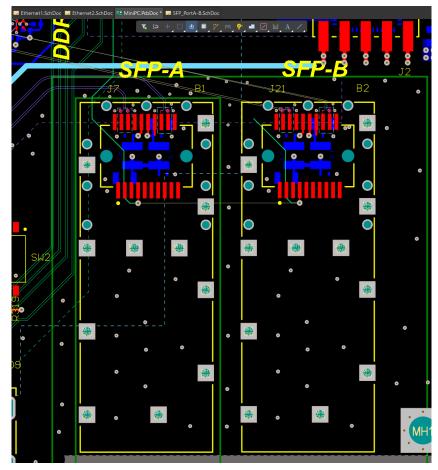


Figure 12. Position for SFP-B

31. Delete the track and via from J21- Pin 4. This pin has no net for SFP-B but it was included in the selection and therefore is a part of the replication step.

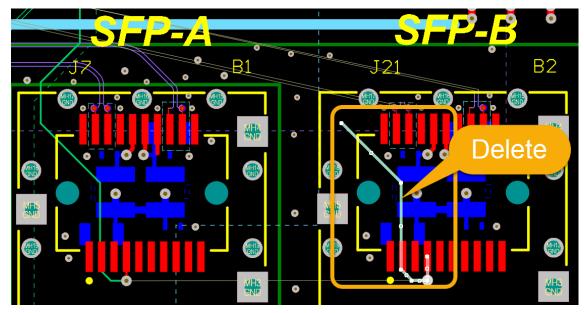


Figure 13. Delete not used Track and Via





- 32. Save all documents using File » Save All.
- 33. Save the modifications to the server:
 - a) In the *Projects* panel, next to the Project name you find the command **Save to Server**Save to Server
 - b) Select Save to Server.
 - c) In the dialog Save [Project Name]:
 - i) Add the comment PCB Layout Replication [Add Your Name] -Finished.
 - ii) Select **OK**.
- 34. When ready, close the project and any open documents, Window » Close All.







Congratulations on completing the Module!

PCB Layout Replication

from

Altium Designer Advanced Training with Altium 365

Thank you for choosing **Altium Designer**



