Altium Designer

Advanced Training with Altium 365

PCB Layout Replication

Software, documentation and related materials:

Table of Contents

PCB Layout Replication

1 Purpose

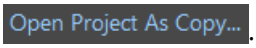
With the PCB Layout Replication tool, you can replicate the layout for repetitive blocks of circuitry in a flat PCB design. As an informal reuse feature, this tool is a perfect fit when you need to quickly replicate placement of a group of components to another group of the same, not-yet-placed components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

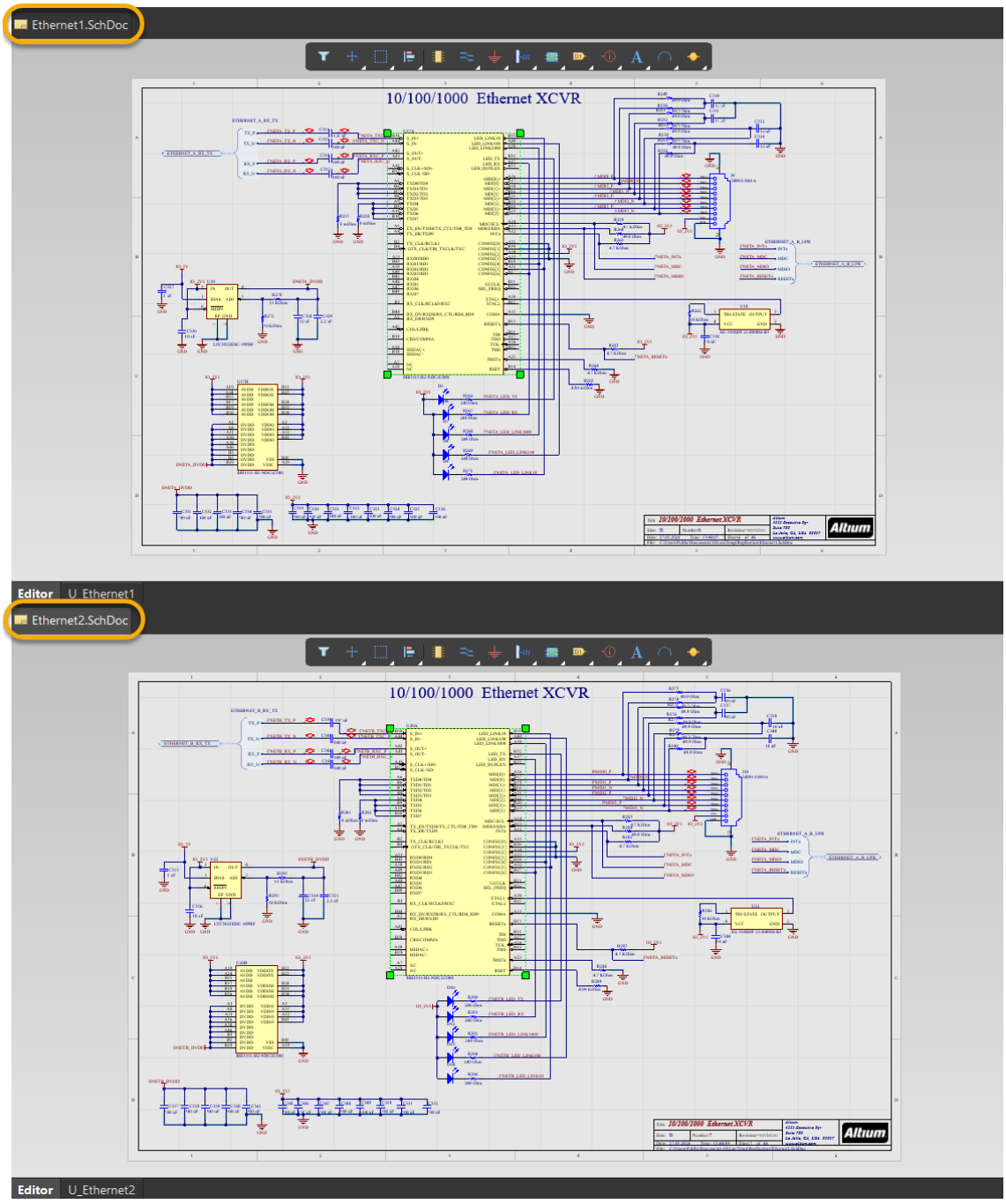


Figure 1. Ethernet 1.schdoc and Ethernet 2.schdoc

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.

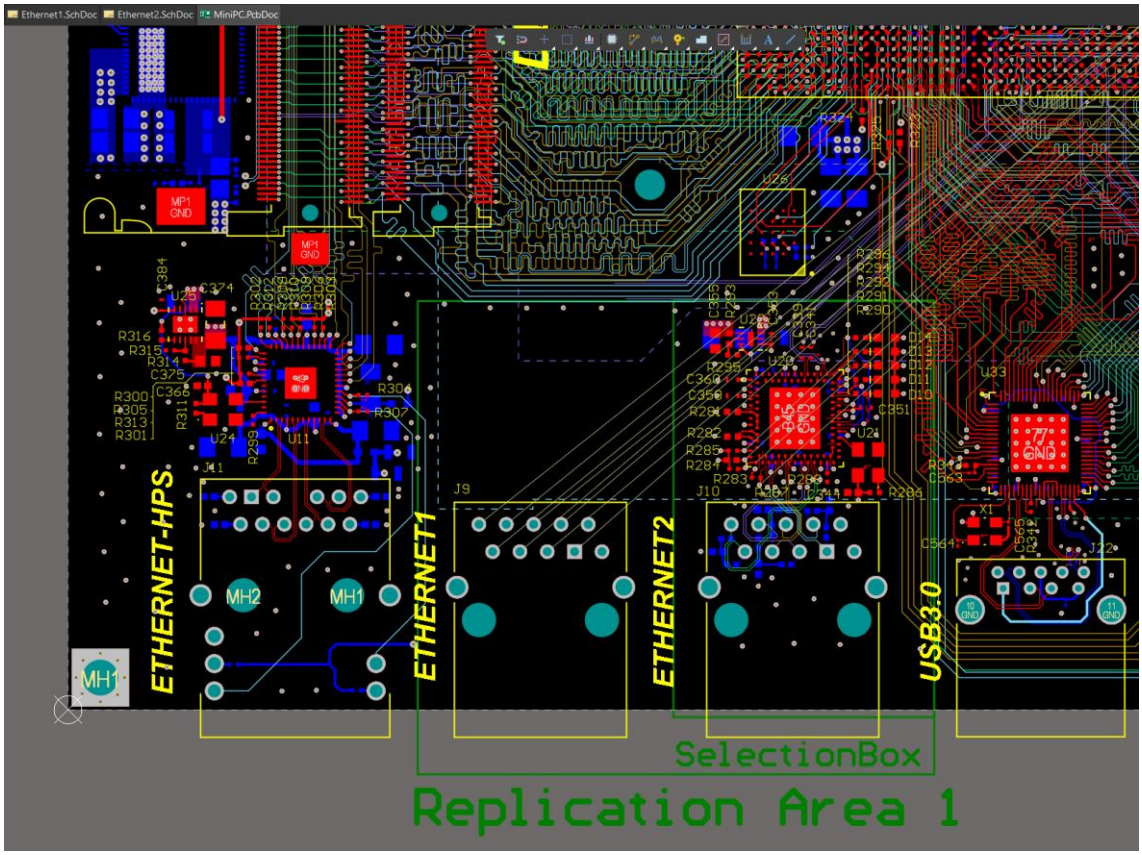


Figure 2. MiniPC - Ethernet1 and Ethernet 2

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.

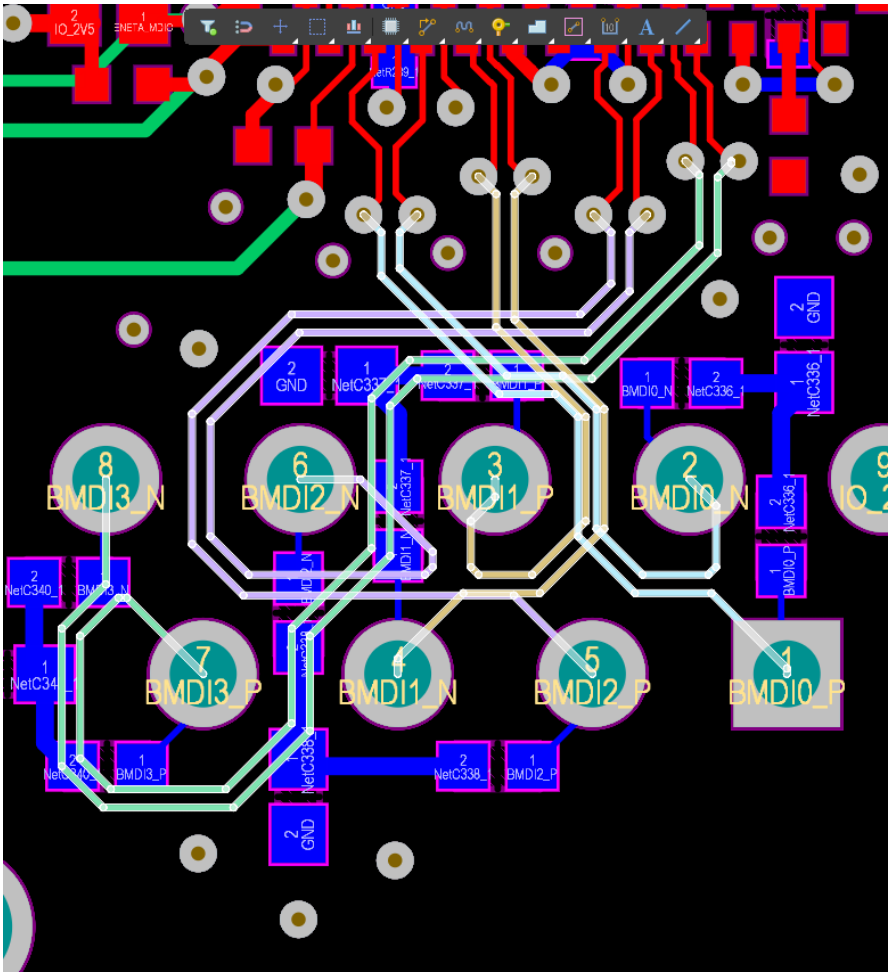


Figure 3. Reference for Differential Pairs selection

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.

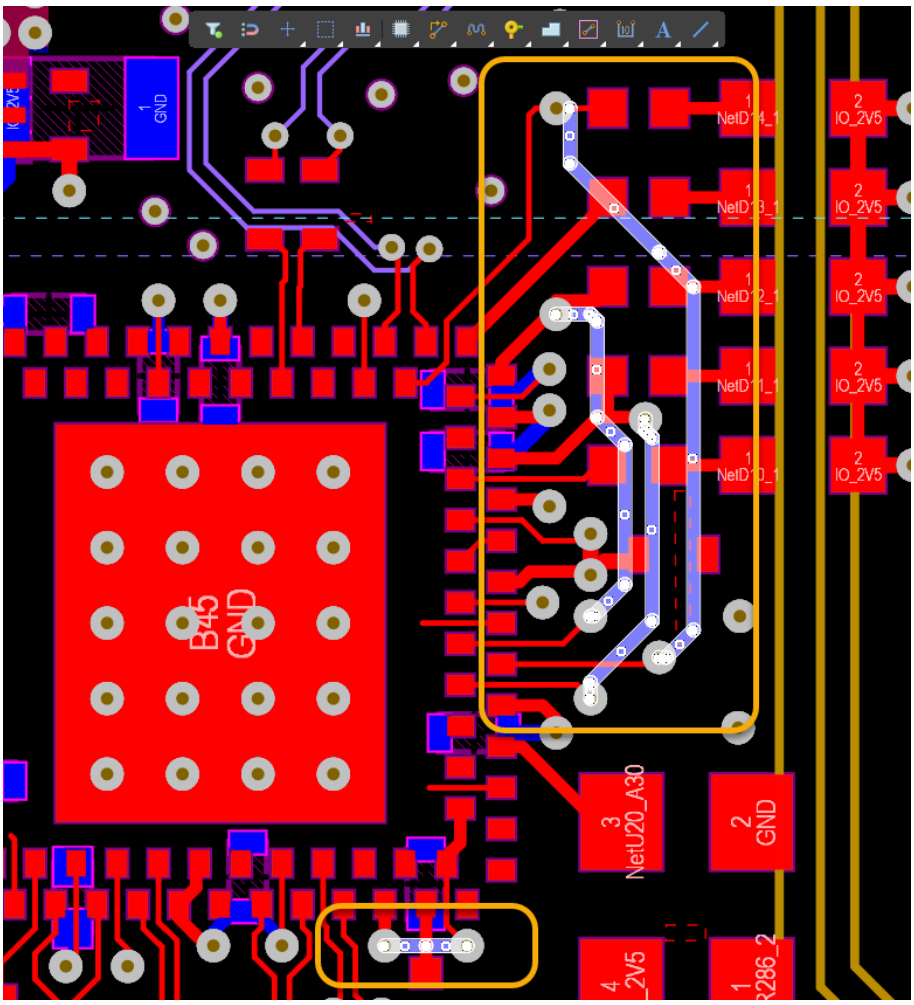


Figure 4. Add the four Tracks on Bottom Layer for Replication Command

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.

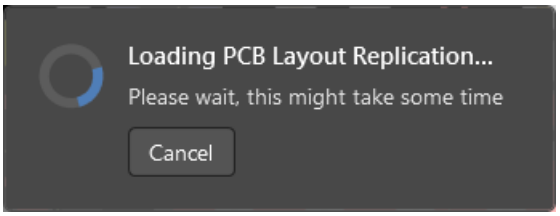


Figure 5. PCB Replication preparation

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.

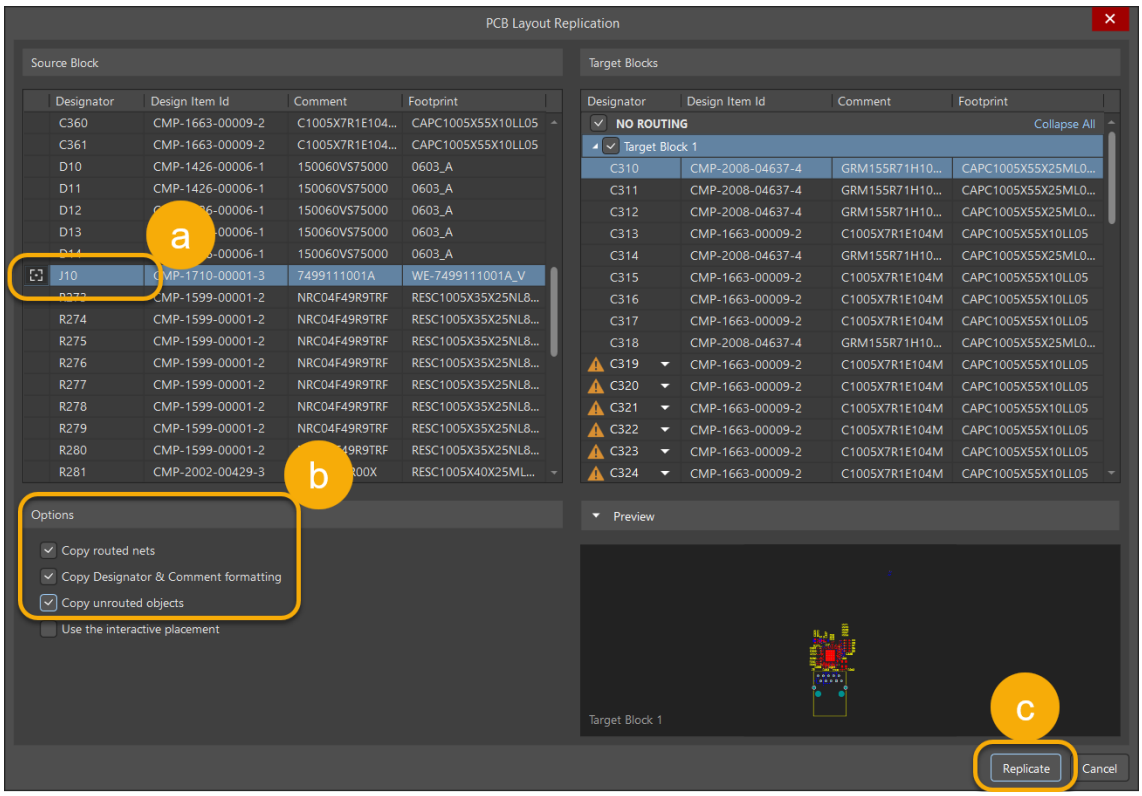


Figure 6. Replication Dialog

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.

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.

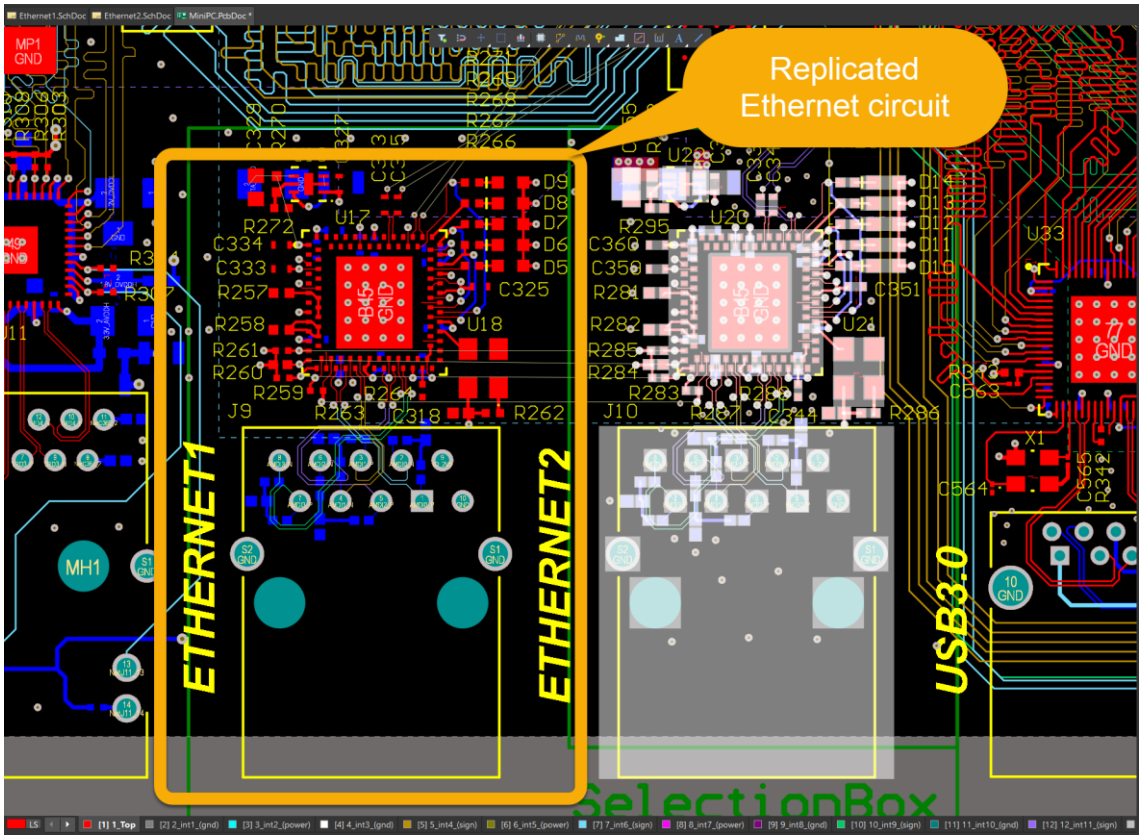


Figure 7. Replicated Ethernet Circuit

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.

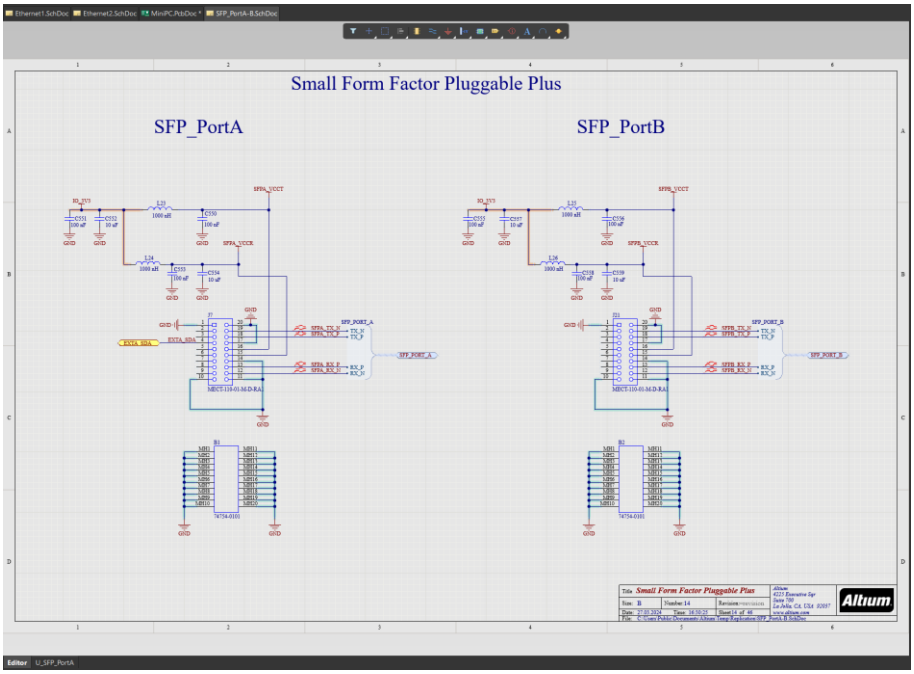


Figure 8. SFP Port A and SFP Port B

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.

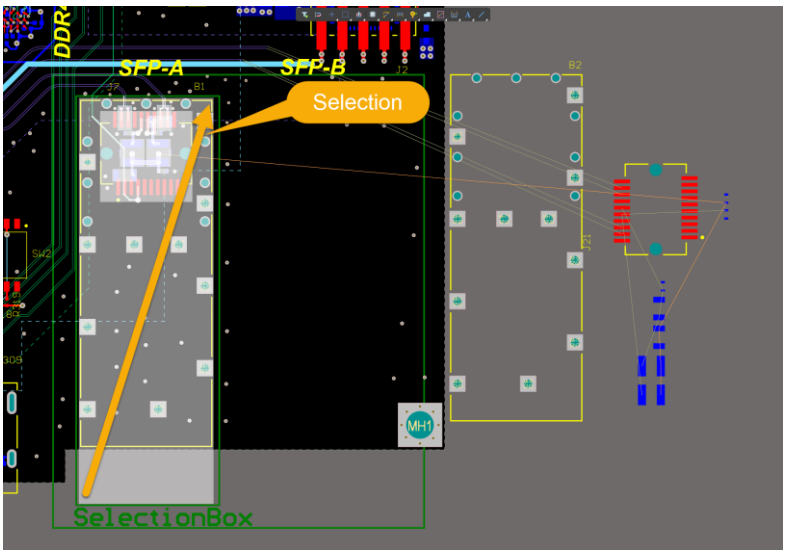


Figure 9. Select SFP-A items

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.

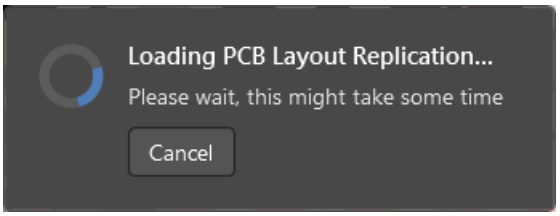


Figure 10. PCB Replication preparation

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.

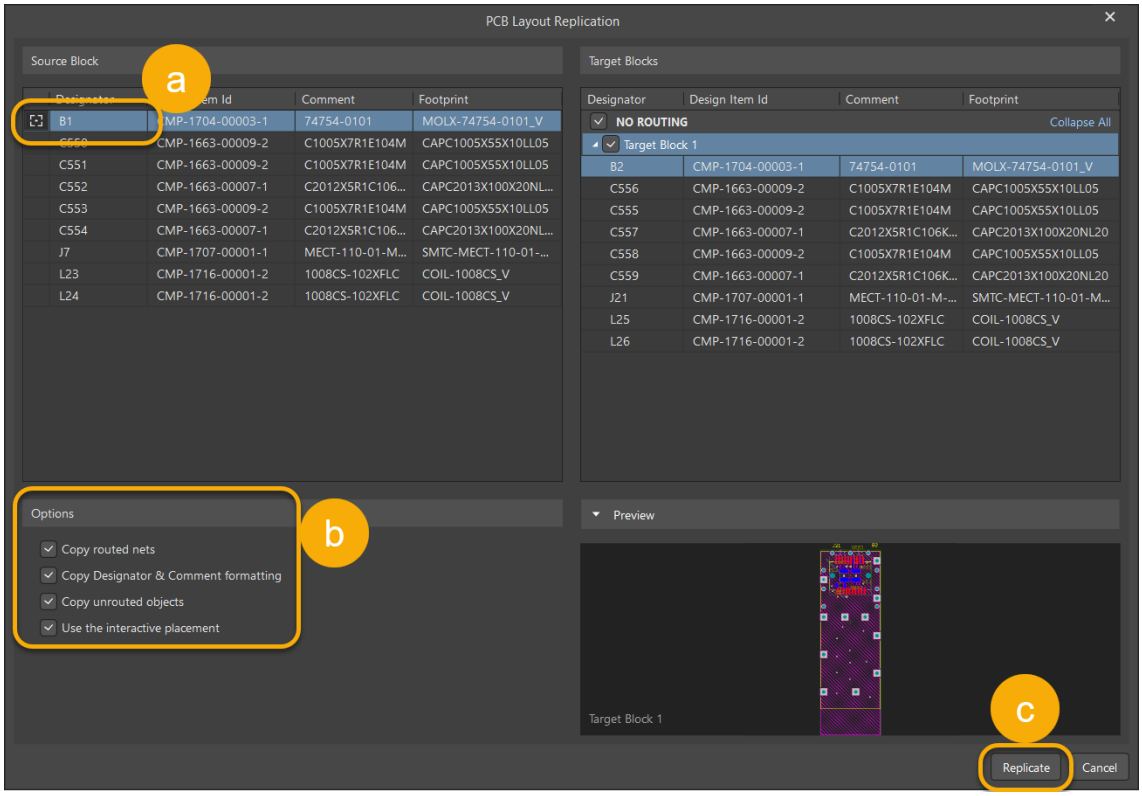


Figure 11. Replication for SFP-B

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

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.

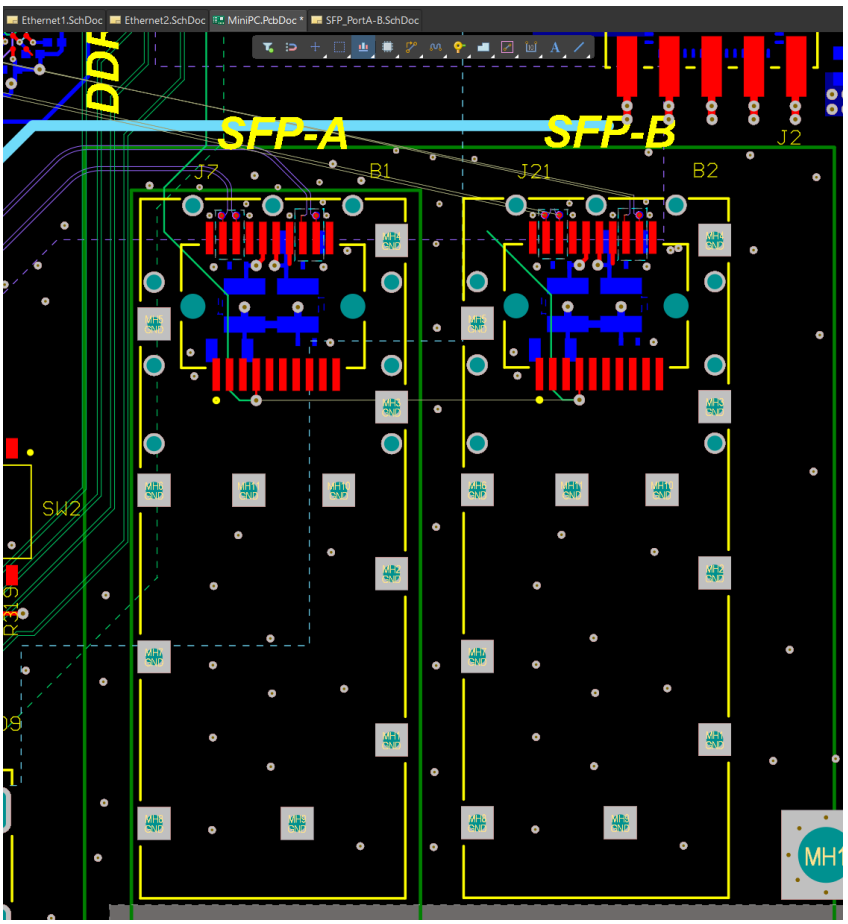


Figure 12. Position for SFP-B

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.

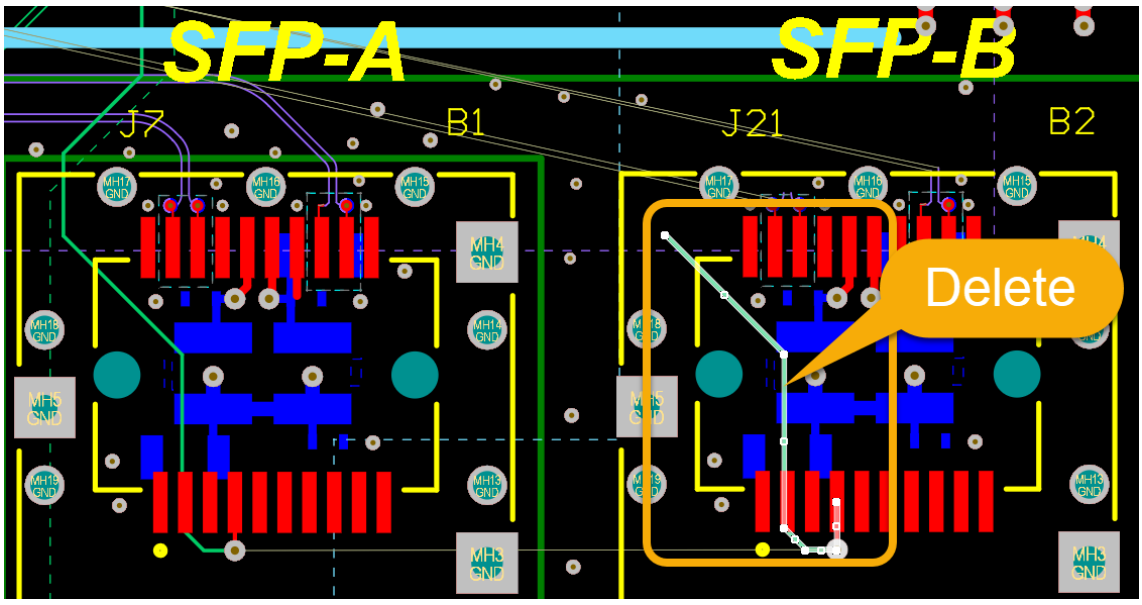
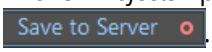


Figure 13. Delete not used Track and Via

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**  .
    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**