



# Altium Designer

## Advanced Training with Altium 365

### Multi-Channel Design 3 - Board Level Annotation

**Altium**  
TRAINING





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.





# Table of Contents

<b>Multi-Channel Design 3 - Board Level Annotation</b>	<b>3</b>
<b>1 Purpose</b>	<b>3</b>
<b>2 Shortcuts</b>	<b>3</b>
<b>3 Preparation</b>	<b>4</b>
<b>4 Changing Schematic Designators</b>	<b>5</b>
4.1 Schematic Changes	5
4.2 Synchronizing the PCB	8
<b>5 Designators Based on PCB Location</b>	<b>9</b>
5.1 Reordering Designators	9
5.2 Back-Annotating to the Schematics	10





# Multi-Channel Design 3 - Board Level Annotation

## 1 Purpose

Board level annotation is the process of reformatting reference designators across the entire design in a variety of formats. This includes Global Index, which provides unique designators in a simple format for multi-channel designs, for example, R15 vs. R15\_ChannelName\_Index.

## 2 Shortcuts

Shortcuts used when working with Multi-Channel Design 3 - Board Level Annotation


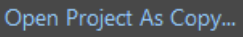

F1	Help
Ctrl+L	Board Level Annotation (SCH)
CTRL+S	Save Document





## 3 Preparation

---

1. Close all existing projects and documents.
2. Next, create a copy of the Training Project: Multi-Channel Design 3 - Board Level Annotation.
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 3 - Board Level Annotation (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: Multi-Channel Design 3 - Board Level Annotation - [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 Changing Schematic Designators

### 4.1 Schematic Changes

- Open schematic `Input_channel.SchDoc` in the project. Next, zoom to one of the components to observe the Designators, notice they are linked to the Channel Index, as shown in Figure 1. In this exercise we will change these to Global Index, so they are all numbered sequentially.

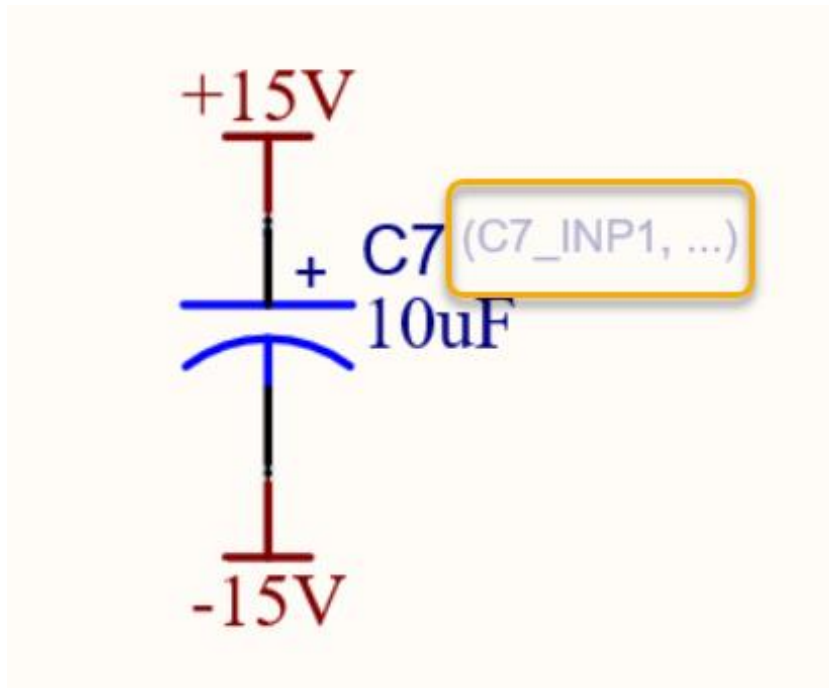


Figure 1. Physical Channel Index lined designator

- From **Tools » Annotation** menu, select **Board Level Annotate...**, as shown in Figure 2.

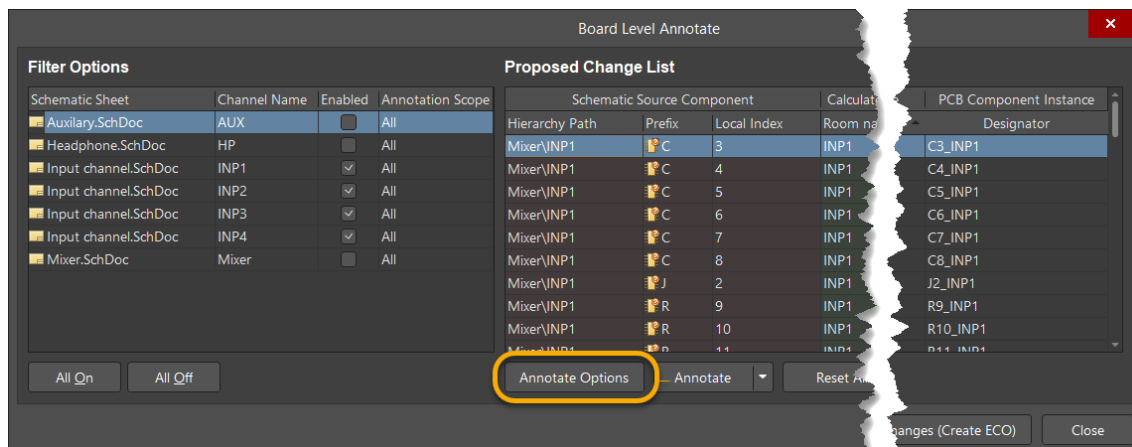


Figure 2. Board Level Annotate dialog





11. Select the **Annotate Options** button.

- In the dialog that opens, see Figure 3, change the Naming Scheme from the drop-down to `$ComponentPrefix$GlobalIndex`. This selection sets a flat naming convention, reminiscent of the standard annotation settings.
- Change the Process Location of drop-down from **Designator** to **Part**.
- Select **OK** to save your changes and return to the *Board Level Annotate* dialog.

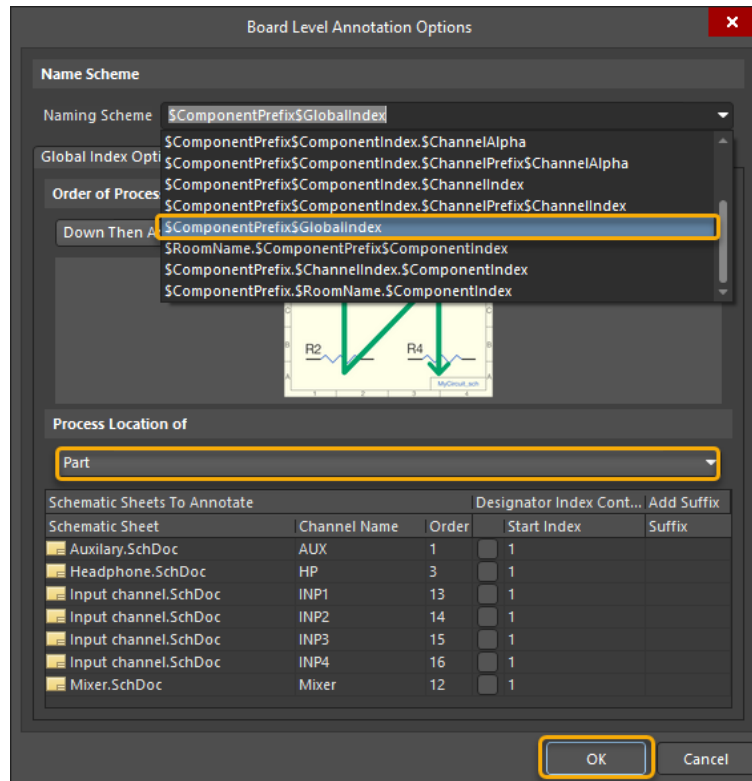


Figure 3. Board Level Annotation Options

12. Select **Reset All**. Undesignated components have a small red question mark in the prefix column, Figure 4.

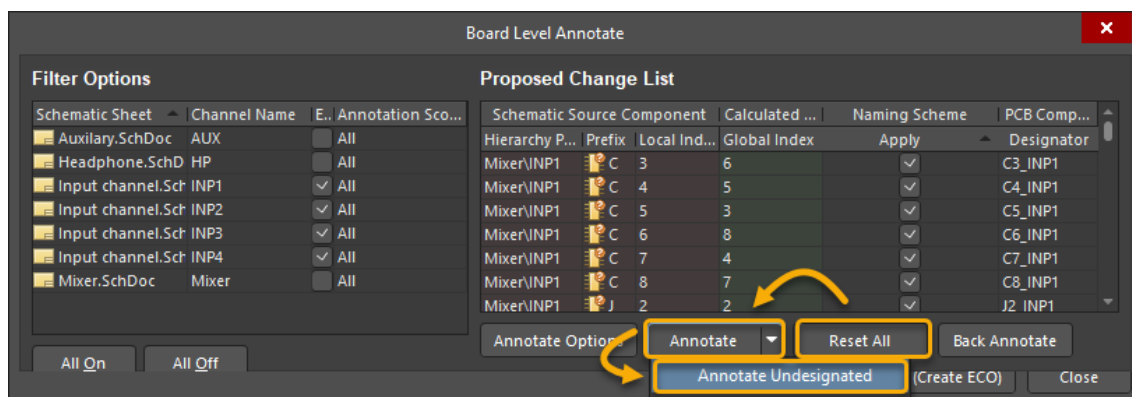


Figure 4. Undesignated Component

13. Click on the **Annotate** button and select **Annotate Undesignated**.

14. Select **OK** to dismiss the information window with the number of changes made. In this exercise, 60 changes have been made.

15. Select the **Accept Changes (Create ECO)**.





16. Select **Execute Changes** button in the *ECO* dialog.
17. Select **Close** to close the *Board Level Annotation* dialog.
18. Open the Input channel schematic and click on each of the four generated tabs at the bottom, next to the Editor tab. (Validate the project if the tabs aren't displayed). Notice that the designator values in each of the four blocks have the standard designation and they all vary.
19. Select **File » Save All**.





## 4.2 Synchronizing the PCB

20. Open the PCB document, `Mixer Placed.PcbDoc`. Once opened, view the *Properties* panel and under the *Other* section, ensure the *Designator Display* option is set to **Physical**, as shown in Figure 5.

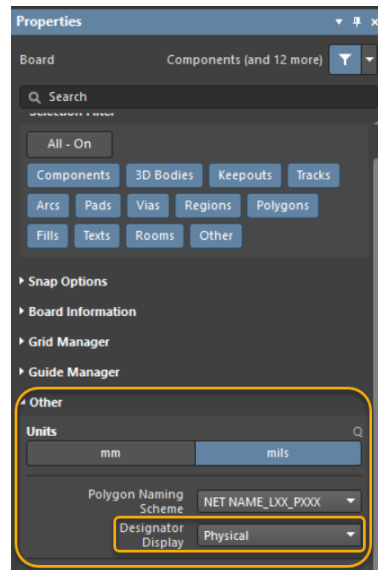


Figure 5. Select Physical Designator

21. Look at the designators in the placed rooms within the board area. They have the format, `Designator_Channel`. For example, `R10_INP4`, as shown in Figure 6.

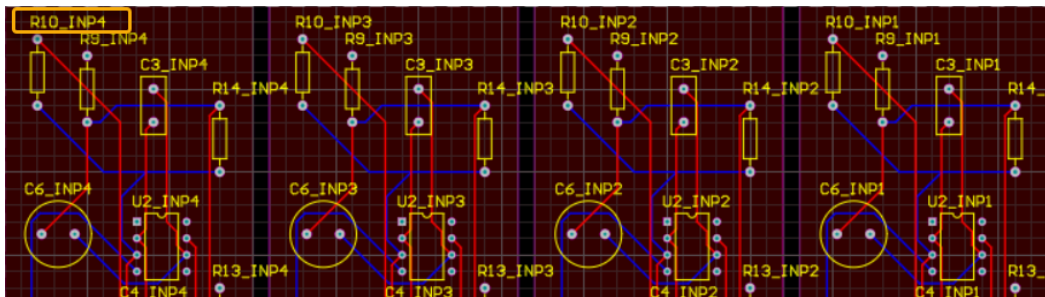


Figure 6. Channel Linked Designator

22. Import the designator changes using **Design » Import Changes From Board Level Annotation.PrjPCB**.
23. Select the **Execute Changes** button in the *ECO* dialog and close the *ECO* dialog.
24. Return to the PCB and examine the newly updated designators, as shown in Figure 7.

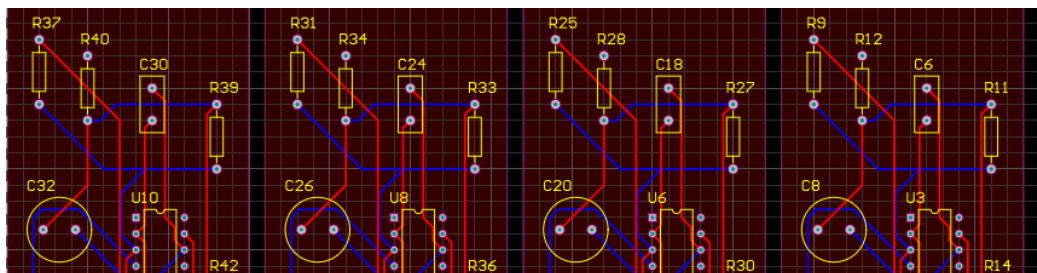


Figure 7. Rooms have unique designators after Board Level Annotate with Global Index

## 5 Designators Based on PCB Location

Optionally, designators in a multi-channel or regular design can be modified to reorder the designators based on the PCB location. For this to work properly in a design with repeated channels, a Board Level Annotate needs to be done first, similarly to the steps above.

### 5.1 Reordering Designators

25. To change designators based on their physical location in the PCB, select **Tools » Re-Annotate**.
26. Select option **2** in the *Positional Re-Annotate* dialog, *Annotate Direction* by **Ascending X Then Descending Y**, Figure 8.

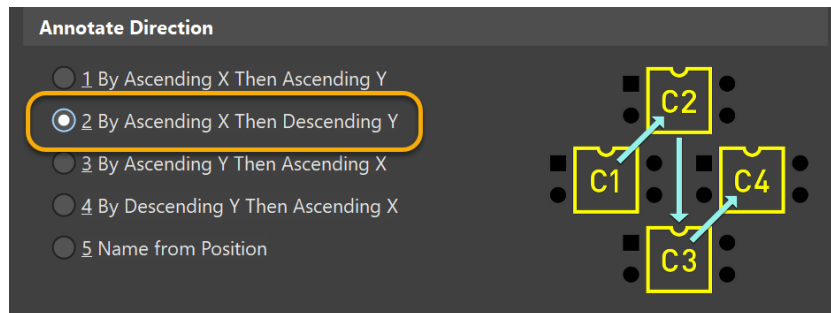


Figure 8. PCB Annotation with option 2

27. Select **OK** to accept the change. The revised board should look like Figure 9 with R1 in the top left corner.

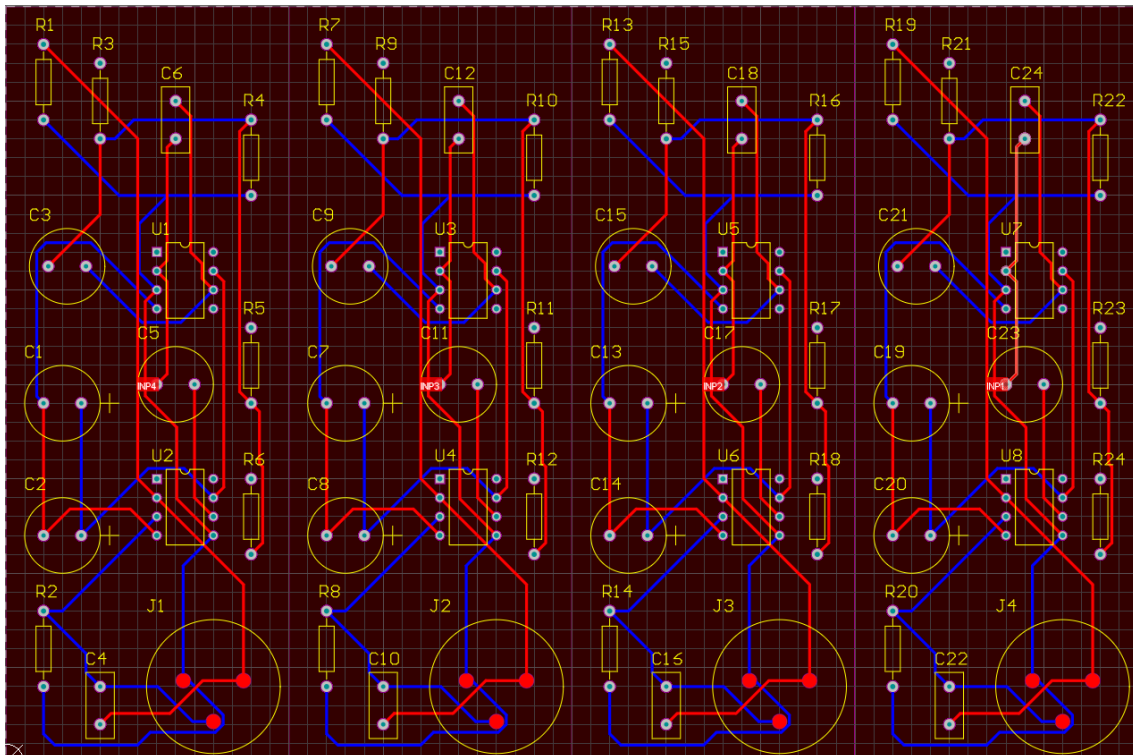


Figure 9. Designators reordered based on location on the PCB



## 5.2 Back-Annotating to the Schematics

You can back-annotate from the PCB to the Schematic only after you create an annotation file using Board Level Annotation from the schematic.

28. Run **Project » Validate Multi-Channel Design 3.PrjPCB**

29. From **Design** menu, select **Update Schematics in Multi-Channel Design 3.PrjPCB** so that the schematic has matching designators.

- In the *Comparator Results* dialog, select **Yes** to continue and create the ECO.
- Select the **Execute Changes** button to apply the changes.
- Select **Close** to close the *ECO* dialog.

30. Open the `Input Channel.SchDoc` file and examine designators in all the tabs. Note the superscript numbers. The displayed value in the compiled tab corresponds to the value on the PCB, and the superscript value is the value in the Editor tab, as seen in Figure 10.

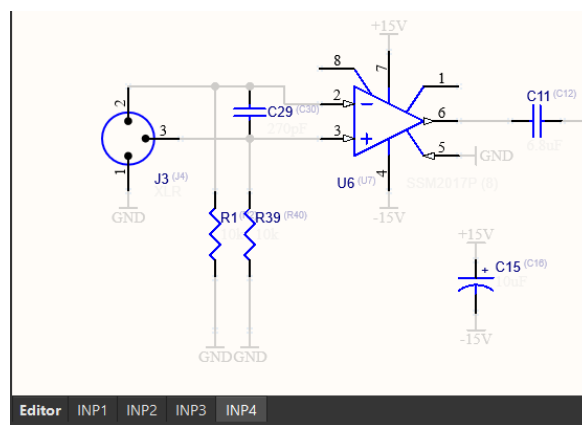


Figure 10. Back-annotate schematics synchronized with the PCB

31. Let's update the nets and class information.

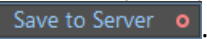
- First, validate the project (**Project » Validate PCB Project**).
- Go to **Design » Update PCB Document Mixer Placed.PcbDoc**.
- Select **Execute Changes** and close the *ECO* dialog box.

32. Now, to confirm that your design is synchronized:

- Go to **Project » Show Differences...**
- Select the `Mixer.PcbDoc`.
- Confirm with **OK**. You should see *No Differences Detected*.
- Close this dialog.





33. Save all documents using **File » Save All**.
34. 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 `Multi-Channel Design 3 - Board Level Annotation - [Add Your Name] - Finished`.
    - ii) Select **OK**.
35. When ready, close the project and any open documents, **Window » Close All**.





**Congratulations on completing the Module!**

Multi-Channel Design 3 - Board Level  
Annotation

from

**Altium Designer Advanced Training  
with Altium 365**

Thank you for choosing **Altium Designer**

