



Altium Designer

Essentials Training with Altium 365

Module 12: Schematic Updating

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

Module 12: Schematic Updating	3
1 Purpose	3
2 Shortcuts	3
3 Preparation	4
4 Align	5
4.1 Align Right	5
5 Net Color Highlighting	6
6 Schematic Annotation	8
7 Adding Directives for PCB	10
7.1 Assigning PCB Directives – Rule and Class Information	10
8 Number Schematic Sheets	13





Module 12: Schematic Updating

1 Purpose

In this exercise, you will add text and graphics to the schematics to provide documentation and organization. You will also number schematic sheets.

2 Shortcuts


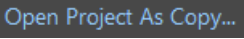

Shortcuts used when working with Module 12: Schematic Updating

E » G » G	Align to Grid
Shift+Ctrl+D	Align to Grid
E » G » R	Align Right
Shift+Ctrl+R	Align Right
T » A » N	Force Annotate
T » A » U	Annotate Quietly
T » A » A	Annotate Schematic
Shift+C	Clear the highlight and mask
F5	Net Color Highlighting on / off
C » O	Options for Project (Project Options)





3 Preparation

1. Close all existing projects and documents.
2. Next, create a Copy / Clone of the Training Project Module 12 Schematic Updating.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Enable the folder view button .
5. Navigate to the predefined Training Project Module 12 Schematic Updating (Top\Projects\Altium Designer Essentials Training Course\...)
6. Select **Open Project as Copy...** .
7. At the new dialog *Create Project Copy*
 - a) Add your name to the project name: Module 12 Schematic Updating - [Your Name] .
 - b) Add a description: Altium Essential Training - Module 12 - [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 opens the project in the *Projects* panel; this can take up to 1 minute.

Hint: For details how to Copy / Clone the predefined training project see Module 9 Making the Connection, Step 3 Preparation.



4 Align

4.1 Align Right

Alignment tools are used to ensure that objects are aligned to one another, or to the grid to achieve a clean document.

9. Open the `Processor_Interface.SchDoc` from the *Projects* panel. The **Messages** panel may display errors. Please ignore it for now.
10. Select the ports on the left of the schematic using a left to right selection rectangle.
11. Go to **Edit » Align » Align Right** or use the **Active Bar** to align the ports, Figure 1.

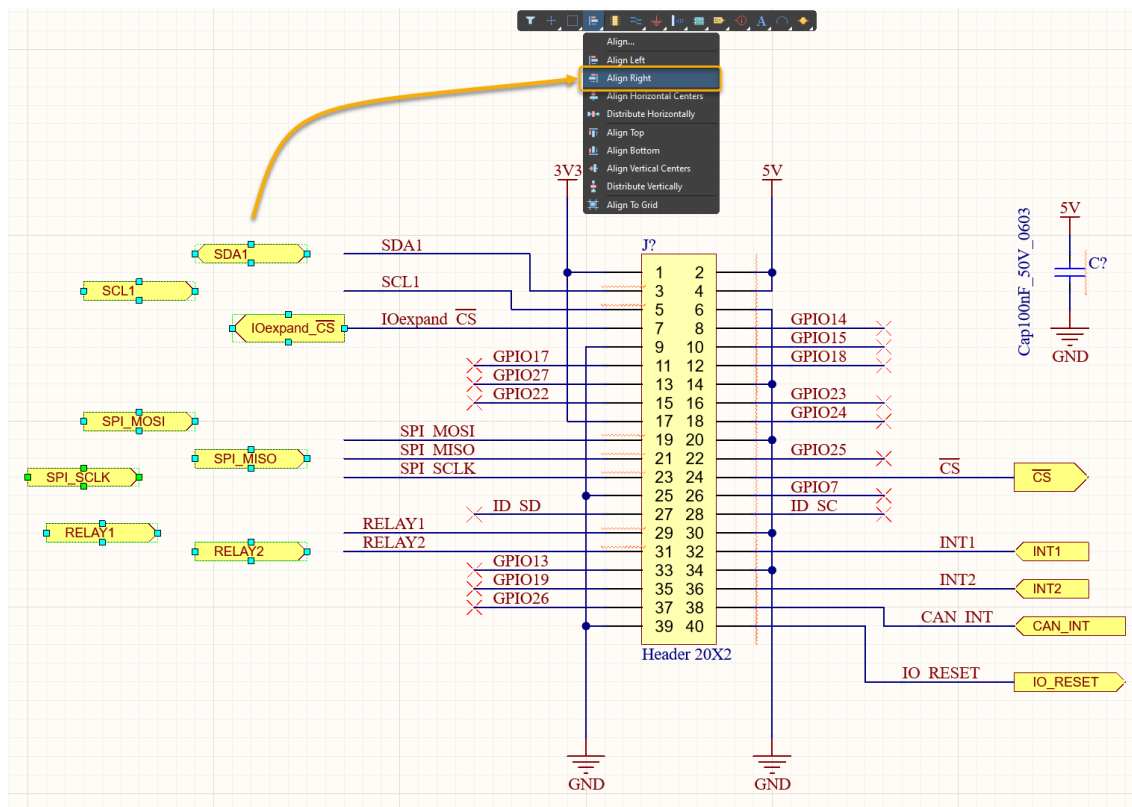


Figure 1. Align for unconnected Ports

Hint: Use the **Edit » Align » Align to Grid** to align objects within inherited/imported designs that were not captured on grid. This is to maintain proper connectivity between electrical objects.

5 Net Color Highlighting

12. Open the `Processor_Interface.SchDoc` from the *Projects* panel.
13. From the **View** menu, select **Set Net Colors** and select **Red**, as shown in Figure 2 below.

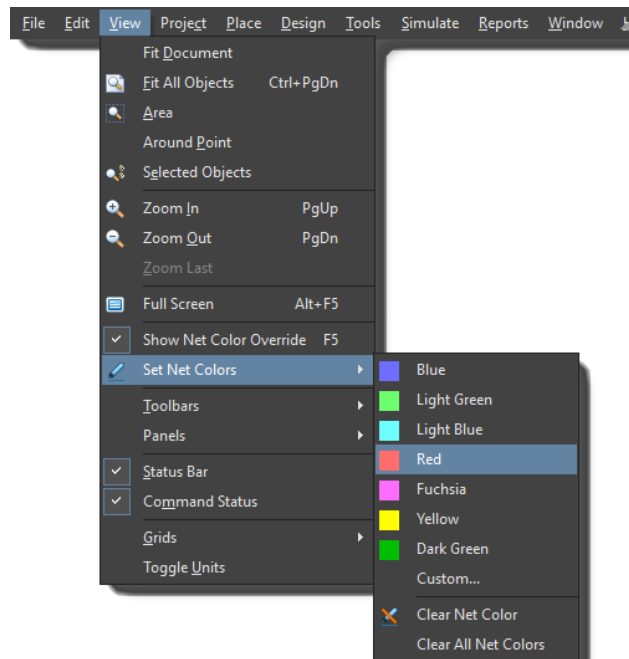


Figure 2. Set Net Colors menu

14. Click on a wire connected to a 5V Power Port to highlight the entire net in red, as shown in Figure 3.
15. Then, **right-click** to exit the command.

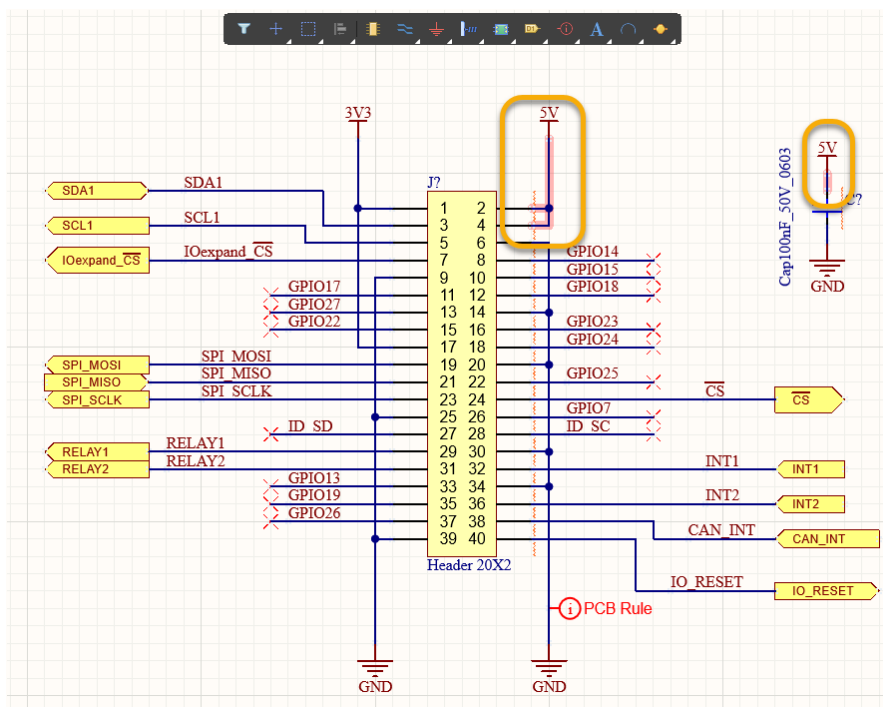


Figure 3. `Processor_Interface.SchDoc` with 5V net highlighted in red



Note: Notice that power ports that are directly connected to a component will not have a red highlight because there are no wires to highlight on.

16. All the wires connected to 5V in the other schematics such as in `Digital_IO.SchDoc` and `Power Supply.SchDoc` are also highlighted in red.
17. Repeat Step 12 to 14 for the Net 3V3 Volt (orange) and GND (blue).
18. Repeat Step 12 to 14 for the Net 12V Volt (yellow). Use the *Navigator* panel or the *Project* panel to find the 12V net.
19. To activate / deactivate the Net Color Highlighting press **F5**.



6 Schematic Annotation

20. Head to **Tools » Annotation » Annotate Schematics...** to open the *Annotate* dialog. The window shown in Figure 4 below will appear.

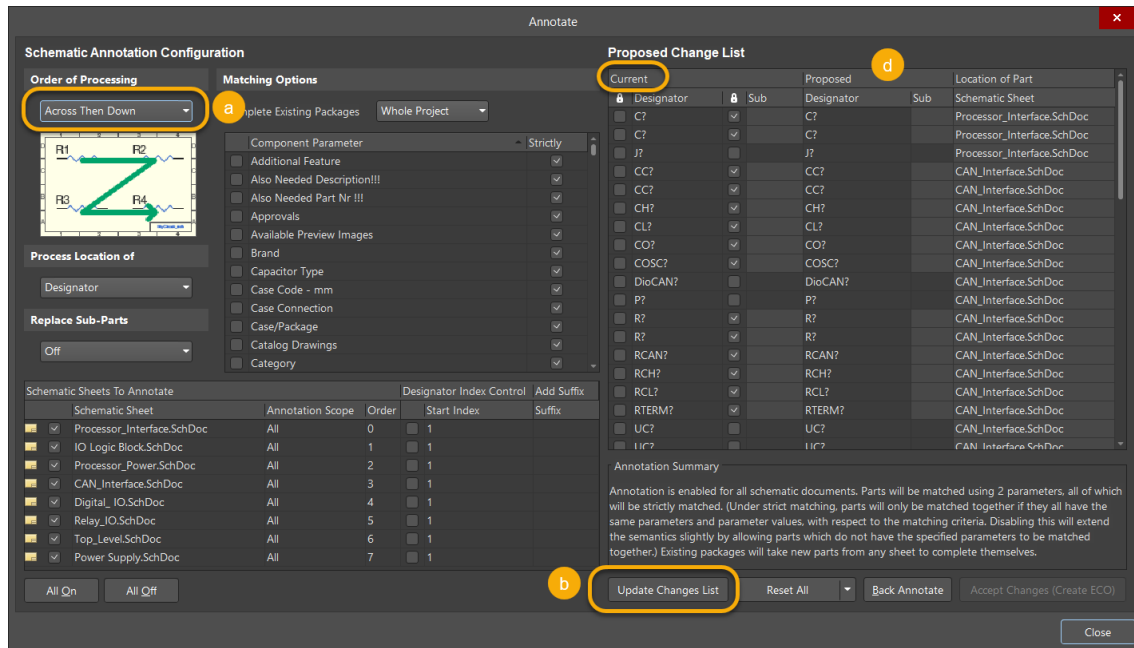


Figure 4. Annotate Dialog

- Set the *Order of Processing* to **Across Then Down** using the pull-down menu.
- Click the **Update Changes List** button.
- Click **OK** on the pop-up dialog that appears telling us which changes have been made.
- A preview of the proposed designator values will be displayed similar to what is shown in Figure 5.

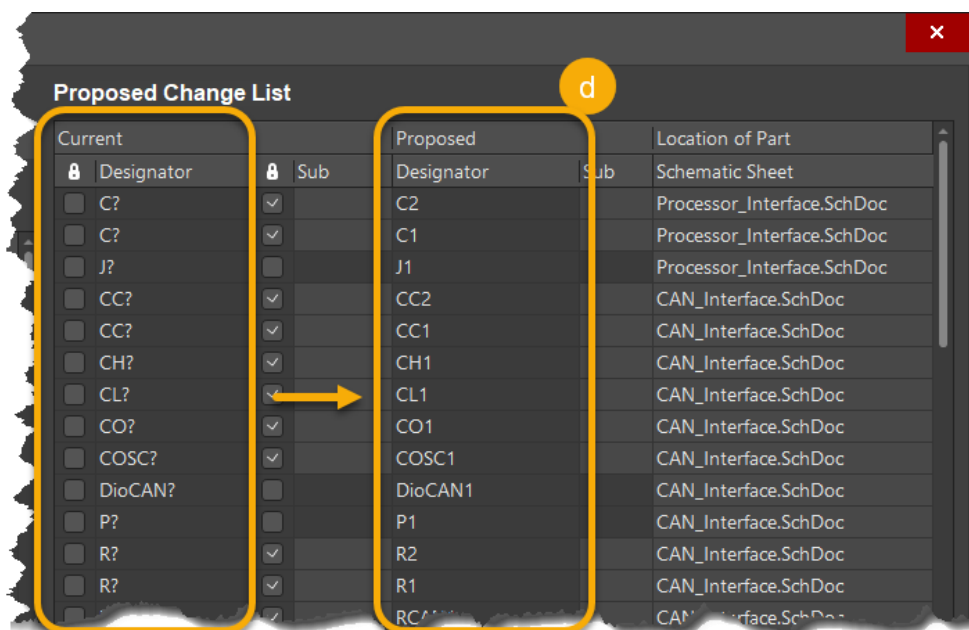


Figure 5. Proposed Designator changes based on the Annotation settings

21. These changes are only proposed so far. To commit this change to the design:

- Press the **Accept Changes (Create ECO)** button.
- Execute Changes** within the resulting *Engineering Change Order as shown in* Figure 6. All undesignated parts should now have a unique designator.
- Tick / Untick the Checkbox for **Only Show Errors** to reduce the list to reported errors.

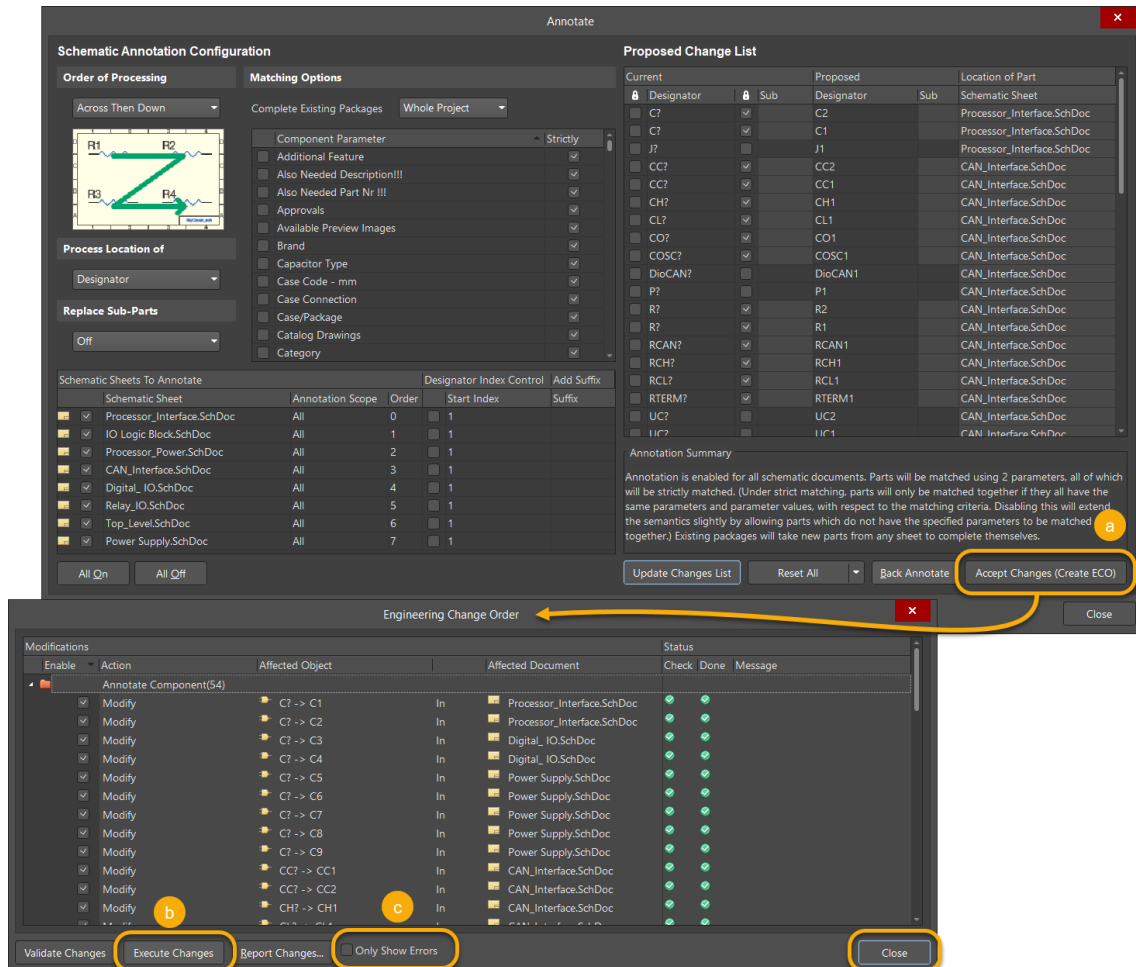


Figure 6. Engineering Change Order for the Annotation of components

22. Press the **Close** button to close the completed ECO, and then again press the **Close** button to close the *Annotate* dialog.

23. Review the schematics to confirm that they now have unique designators.

Hint: If there are any "old" designators R1^(R?), they are referenced as a superscript in light gray next to the newly named designators. Validating the project will remove the superscript of the "old" designators.



7 Adding Directives for PCB

Note: It is important to note Directives in the schematic are only applicable if working with design rules using the classic *Design Constraints Editor* in the PCB.

7.1 Assigning PCB Directives – Rule and Class Information

24. With `Processor_Interface.SchDoc` as active document:

25. Use the command **Place » Directives » Parameter Set**.

- a) With the directive on your cursor, press the **Tab** key to edit its properties.
- b) In the *Properties* pane, ensure the value for *Label* is `PCB Rule`.
- c) Under the *Parameters* pane, click **Add... » Rule**, as shown in Figure 7.

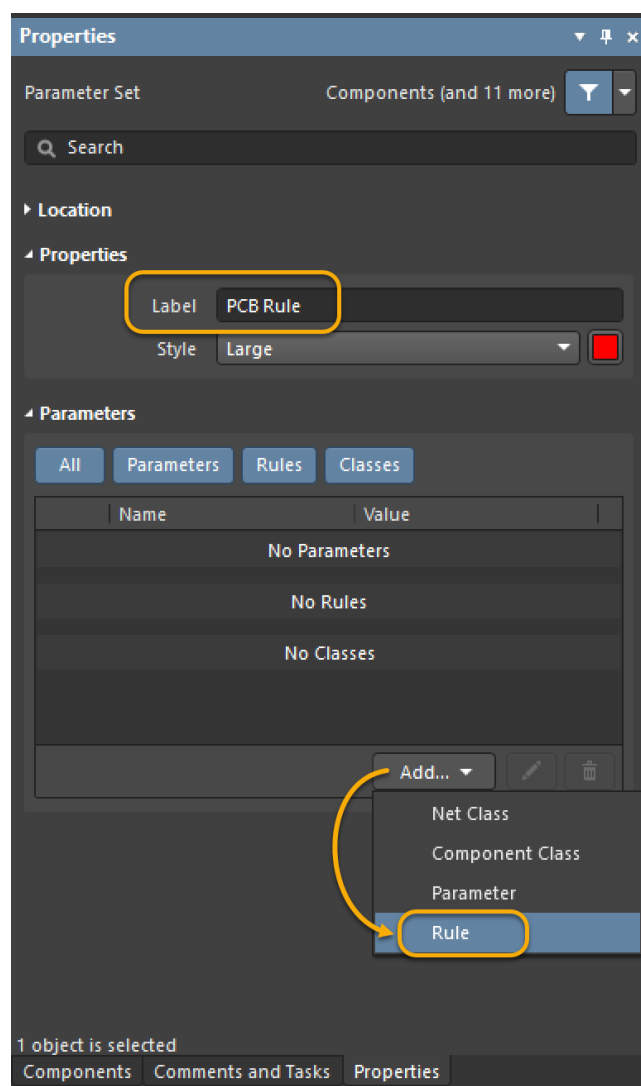


Figure 7. Adding PCB rule directive from the schematic





d) Select the *Width Constraint* rule from within the *Routing* section shown in Figure 8 below.

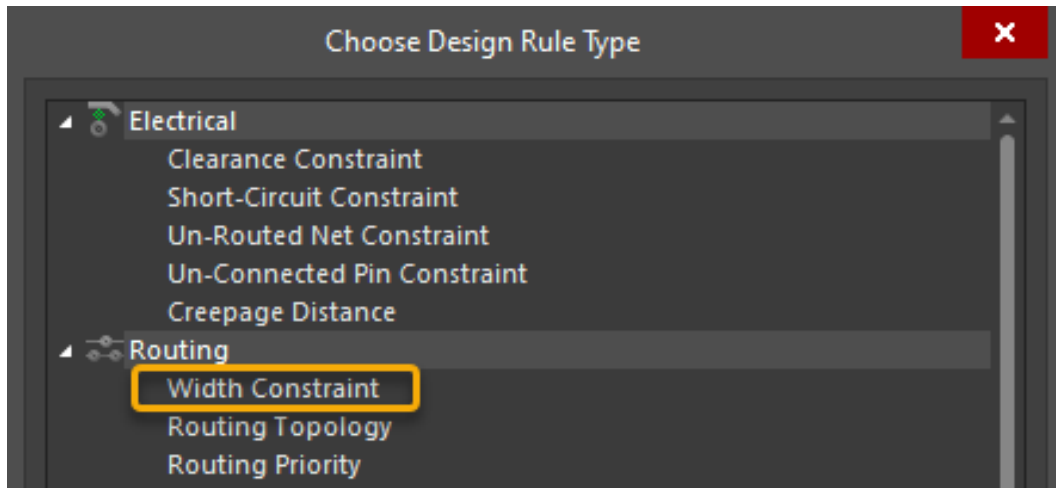


Figure 8. Width Constraint Rule

e) Press the **OK** button to continue.

f) In the upper section of the rule, set the following values as shown in Figure 9 below:

- i) *Min Width* to 30mil.
- ii) *Preferred Width* to 50mil.
- iii) *Max Width* to 100mil.

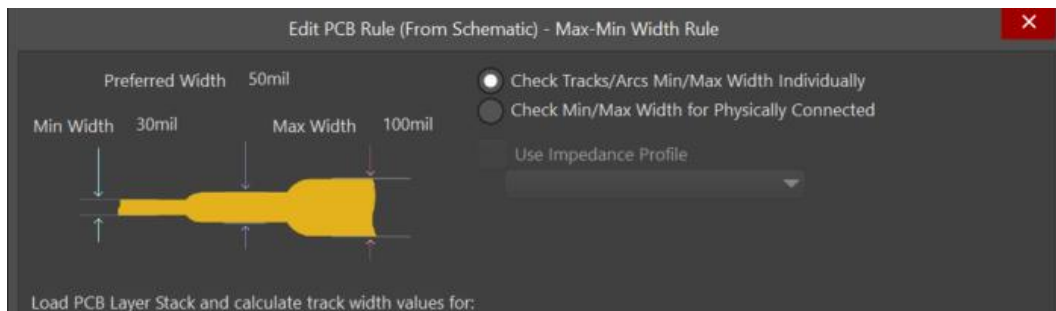


Figure 9. Width Rule settings

iv) Press the **OK** button to close the PCB Rules Dialog.





- g) Still in the *Properties* panel, Figure 10:
 - i) Click **Add... » Net Class**.
 - ii) A new line with the name **Net Class Name** is generated.
 - iii) Add the name **Power** in the right cell of the new line.

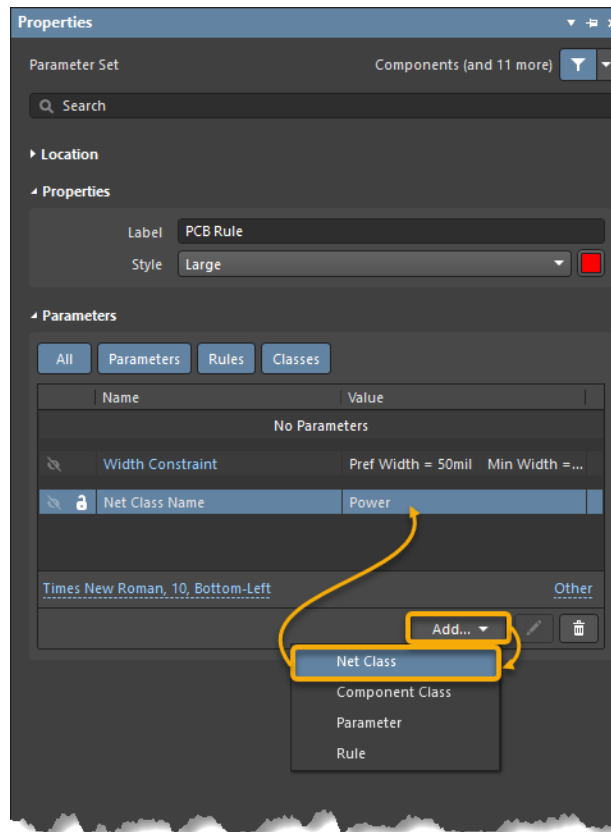



Figure 10. Adding Net Class from the schematic

- h) Click the  Pause Symbol to continue the placement of the PCB Rule Directive.
- i) Place the PCB Rule Directive on both the 3V3 and 5V wires extending from the top of the J1 component as shown in Figure 11.
- j) Place the PCB Rule Directive on one of the GND wires at the bottom of the J1.

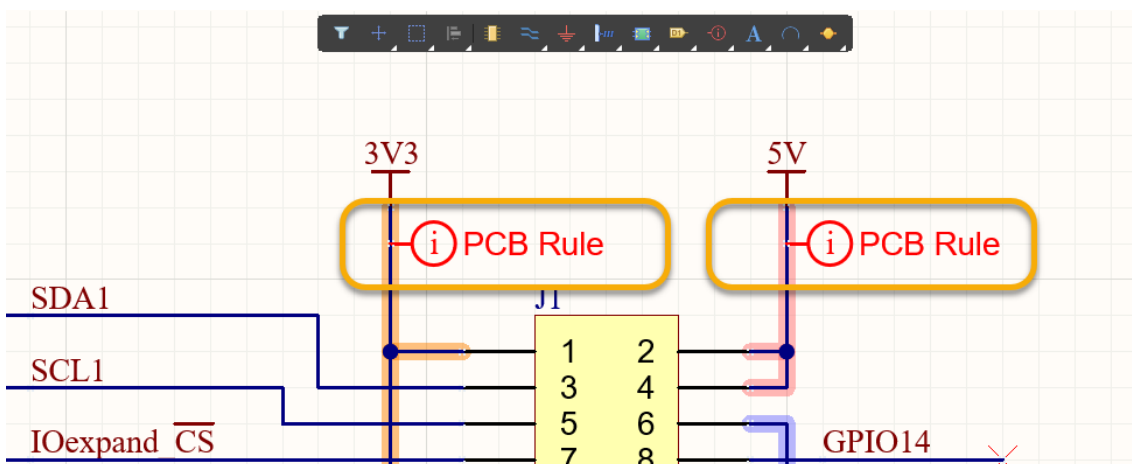


Figure 11. Placing the PCB Rule directives in the schematic

- k) Right-click to exit the command.





8 Number Schematic Sheets

In this exercise, we will demonstrate how to add sheet numbers to the schematic sheets in our project using parameters; a parameter is a special string, that has a Name and a Value. In this case the special strings have been pre-placed in the schematic templates. We can also see the parameters and their values in the *Properties* panel. In this case the value has been automatically applied to the SheetNumber and SheetTotal parameters. See Figure 12.

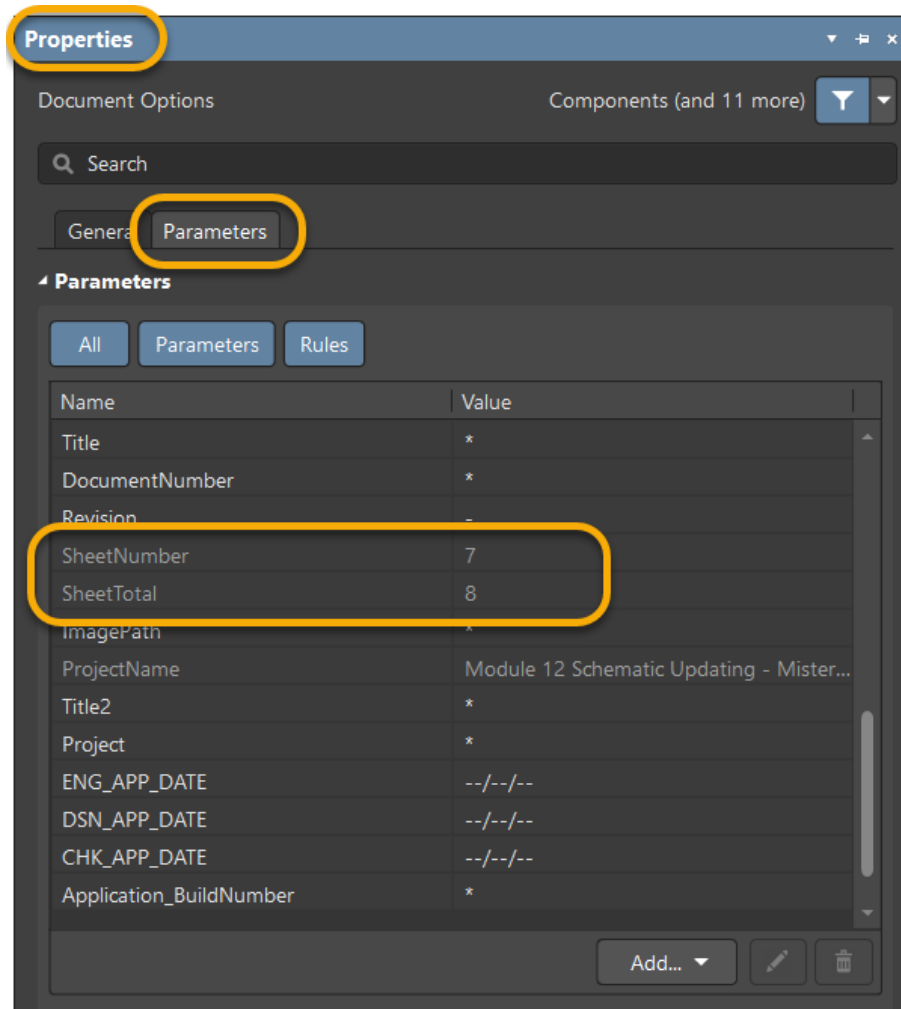


Figure 12. Sheet Number Parameters



26. From the **Project** menu, select **Project Options...**

The *Options for PCB Project* dialog will appear, as shown at Figure 13 below.

- Select the **Options**.
- Notice that the **Automatic Sheet Number** is active to assign numbers to the sheets.

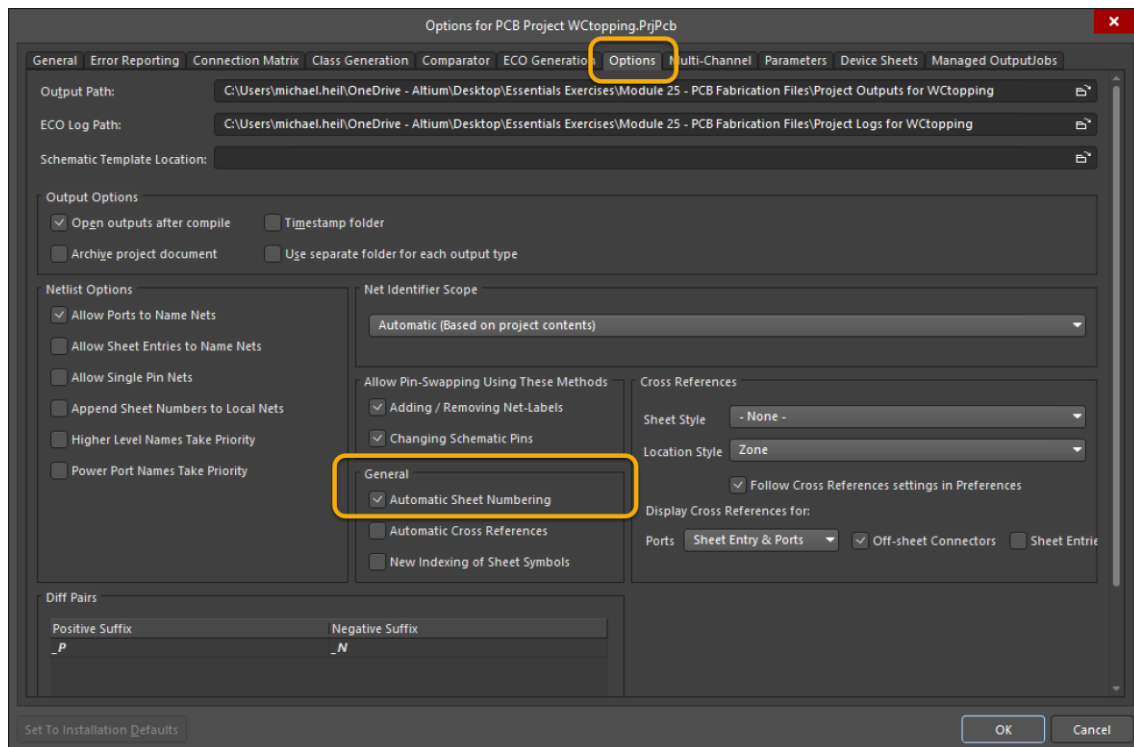


Figure 13. Sheet Annotation dialog with Sheet Numbering

27. The title block in each schematic will now show the proper sheet number according to the position in the Project Panel, as shown in Figure 14 below.

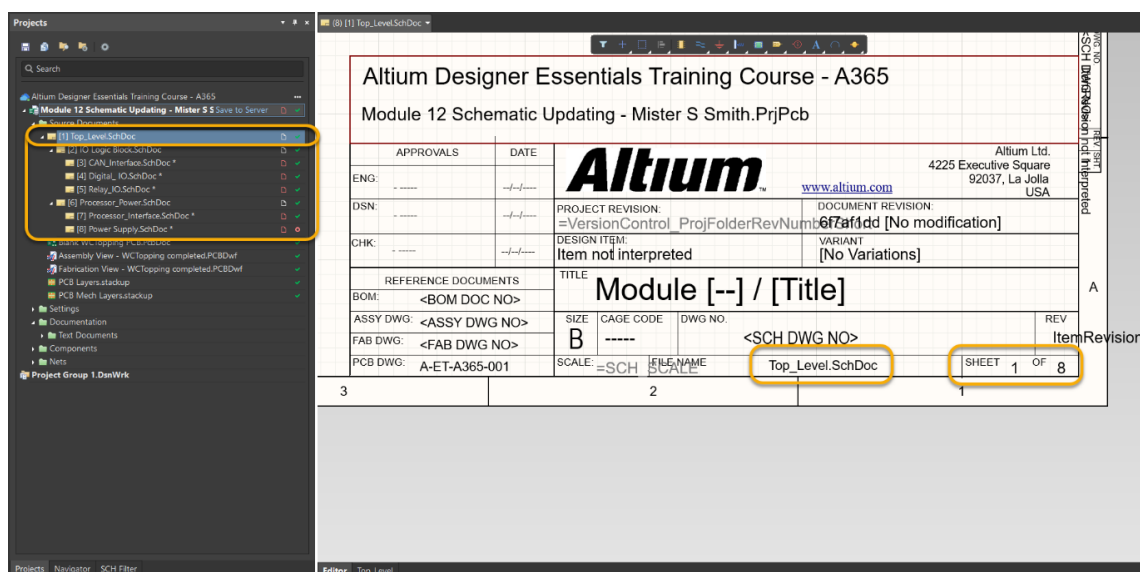


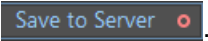
Figure 14. Title Block with Sheet number

28. Drag a sheet in the *Project* panel to a new position to see that the Sheet Number is automatically updated.



Hint: Enabling the checkbox for **Automatic Sheet Numbering (Project Option)** would re-number the sheet automatically. In addition, an icon would be added to each schematic sheet in the Projects panel indicating the order number of each sheet.

Hint: As alternative to the full automatic Sheet Numbering Mode Altium Designer offers a Interactive Mode that can be found at the **Tools** menu, **Annotation » Number Schematic Sheets....**

29. Select **File » Save All** to save all modifications.
30. Save the modifications to the server:
 - a) At the *Project* panel, next to the Project name you find the command **Save to Server** .
 - b) Select **Save to Server**.
 - c) At the dialog *Save [Project Name]*,
 - i) Add the comment `Module 12: Schematic Updating - [Add Your Name]- Finished`
 - ii) Click on **OK**
31. When ready, close the project and any open documents, **Window » Close All**.





Congratulations on completing the Module!

Module 12: Schematic Updating

from

**Altium Designer Essentials Training
with Altium 365**

Thank you for choosing **Altium Designer**

