

Altium Designer

Advanced Course

Module: Multi-Channel Design

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

1.1 Purpose	3
1.2 Shortcuts	3
1.3 Preparation	3
1.4 Single Channel Design	3
1.4.1 Examining Design Files.....	3
1.4.2 Synchronizing with PCB.....	4
1.5 Multi-Channel Design	5
1.5.1 Modifying the Schematic.....	5
1.5.2 Validate the Design.....	6
1.5.3 Synchronizing the Design.....	6
1.5.4 Changing Designator Format.....	7

Multi-Channel Design

1.1 Purpose

This exercise will explore simplifying schematic entry for an audio-mixer design that has repetitive circuits.

1.2 Shortcuts



Shortcuts when working with Multi-Channel Design

F1:	Help
C-C:	Validate Design
C-O:	Project Options
CTRL+S:	Save Document

1.3 Preparation

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

1.4 Single Channel Design

To set the stage for a multi-channel design, we will begin by reviewing a simple single channel audio filter design.

1.4.1 Examining Design Files

3. In your *Projects* panel, open `Mixer.SchDoc`. It should appear as shown in Figure 1. Note the green sheet symbol rectangle in the bottom left corner with a designator of INP. It refers to the schematic `Input channel.SchDoc`.

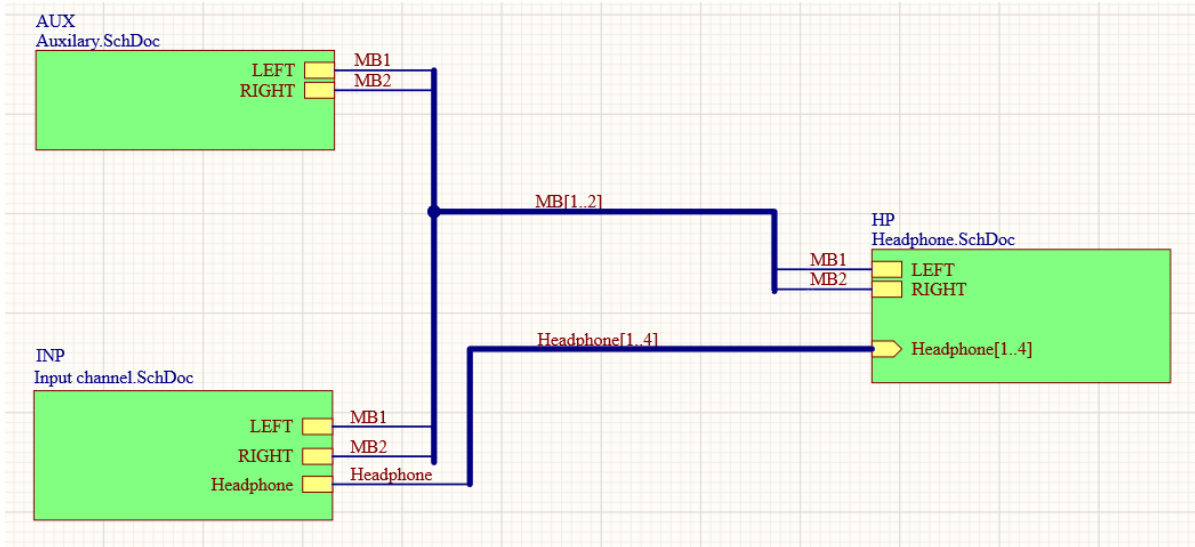


Figure 1. Single Channel design example

4. The lower-level schematic is shown in Figure 2 and contains three ports (Left, Right, and Headphone). These ports have an I/O type of **Unspecified** since these are connected to passive pins.

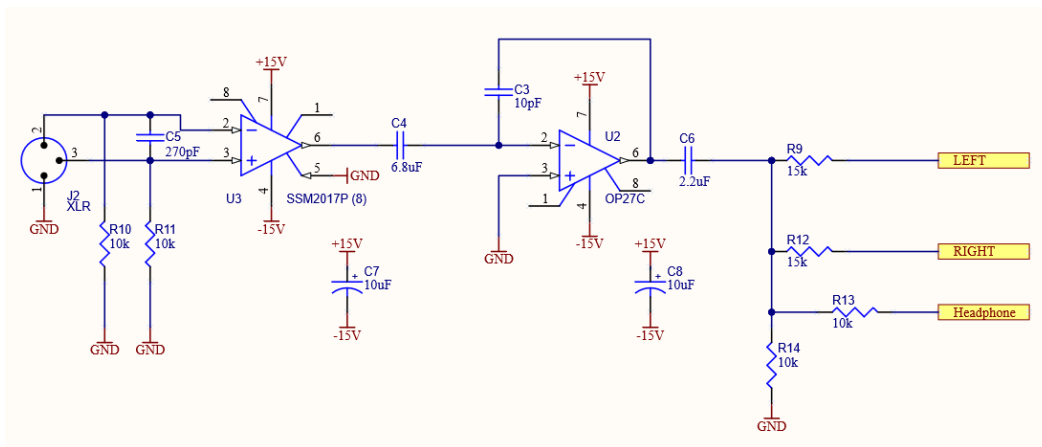


Figure 2. Lower-level schematic Input channel.SchDoc

5. Validate the project using the menus: **Project » Validate PCB Project Mixer.PcbPrj**. Examine the *Messages* panel to verify there are no errors or warnings.

1.4.2 Synchronizing with PCB

6. Open the `Mixer.PcbDoc` file to view the blank PCB document.
7. Synchronize with the schematic using the menus:
Design » Import Changes from Mixer.PrjPcb.
8. Select the **Execute Changes** button, then close the dialog.
9. Notice the three rows of components appearing to the right of the board as shown in Figure 3. The **INP** block is at the top.
10. Close the PCB **WITHOUT** saving the PCB.

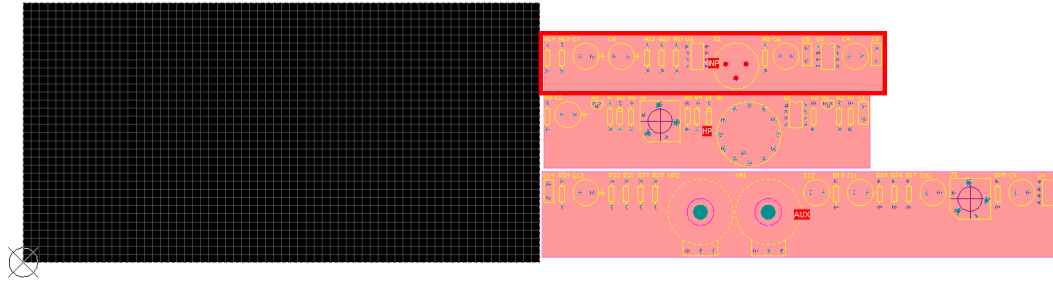


Figure 3. Single Channel design added to PCB document

1.5 Multi-Channel Design

We will now modify this design to a 4-channel mixer by changing the designator on the INP sheet symbol.



Sheet Symbol Indexing Enhancement.

Any digit or number may be used as the first or last index of a repeated Sheet Symbol, including 0. Negative numbers are not allowed. The last index must always be larger than the first index.

In our Example we will start the Indexing with 1.

1.5.1 Modifying the Schematic

11. By replacing the `INP` designator of the input channel sheet symbol with `Repeat (INP, 1, 4)`, the compiler will know to create a total of 4 copies of the input channel schematic and label them `INP1`, `INP2`, `INP3` and `INP4`. Make this change to your schematic as shown in Figure 4.
12. By replacing the Sheet-Entry `Headphone` of the input channel sheet symbol to `Repeat (Headphone)`, the compiler will know to create a connection from bus signal `Headphone1` to Channel 1, `Headphone2` to Channel 2, and so forth, as shown in Figure 4.

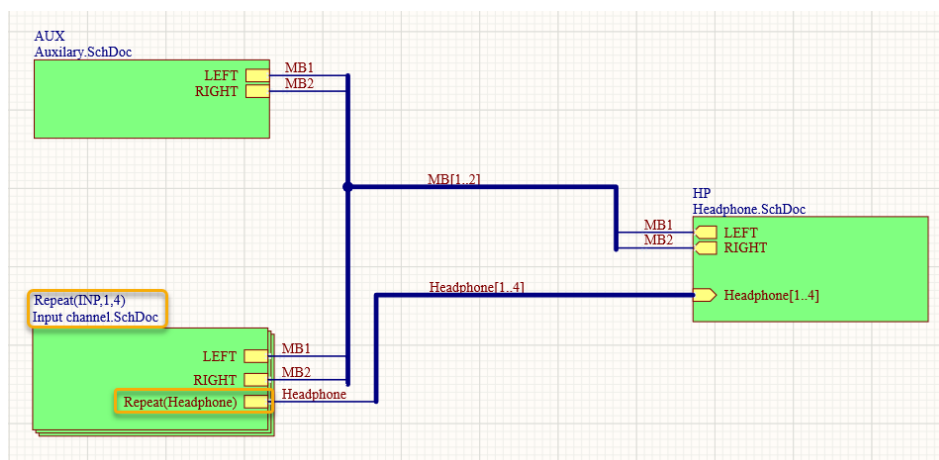


Figure 4. Modifying the input channel block to a 4-channel mixer



Note that `MB1` will connect to all four ports in each instance of the input channel schematic (`INP1`, `INP2`, `INP3` and `INP4`). However, because of the wire to bus connection for **Headphone**, each of the four lower-level ports of `Input channel` will be individually connected to the bus and then connected individually in `Headphone.SchDoc`.

1.5.2 Validate the Design

13. Validate the design and examine the *Messages* panel. You should see no errors or warnings.
14. Open the `Input_channel.SchDoc` document and notice the 4 additional tabs that are now visible in the bottom left corner, as shown in Figure 5. The **Editor** tab is unchanged and allows the user to make any desired changes to the schematic.

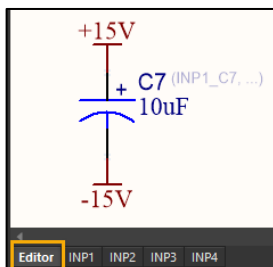


Figure 5. New tabs created in multi-channel schematic page

15. Click on the **INP2** tab and notice that the designator for the bypass capacitor is now `INP2_C7` instead of `C7`, as shown in Figure 6. This is a compiled view showing you the unique designators that will now be assigned to all the components in the multi-channel design.

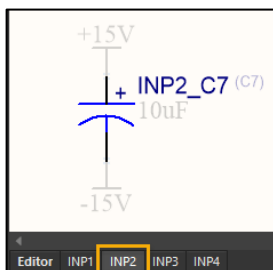


Figure 6. Compile tab for the second channel of the design

1.5.3 Synchronizing the Design

16. Open the `Mixer.PcbDoc` file and make it the current view.
17. Synchronize with the schematic using the menus: **Design » Import Changes from Mixer.PrjPcb**. Select the **Execute Changes** button then close the dialog.
18. You should now see 4 INP blocks appearing to the right of the board as shown in Figure 7.

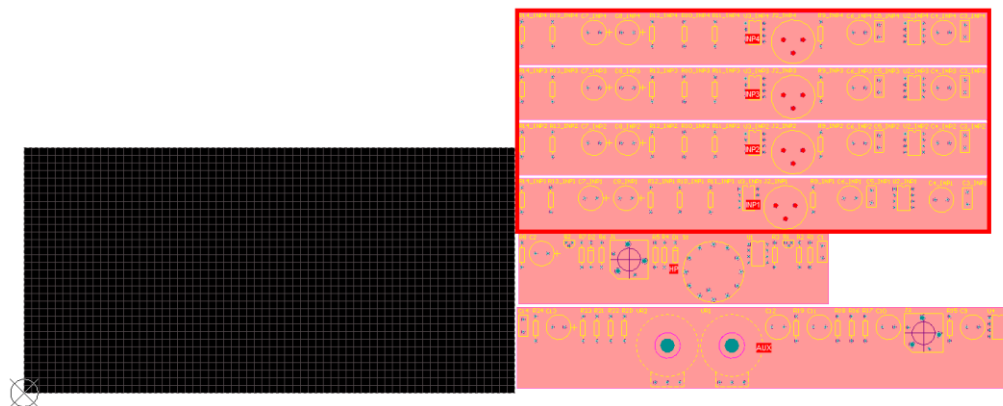


Figure 7. Multi-channel design synchronized with PCB showing 4 input channel blocks

1.5.4 Changing Designator Format

19. Change back to `Input channel1.SchDoc`, INP2 tab.
20. Select the menus: **Project » Project Options...** to open the project options.
21. Select the tab **Multi-Channel** to change the format of designators as shown in Figure 8.

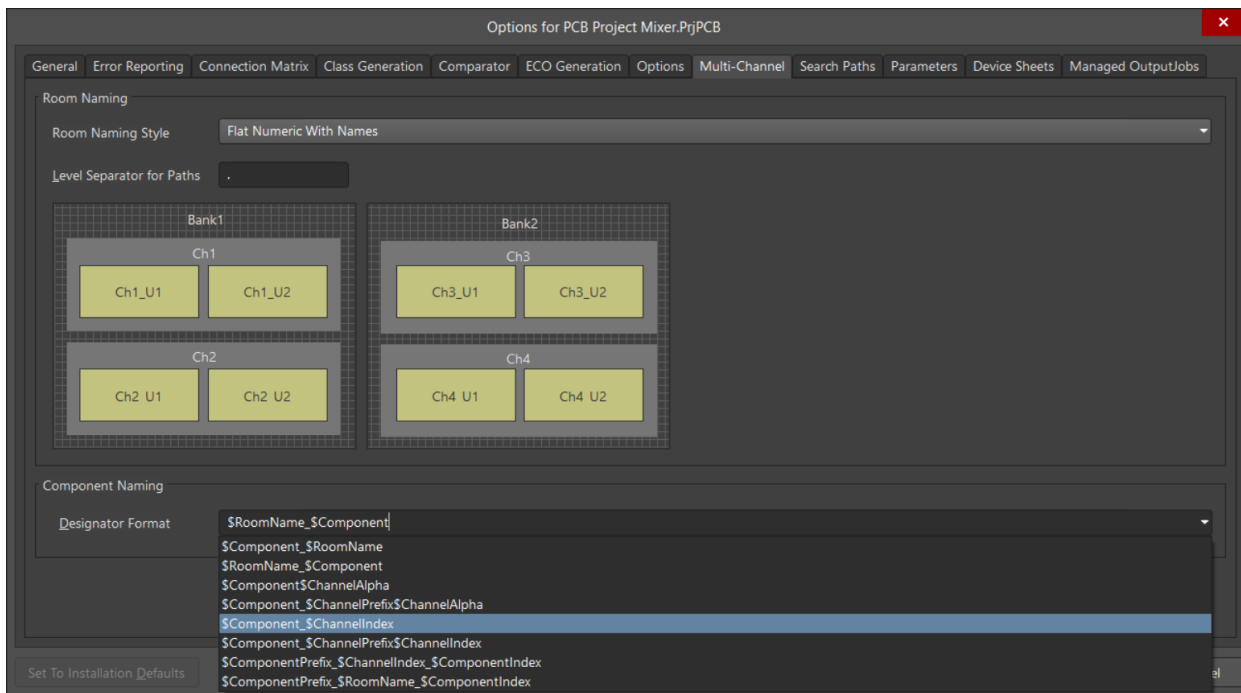


Figure 8. Multi-Channel formatting options

22. Choose `$Component_$ChannelIndex` from the *Designator Format* drop-down and notice the change in the dialog diagram showing how the new designators will appear. (If needed update the information by changing the tab, INP2 to, for example, IMP1).
23. Press the **Cancel** button to exit this dialog without saving.
24. Save all your work.
25. **Close the project and any open documents.**

Congratulations on completing module

Multi-Channel Design

from the
Altium Designer Advanced Course

Thank you for choosing Altium Designer