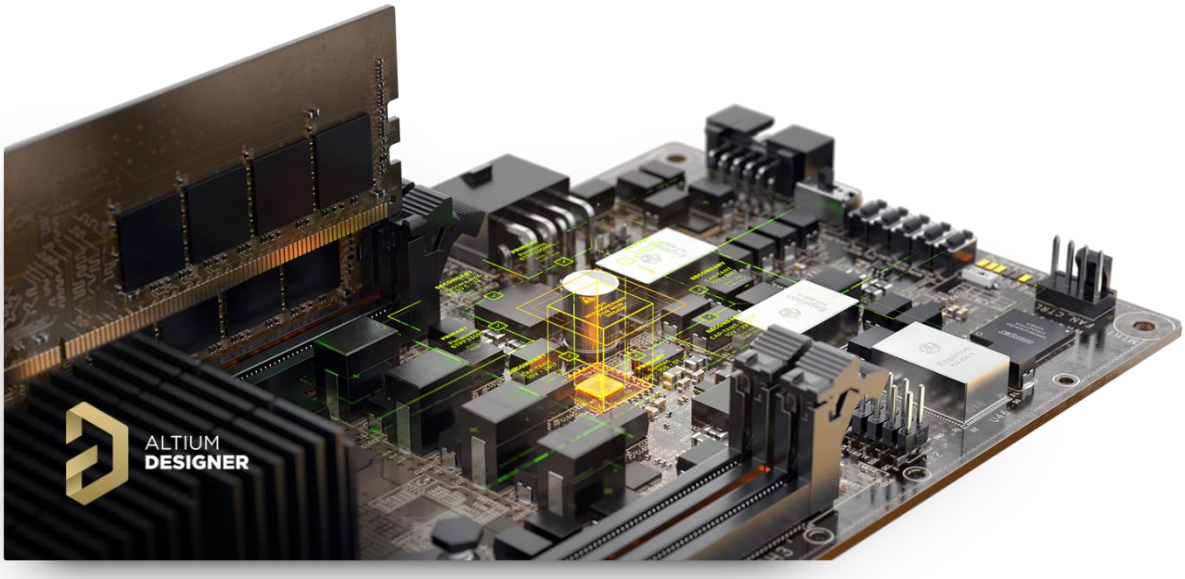

Altium Designer

Advanced Course

Module: Board Level Annotation

Software, documentation and related materials:

Table of Contents

Board Level Annotation

1.1 Purpose

Board level annotation is the process of reformatting reference designators across the entire design in a variety of possible formats. This including Global Index, which provides unique designators in a simple format for multi-channel designs. For example, R15 verses R15_ChannelName_Index.

1.2 Shortcuts

Shortcuts when working with Board Level Annotation

F1:	**Help**
CtrL+L:	**Board Level Annotation (SCH)**
CTRL+S:	**Save Document**

1.3 Preparation

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

1.4 Changing Schematic Designators

1.4.1 Schematic Changes

3. 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.

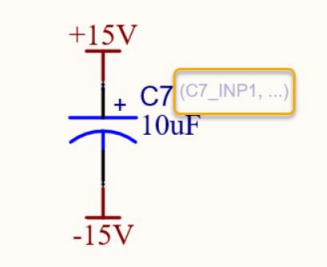

Figure 1 Physical Channel Index lined designator

4. Open the *Board Level Annotate* dialog using the menus: **Tools » Annotation » Board Level Annotate…** as shown in Figure 2

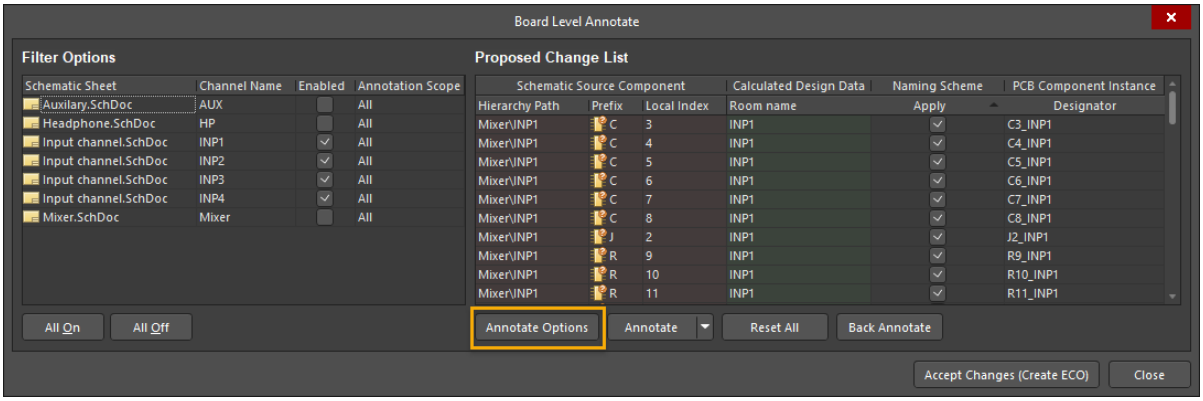

Figure 2. Board Level Annotate dialog

5. Click on the **Annotate Options** button.
   a) In the dialog that opens change the *Naming Scheme* drop-down to *$ComponentPrefix$GlobalIndex*. This selection sets a flat naming convention, reminiscent of the standard annotation settings.
   b) Change the Process Location *of* drop-down from Designator to **Part.**
   c) Click **OK** to save your changes and return to the *Board Level Annotate* dialog.
6. Click **Reset All**. Undesignated components have a small red question mark in the prefix column, Figure 3.

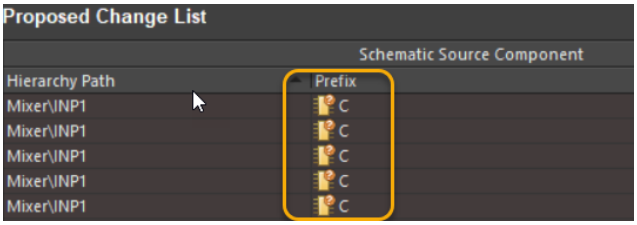

Figure 3. Undesignated Component

7. Click the **Annotate** button and choose **Annotate Undesignated**. Click **OK** to dismiss the information window saying 60 changes made.
8. Click the **Accept Changes (Create ECO).**
9. Click **Execute Changes** button in the *ECO* dialog.
10. Click **Close** to close the *Board Level Annotation* dialog.
11. Open the Input channel schematic and click on each of the 4 generated tabs at the bottom next to the Editor tab. (Validate the project if the tabs are not present). Notice that the designator values in each of the 4 blocks have the standard designation and are all different.
12. Select **File » Save All**. Save the Files Local but do Not sync with the Workspace

1.4.2 Synchronize the PCB

13. Open the PCB document, `Mixer.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 4.

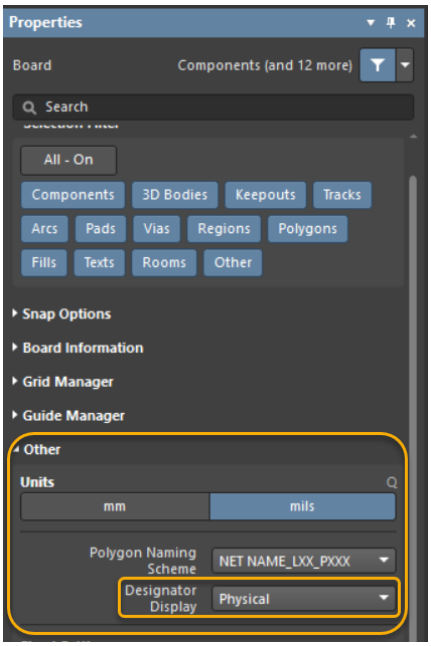

Figure 4. Select Physical Designator

14. 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 5.

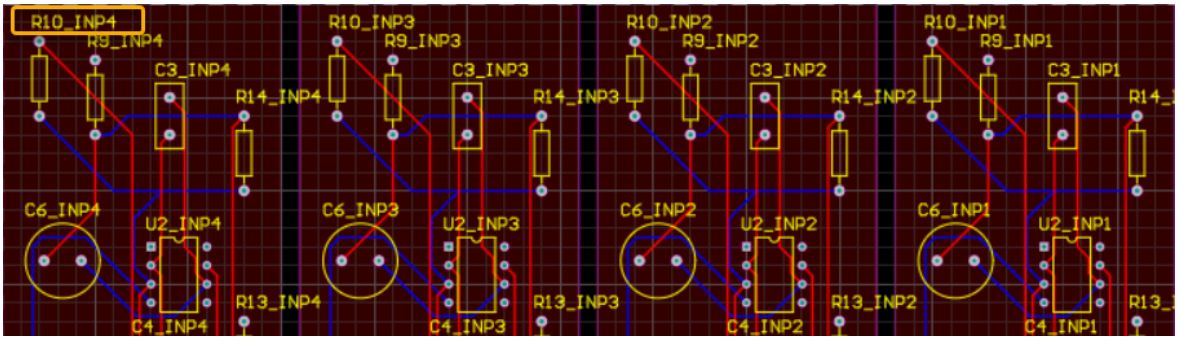

Figure 5. Channel Linked Designator

15. Import the designator changes using **Design » Import Changes From Mixer.PrjPcb.**
16. Click the **Execute Changes** button in the *ECO* dialog.
17. Click **Close** to close the *ECO* dialog.
18. Return to the PCB and examine the newly updated designators, as shown in Figure 6

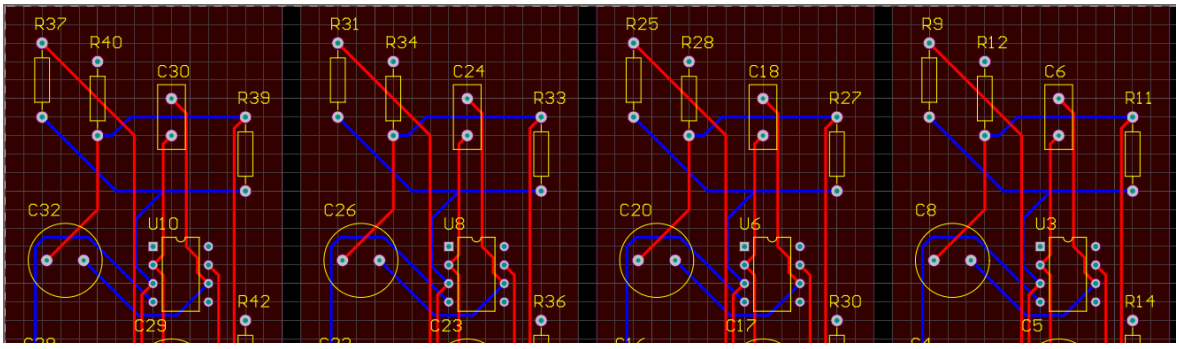

Figure 6. Rooms have unique designators after Board Level Annotate with Global Index

1.5 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 is required to be performed first, similarly to the steps performed above.

1.5.1 Reorder Designators

19. To change designators based on their physical location in the PCB, select **Tools » Re-Annotate**.
20. Choose option **4** in the *Positional Re-Annotate* dialog, *Annotate Direction* by **Descending Y then Ascending X**.
21. Click **OK** to accept the change. The revised board should look like Figure 7 with *R1* in the top left corner.

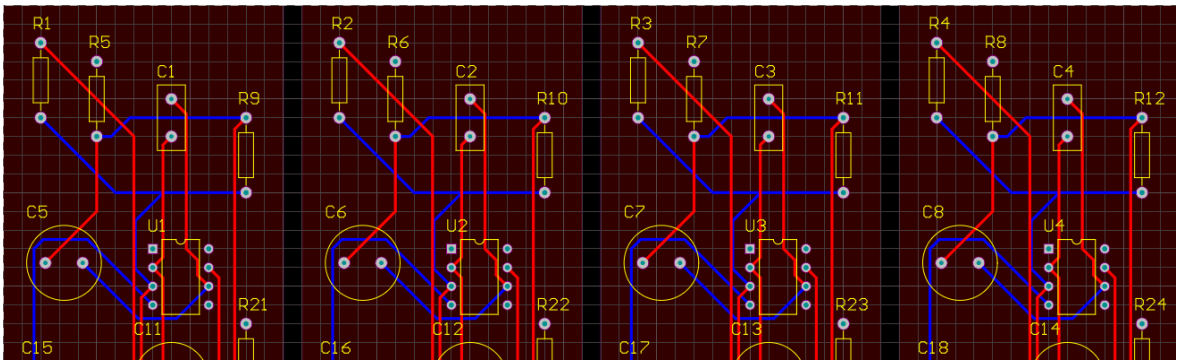

Figure 7. Designators reordered based on location on the PCB

1.5.2 Back-Annotating to the Schematics

Back annotating from the PCB to the Schematic can only be successfully achieved after an annotation file is created by performing a Board Level Annotation from within the schematic.

22. Run **Project » Validate PCB Project Mixer.PrjPcb**
23. Use the menus: **Design » Update Schematics in Mixer.PrjPcb** so that the schematic has matching designators.
    a) In the *Comparator Results* dialog, click **Yes** to continue and create the ECO
    b) Click the **Execute Changes** button to apply the changes.
    c) Click **Close** to close the *ECO* dialog.
24. 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 can be seen in Figure 8.

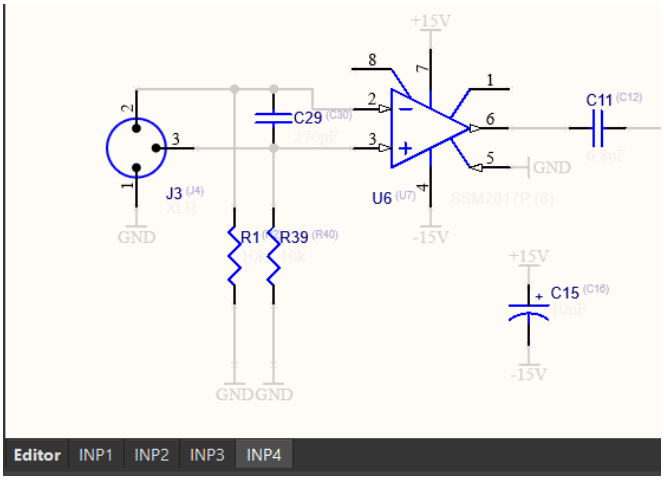

Figure 8. Back-annotate schematics synchronized with the PCB

25. We must now update the nets and push this through to the PCB. First, Validate the project (**Project » Validate PCB Project**). Go to **Design » Update PCB Document Mixer.PcbDoc**. Click **Execute Changes** and close the *ECO* dialog box.
26. Now to confirm that our design is synchronized, go to **Project » Show Differences…** Select the `Mixer.PcbDoc` and click **OK**. We should now see No Differences Detected. Close this dialog.
27. Save all your work.
28. **Close the project and any open documents.**

Congratulations on completing module

Board Level Annotation

from the

Altium Designer Advanced Course

Thank you for choosing Altium Designer