ALTIUM
DESIGNER

ALTIUM
365

Altium Designer

Advanced Training with Altium 365

Multi-Channel Design 1 - Schematic Entry

Altium®
TRAINING

Software, documentation and related materials:

1

Table of Contents

Multi-Channel Design 1 - Schematic Entry

1 Purpose

This module explores simplifying schematic entry for an audio-mixer design that has repetitive circuits.

2 Shortcuts

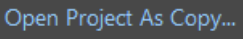Shortcuts used when working with Multi-Channel Design 1 - Schematic Entry

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

3 Preparation

1. Close all existing projects and documents.

2. Next, create a copy of the Training Project: `Multi-Channel Design 1 - Schematic Entry`.

3. Select **File » Open Project...** to open the *Open Project* dialog.

4. Enable the folder view button.

5. Navigate to the predefined Training Project `Multi-Channel Design 1 - Schematic Entry (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: `Multi-Channel Design 1 - Schematic Entry – [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 Single Channel Design

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

4.1 Examining Design Files

9.   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`.
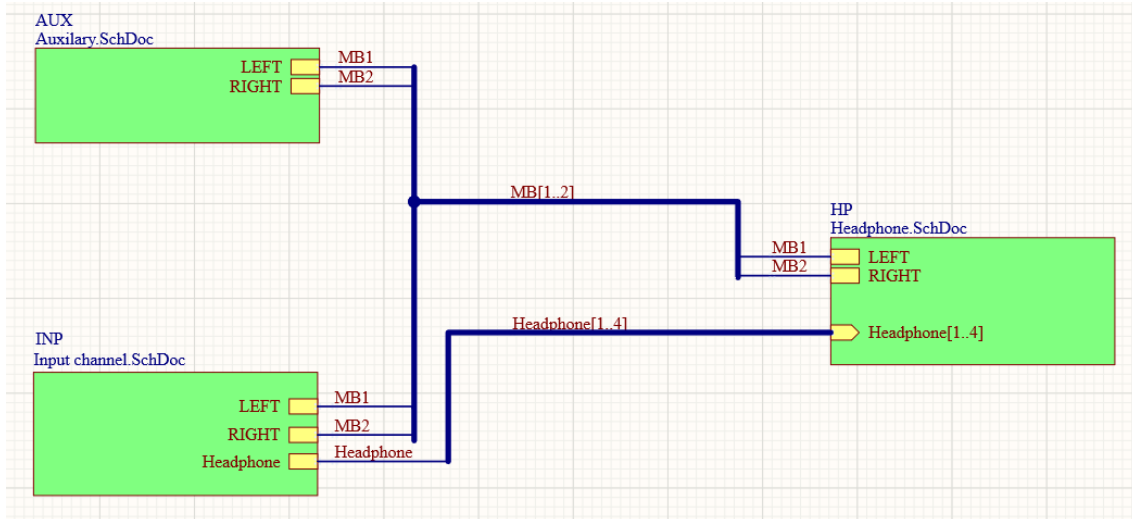
Figure 1. Single Channel design example

10. 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.
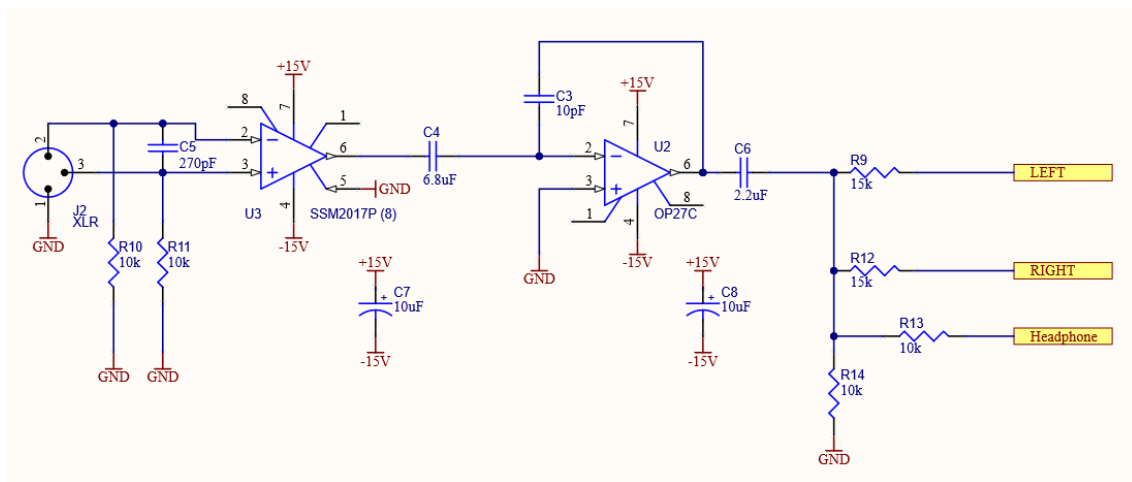
Figure 2. Lower-level schematic Input channel.SchDoc

11. Validate the project using the **Project » Validate PCB Project Multi-Channel Design.PcbPrj** menu. Examine the *Messages* panel to verify there are no errors or warnings.

4.2 Synchronizing with PCB

12. Open the `Mixer.PcbDoc` file to view the blank PCB document.

13. Synchronize with the schematic using the **Design » Import Changes from Multi-Channel Design.PrjPcb** menu.

14. Select the **Execute Changes** button, then close the dialog.

15. 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.

Figure 3. Single Channel design added to PCB document

16. Close the PCB **without saving** it.

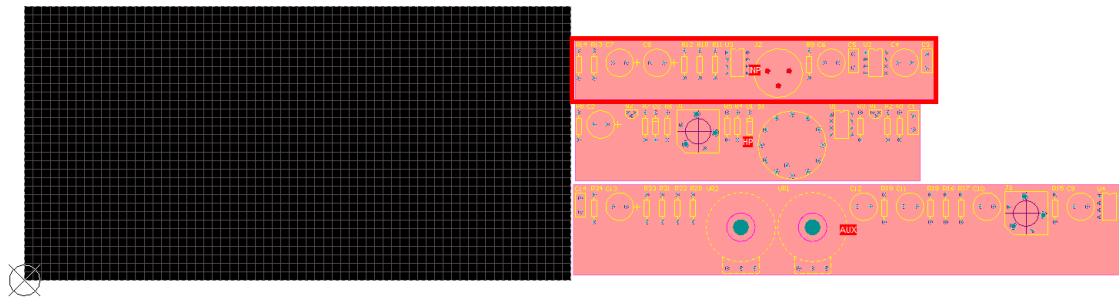# 5 Multi-Channel Design

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

> Hint: 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 aren't allowed. The last index must always be larger than the first index. In this example, you will start the Indexing with 1.

5.1 Modifying the Schematic

17. By replacing the `INP` designator of the input channel sheet symbol with `Repeat (INP,1,4)`, the compiler will 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.

18. By replacing the Sheet-Entry `Headphone` of the input channel sheet symbol to `Repeat (Headphone)`, the compiler will create a connection from bus signal Headphone1 to Channel 1, Headphone2 to Channel 2, and so forth, as shown in Figure 4.

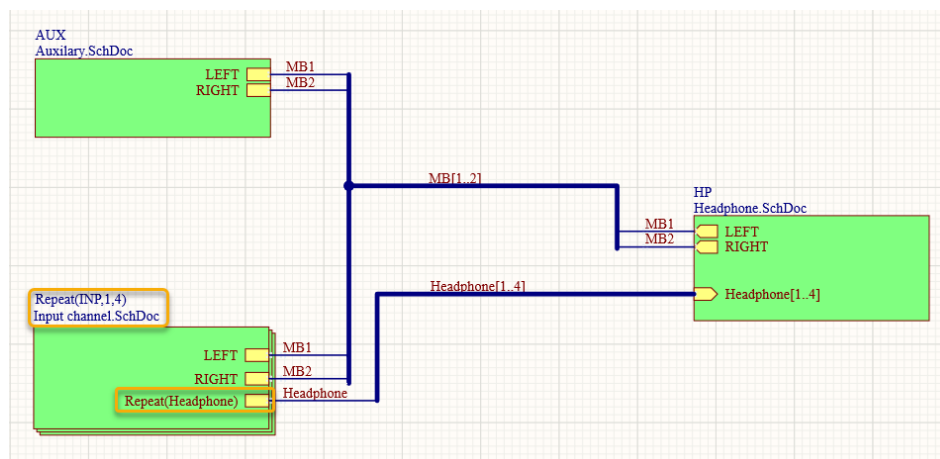*Figure 4. Modifying the input channel block to a 4-channel mixer*

> Note: `MB1` will connect to all four ports in each instance of the input channel schematic (`INP1, INP2, INP3 and INP4`). However, due to 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`.

5.2 Validating the Design

19. Validate the design and examine the *Messages* panel. There should be no errors or warnings.

20. Open the `Input channel.SchDoc` document and notice the four additional tabs that are now visible in the bottom left corner, as shown in Figure 5. The **Editor** tab is unchanged and allows you to make any changes to the schematic.
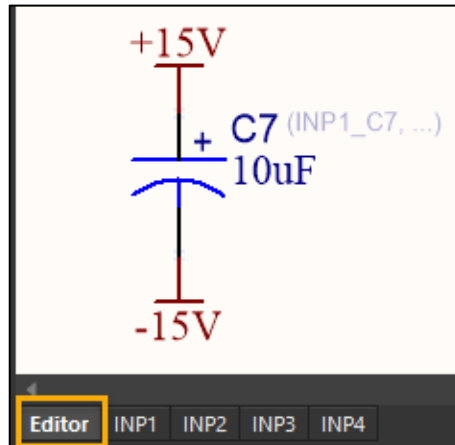

Figure 5. New tabs created in multi-channel schematic page

21. Select 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 be assigned to all the components in the multi-channel design.
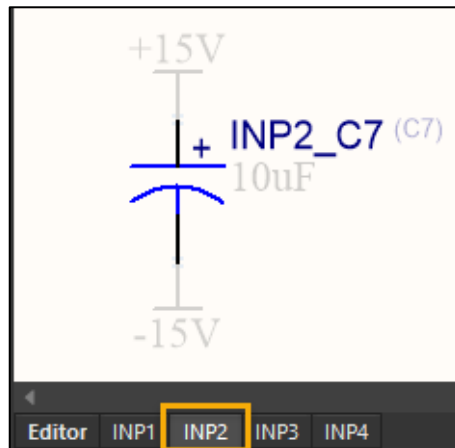

Figure 6. Compile tab for the second channel of the design

5.3 Synchronizing the Design

22. Open the `Mixer.PcbDoc` file and make it the current view.

23. Synchronize with the schematic from **Design » Import Changes from Multi-Channel Design.PrjPcb** menu. Select the **Execute Changes** button and close the dialog.

24. You can see four INP blocks appearing to the right of the board, as shown in Figure 7.
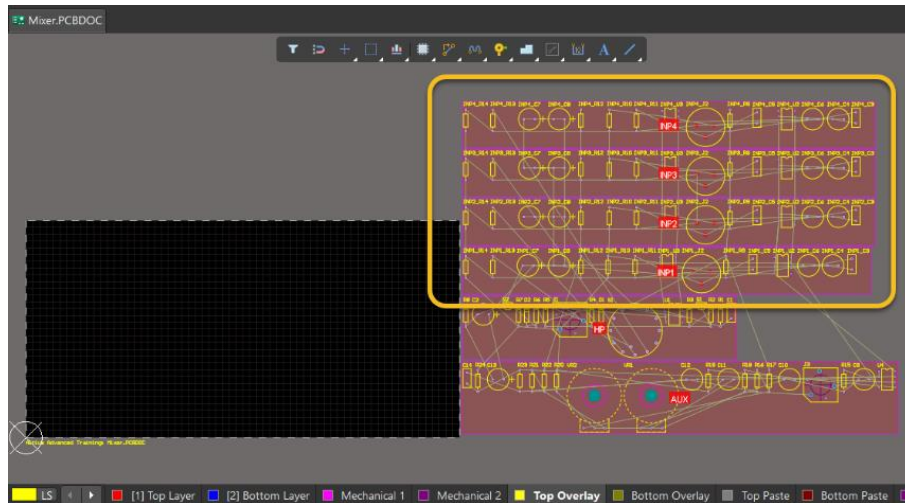

Figure 7. Multi-channel design synchronized with PCB showing four input channel blocks

5.4 Changing Designator Format

25. Change back to `Input channel.SchDoc`, INP2 tab.

26. From **Project » Project Options…** menu, open the project options.

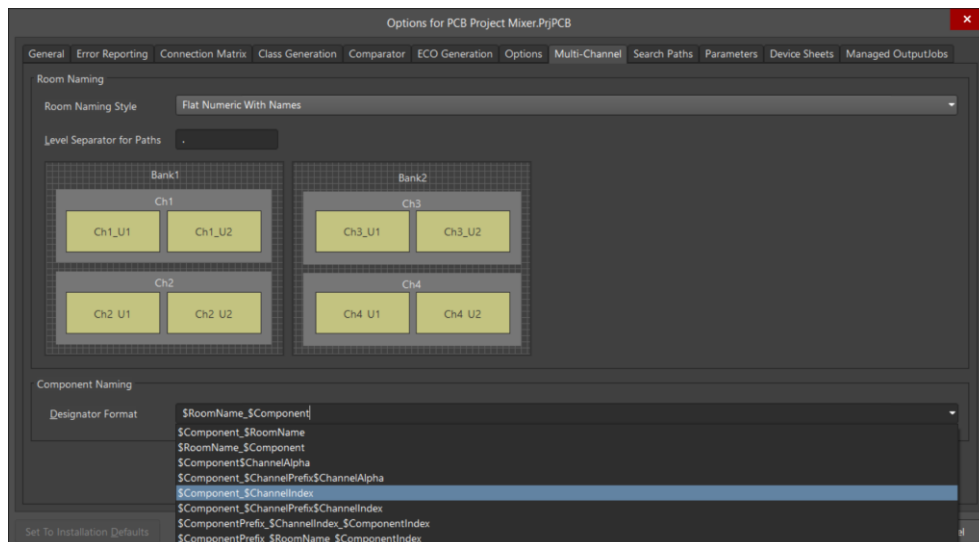27. Select the **Multi-Channel** tab to change the format of designators, as shown in Figure 8.

Figure 8. Multi-Channel formatting options

28. Choose **$Component_$ChannelIndex** from the *Designator Format* drop-down and notice the change in the dialog diagram showing how the new designators will appear.

29. Press the **Cancel** button to exit this dialog without saving.

30. Save all documents using **File » Save All**.

31. 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 `Multi-Channel Design 1 - Schematic Entry - [Add Your Name] - Finished.`

        ii) Select **OK**.

32. When ready, close the project and any open documents, **Window » Close All**.

Altium
TRAINING

Congratulations on completing the Module!

Multi-Channel Design 1 - Schematic Entry

from

Altium Designer Advanced Training
with Altium 365

Thank you for choosing **Altium Designer**

**ALTIUM®
TRAINING**