Altium Designer

Advanced Training with Altium 365

Pin Swapping

Software, documentation and related materials:

Table of Contents

Pin Swapping

1 Purpose

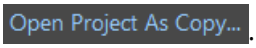
The pin, differential pair, and part swapping system work in harmony with Altium Designer interactive routing and BGA escape routing capabilities. The pin swapping feature provides all the benefits of traditional pin swapping systems but takes advantage of Altium Designer intimate understanding of the net assignments in the design. During a pin swap operation, Altium Designer analyzes the net assigned to a selected pin, and dynamically reassigns the net on the pin and any connected copper. This module shows how to set up pin swapping in preparation for interactive routing.

2 Shortcuts

Shortcuts used when working with Pin Swapping

F1	Help
C » O	Project Options
T » W » C	Swap Configuration
T » W » A	Automatic Net/Pin Optimizer...
D » U	Update Schematic...
CTRL+S	Save Document

3 Preparation

1. Close all existing projects and documents.
2. Next, create a copy of the Training Project: `Pin Swapping`.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Enable the folder view button  .
5. Navigate to the predefined Training Project `Pin Swapping (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: `Pin Swapping – [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 Pin Swapping

4.1 Overview

Preliminary routing consists of creating net stubs for escape routing from configured components that have swappable pins and routing components that connect to the swappable nets. These routes, while not making the full routed connection, are close enough so that the Net/Pin optimization can then be performed to help uncross the traces, to provide a direct route completion path, as shown in Figure 1.

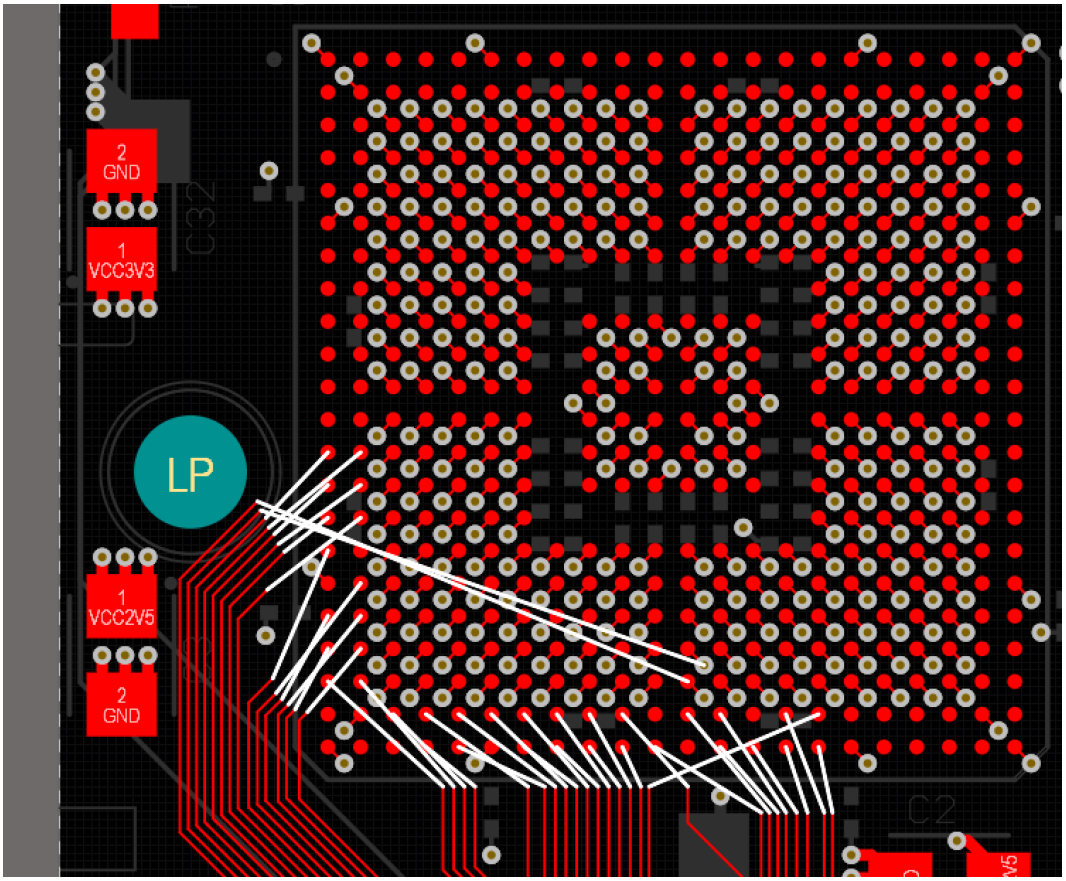


Figure 1. Direct route completion with Pin Swapping

Caution: You need to configure pin swaps for the component before running the Net/Pin optimizer.

After the Fanout process is complete, the Net cross-status can be quite severe. This is due to the fact the Net allocation and component placement was largely determined during schematic portion of the design process. Resolving this routing complexity has been a major challenge in the past. Fortunately, Altium Designer has powerful Pin/Net Swapping features that help designers redefine the FPGA I/O Net configurations automatically, and the changes can be updated back to the schematic.

4.2 Pin Swapping Configuration

9. Open the PCB by double-clicking on the `NBP-28 Pin_Swap_Example.PcbDoc` and `FPGA.SchDoc` schematic files in the *Projects* panel.
10. First, restrict the swapping allowed to exclude moving Schematic Pins. This prevents the component on the schematic becoming out of sync with the original library definition. Select **Project » Project Options** from the main menu.
    a) In the *Options* tab, locate the *Allow Pin Swapping Using These Methods* section and make sure only **Adding/Removing Net-Labels** is checked as shown in Figure 2. **Changing Schematic Pins** shouldn't be selected.
    b) Select **OK** to quit the dialog.

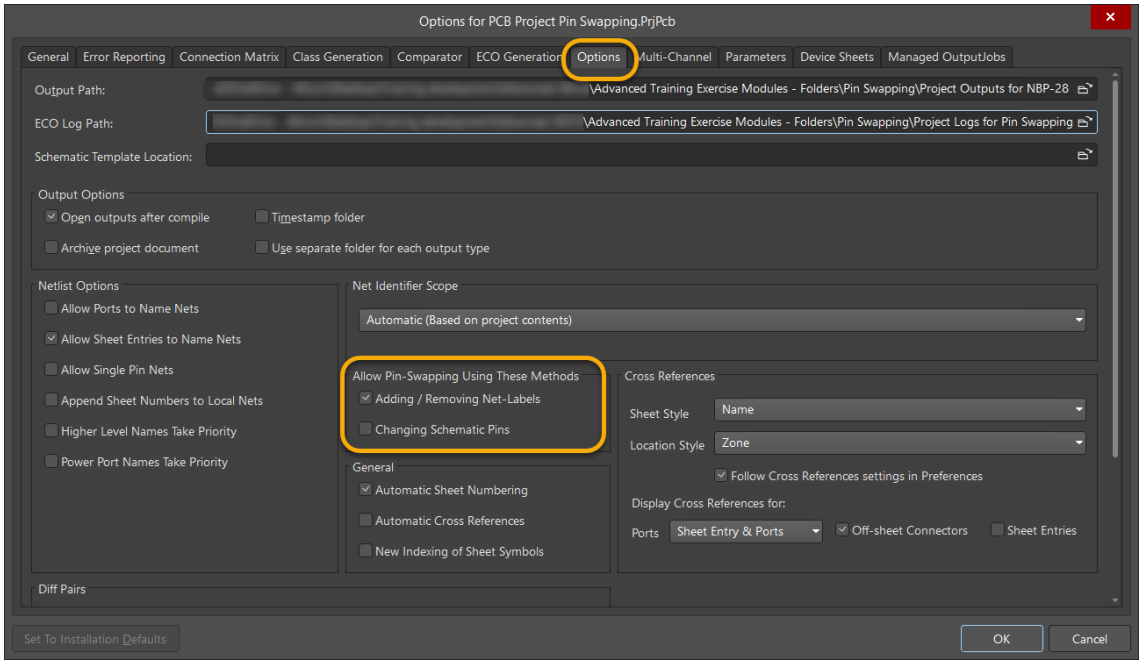


Figure 2. Project Options » Options Tab

11. Locate the FPGA device `U1`, in the PCB Editor.
12. Zoom in and put the FPGA in the center of the viewable area.
13. Double-click on the device to open its properties in the *Properties* panel.
14. Look for the Swapping Options section and tick the **Enable Pin Swaps** checkbox, as shown in Figure 3 below.

> Hint: You can also do this in the Pin/Part dialog after defining the swap groups.

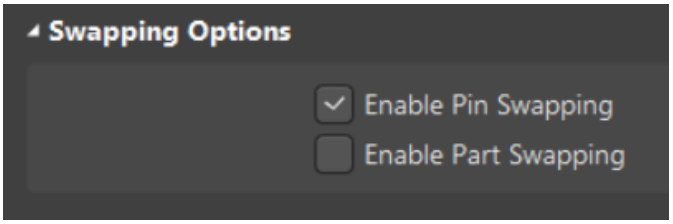


Figure 3. Component Properties U1

15. Select **Tools » Pin/Part Swapping » Configure** to open the *Configure Swapping Information In Components* dialog. Locate and select the FPGA (U1A) in the list and select the **Configure Component...** button (lower left), as shown in Figure 4. If the FPGA isn't listed, ensure that the **Only Show Components with Swap Information** checkbox is unchecked.

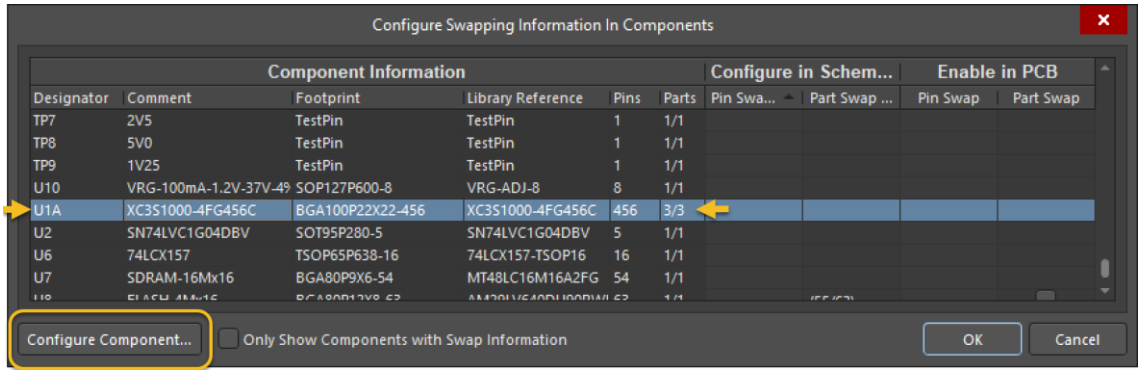


Figure 4. Configure Swapping Information In Components

16. The *Configure Pin Swapping* information for `U1` appears, as shown in Figure 5. Select the *Pin Swapping* tab located in the upper left corner.

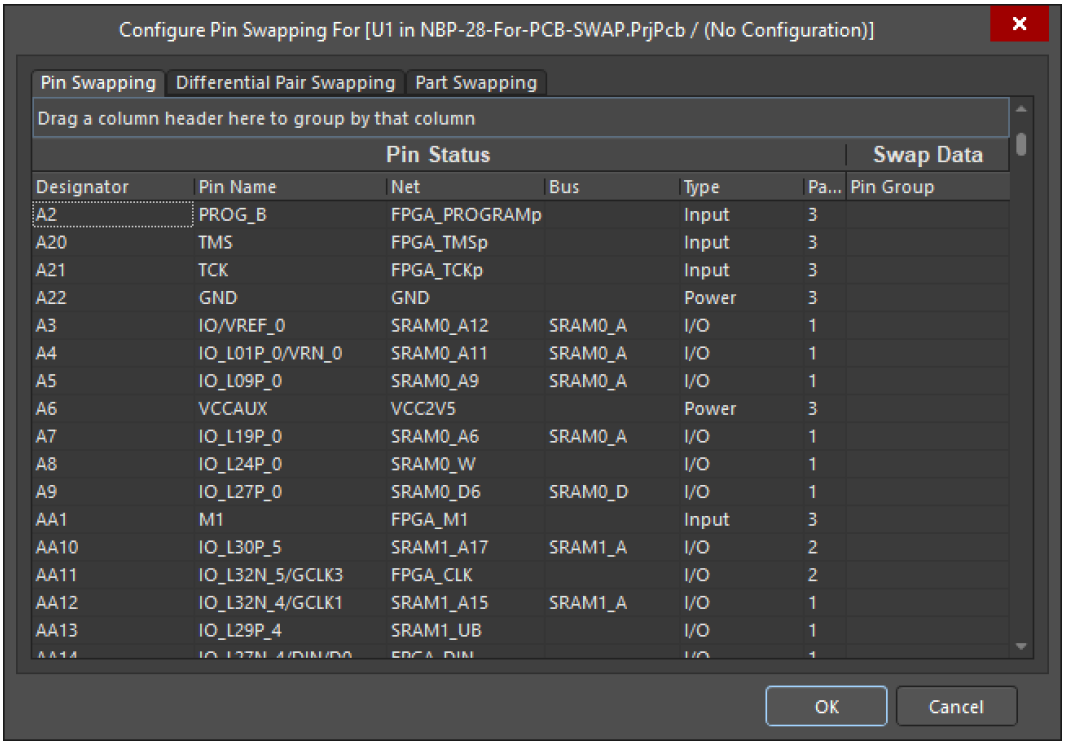


Figure 5. Pin Swapping Configuration

17. Select the *Pin Name* header label at the top to sort the list by pin name. All the pins with prefix IO will be listed together. Select all the rows with the IO_ prefix with the **Shift + Click** keys.
18. Right-click the list and select **Add To Pin-Swap Group » New**. Note that the pins are now allocated to `Group 1` in the Pin Group column.

Hint: All FPGA pins enabled for swapping are included in Group 1 and can be freely swapped within this group. Make sure pins that shouldn't have their net changed (swapped) aren't included in the group.

19. Remove the pin with net FPGA_CLK, FPGA_Done, FPGA_CCLK from `Group 1`. Right-click on its row and select **Remove From Pin Swap Group**.
20. Select **OK** to finalize the Pin Swapping configuration.
21. Ensure that the **Pin Swap** checkbox is enabled for U1A, as shown in Figure 6, and select **OK**.

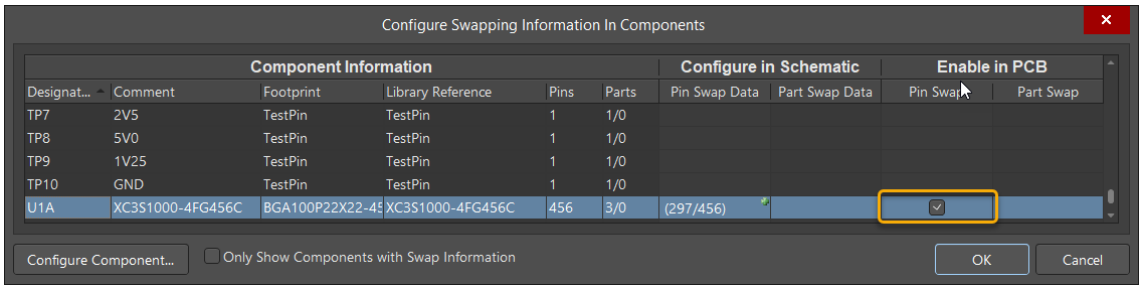


Figure 6. Pin Swap Enabled for U1A in Configure Swapping Information In Components Dialog

22. Select **OK** when prompted to update 3 Sub-Parts in Schematic Sheets, as shown in Figure 7.

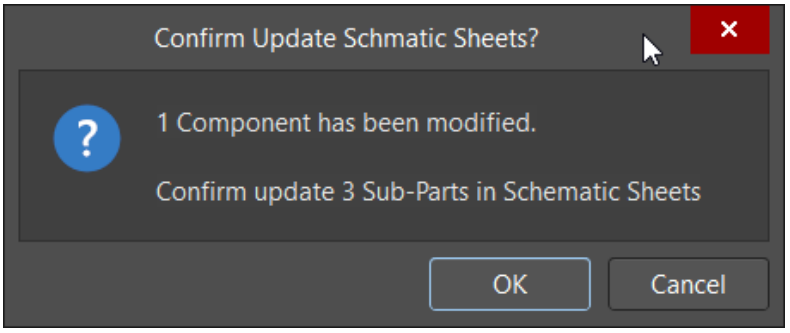


Figure 7. Confirm Update Schematic Sheets

23. Select **File » Save All**.

4.3 Automatic Pin Swapping

24. Now, run the pin swapping optimizer by selecting **Tools » Pin/Part Swapping » Automatic Net/Pin Optimizer**.
    a) Select **Yes** to continue with the iterative optimizer when prompted.
    b) Results of the optimization are summarized in Figure 8.

Hint: To re-optimize the connections, you can re-run the Net/Pin Optimizer if there are any changes in the copper layout.

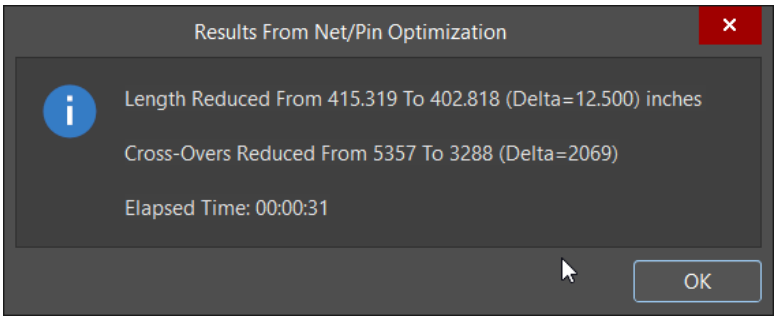


Figure 8. Results from Net/Pin Optimization

Question: Check the PCB and note what changes have occurred to the FPGA nets and wiring. Is this the result you expected? How long would this process have taken if manual pin swapping methods had been used to simplify the connections?

4.4 Swap Information in the Schematic

25. For the last step, from the PcbDoc, select **Design » Update Schematics in Pin Swapping – [Your Name].PrjPcb** and select **Yes** when prompted by the *Comparator*.
26. Select **Execute Changes** in the *Engineering Change Order* dialog.
27. Once the *ECO* is finished, close the dialog and select **File » Save All**.
28. Switch to the `FPGA.SchDoc` to see that the Net labels have changed by executing **Edit » Undo** and then **Edit » Redo**.
29. Save all your work and close all design files.

> Caution: The assigned pin swaps in this example were arbitrary, and valid swappable pins should be set and verified as defined in the manufacturer's datasheet.

30. Select **File » Save All** to save all modifications.
31. 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) AIn the dialog *Save [Project Name]*:
        i) Activate the checkboxes for the files that are not under version control.
        ii) Add the comment `Pin Swapping - [Add Your Name] - Finished`
        iii) Select **OK**.
32. When ready, close the project and any open documents, **Window » Close All**.

Congratulations on completing the Module!

Pin Swapping

from

Altium Designer Advanced Training with Altium 365

Thank you for choosing **Altium Designer**