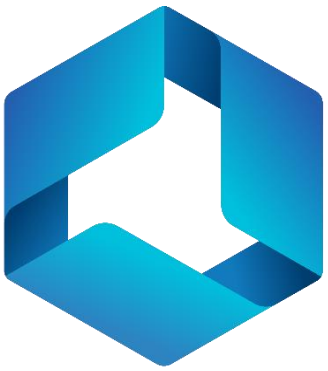


ALTIUM 365

Altium Designer

Essentials Course - Altium 365

Module 12: Schematic Updating

Table of Contents

Module 12: Schematic Updating

1.1 Purpose

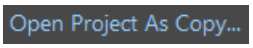
In this exercise, we'll perform modifications to multiple schematic objects at once, such as alignment, formatting and net color highlighting. These operations are what we refer to as Global Editing. We'll also look at the schematic annotation of components.

1.2 Shortcuts

Shortcuts 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)

1.3 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. Navigate to the predefined Training Project `Module 12 Schematic Updating (Top\Projects\Altium Designer Essentials Training Course\...)`
5. Select **Open Project as Copy…** 

6. 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.
7. 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.

For details how to Copy / Clone the predefined training project see Module 8 Making the Connection, Step 1.3 Preparation.

1.4 Align

1.4.1 **Align Right**

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

8. Open the `Processor_Interface.schdoc` from the *Projects* panel.
9. Select the ports on the left of the schematic using a left to right selection rectangle.
10. Go to **Edit » Align » Align Right** or use the **Active Bar** to align the ports, Figure 1.

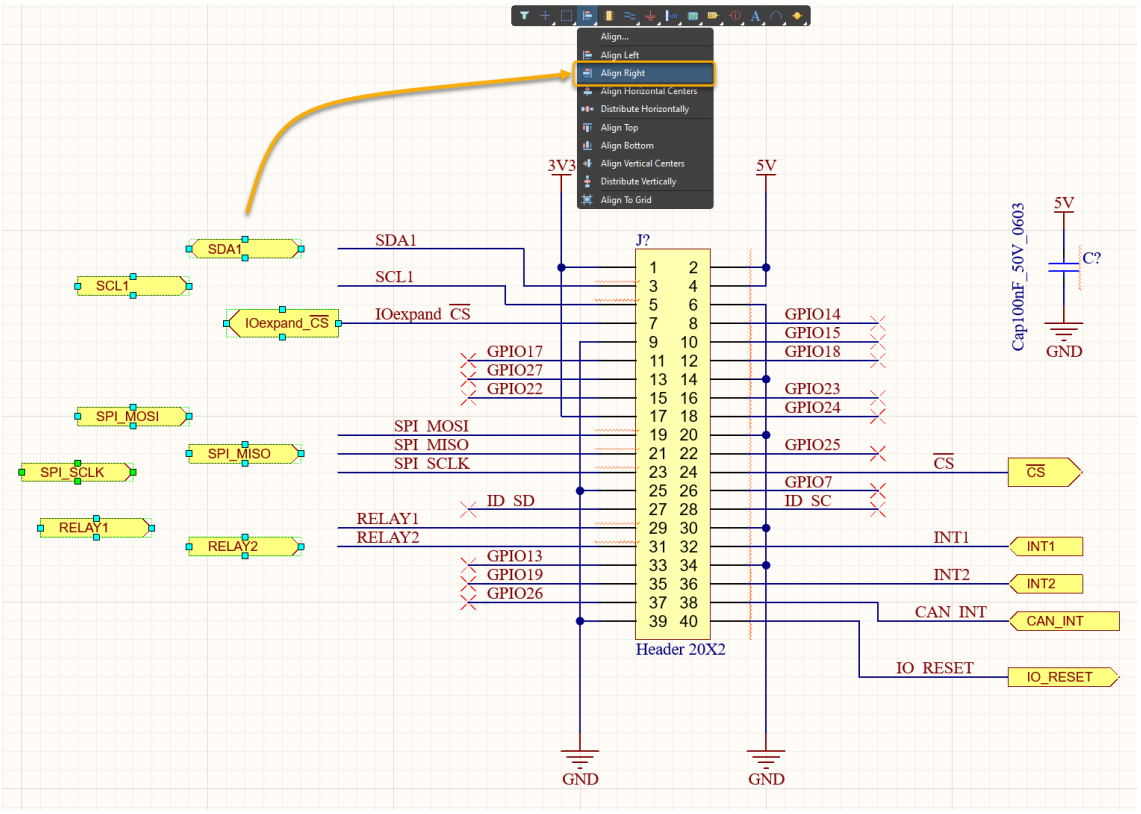


Figure 1. Align for unconnected Ports

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.

1.5 Net Color Highlighting

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

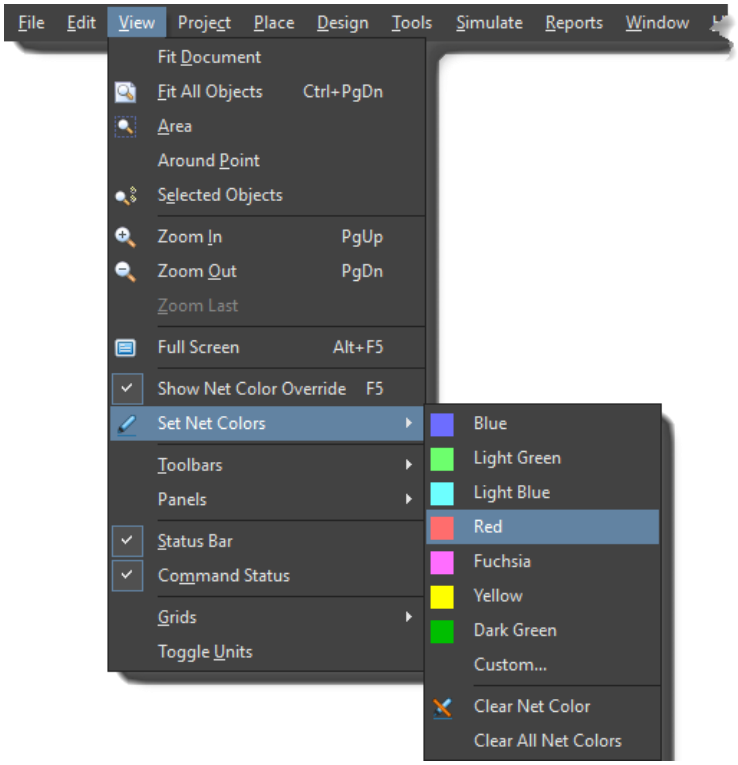


Figure 2. Set Net Colors menu

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

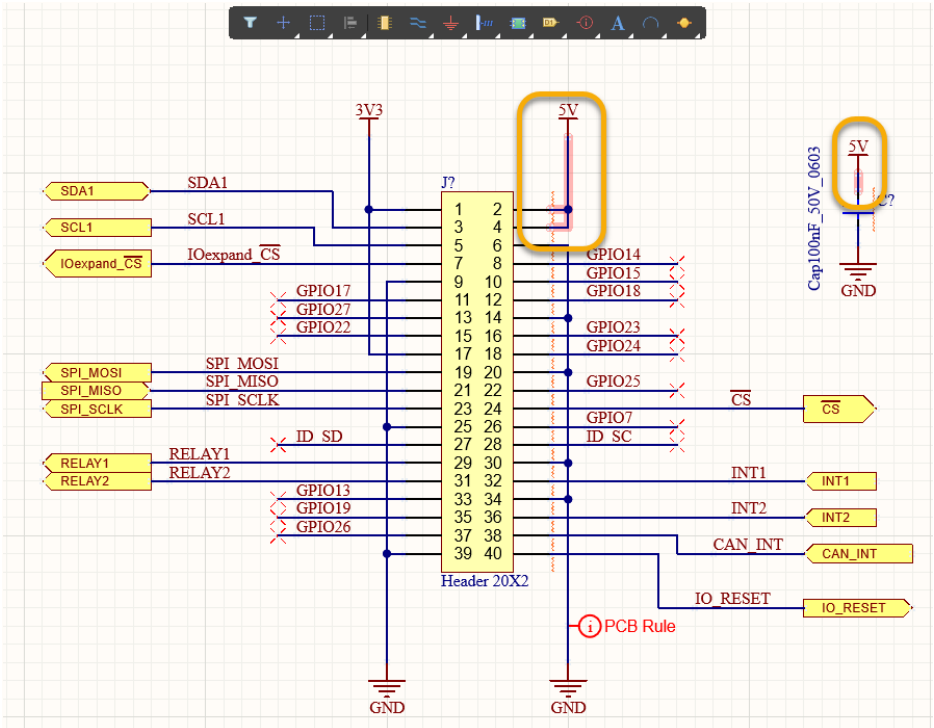


Figure 3. Processor_Interface.SchDoc with 5V net highlighted in red

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.

15. 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.
16. Repeat Step 12 to 14 for the Net `3V3` Volt (orange) and `GND` (blue).
17. Repeat Step 12 to 14 for the Net `12V` Volt (yellow). Use the *Navigator* panel or the *Project* panel to find the `12V` net.
18. To activate / deactivate the Net Color Highlighting press **F5**.

1.6 Schematic Annotation

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

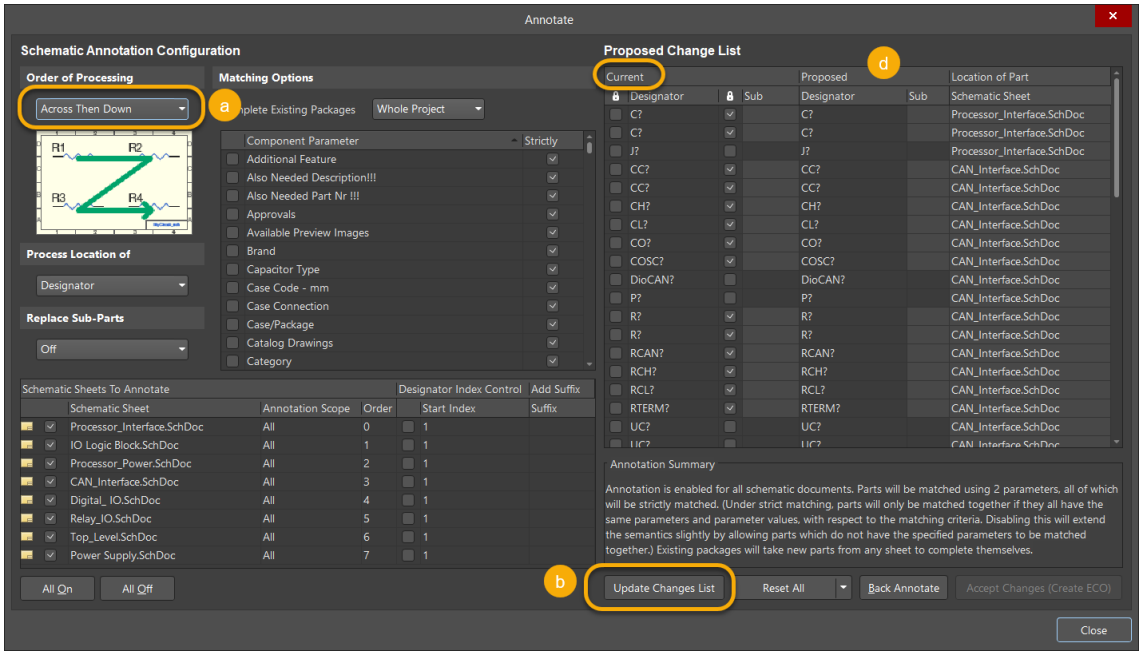


Figure 4. Annotate Dialog

a) Set the *Order of Processing* to **Across Then Down** using the pull-down menu.

b) Click the **Update Changes List** button.

c) Click **OK** on the pop-up dialog that appears telling us which changes have been made.

d) A preview of the proposed designator values will be displayed similar to what is shown in Figure 5.

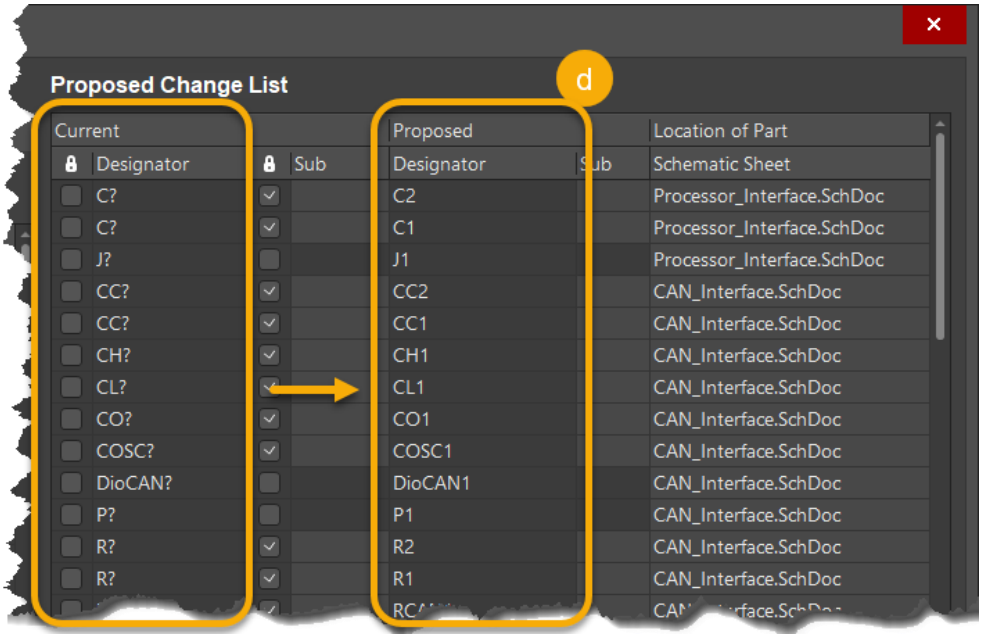


Figure 5. Proposed Designator changes based on the Annotation settings

20. These changes are only proposed so far. To commit this change to the design:
    a) press the **Accept Changes (Create ECO)** button.
    b) and then **Execute Changes** within the resulting *Engineering Change Order* as shown in Figure 6. All undesignated parts should now have a unique designator.
    c) Tick / Untick the Checkbox for **Only Show Errors** to reduce the list to reported errors.

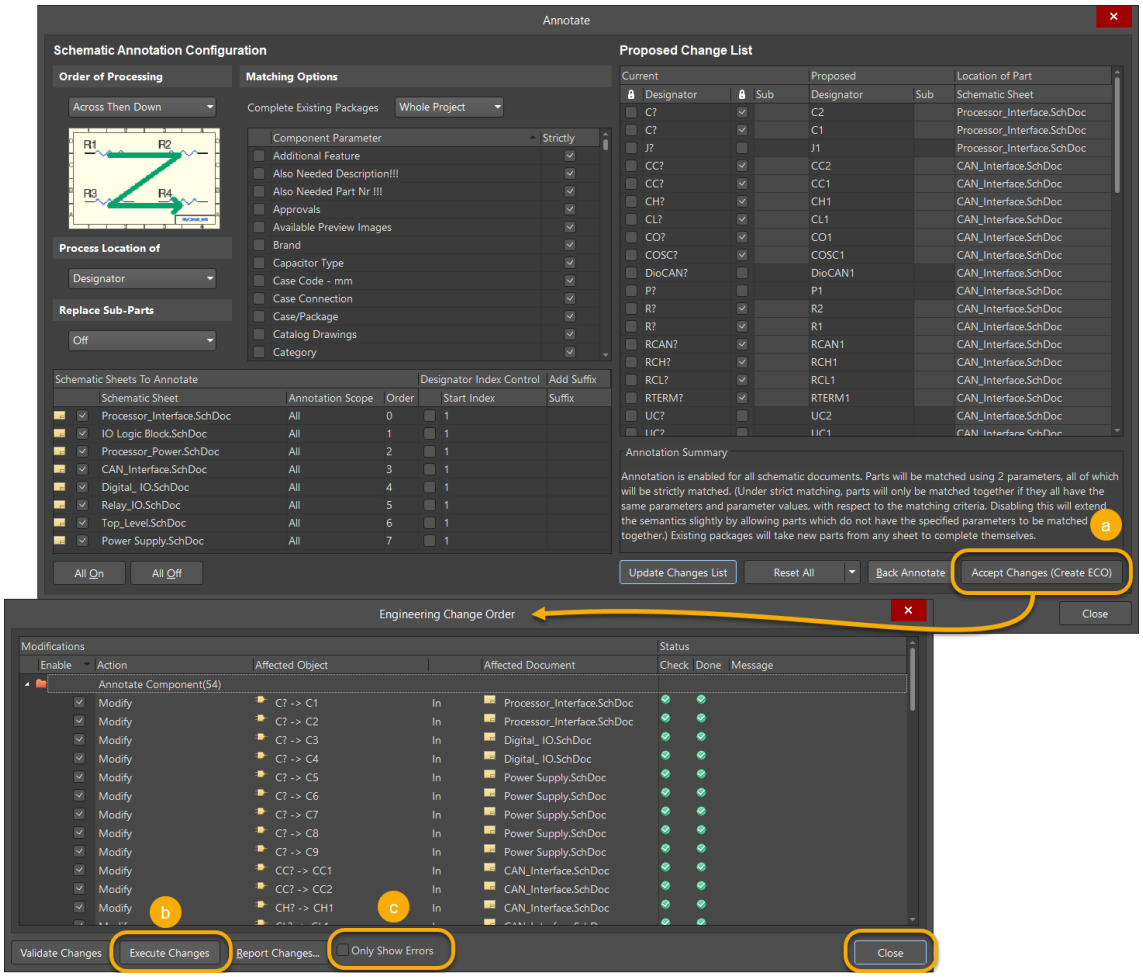


Figure 6. Engineering Change Order for the Annotation of components

21. Press the **Close** button to close the completed *ECO*, and then again press the **Close** button to close the *Annotate* dialog.
22. Review the schematics to confirm that they now have unique designators.

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.

1.7 Adding Directives for PCB

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

1.7.1 Assigning PCB Directives – Rule and Class Information

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

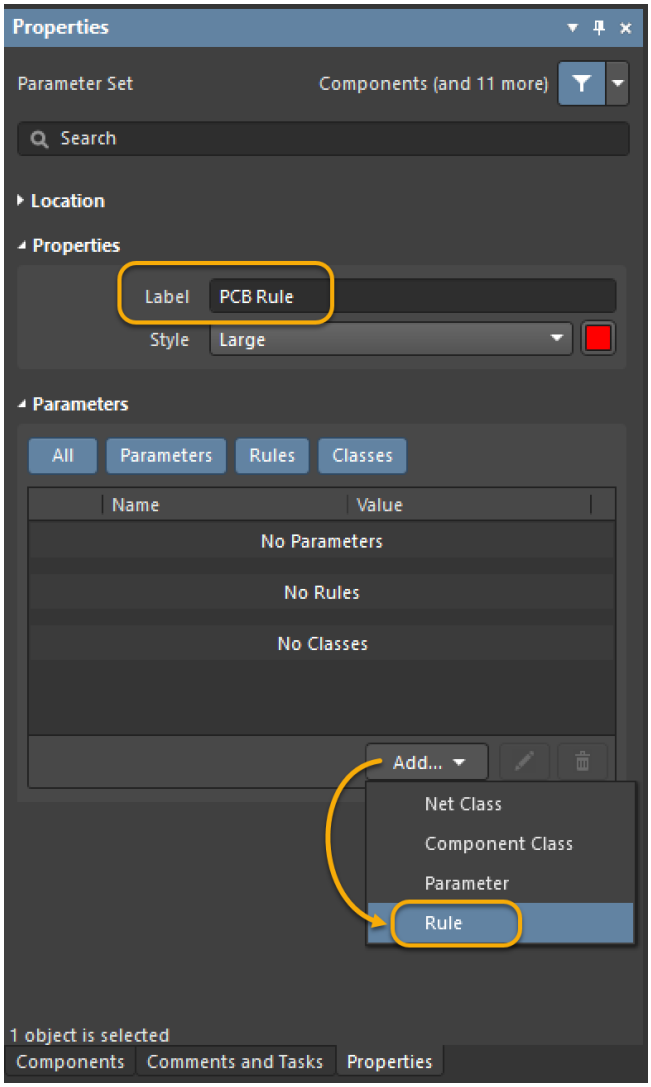


Figure 7. Adding PCB rule directive from the schematic

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

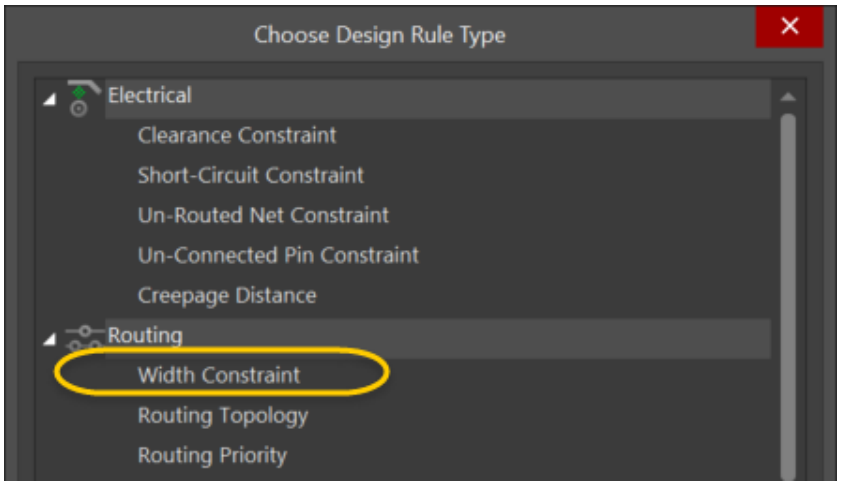


Figure 8. Width Constraint Rule

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`.

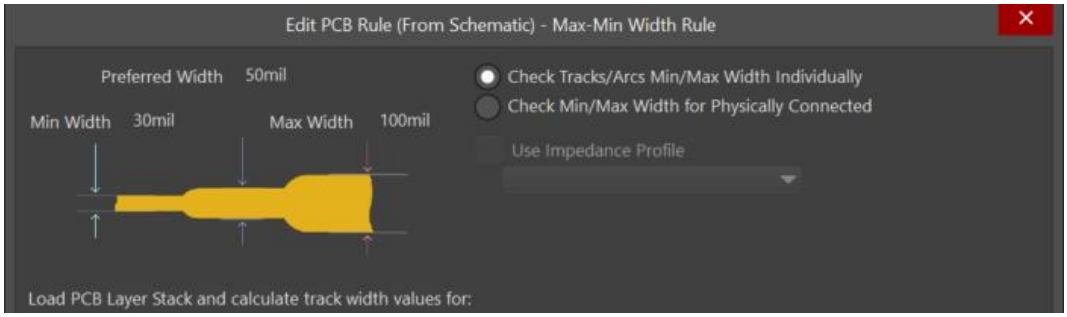


Figure 9. Width Rule settings

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

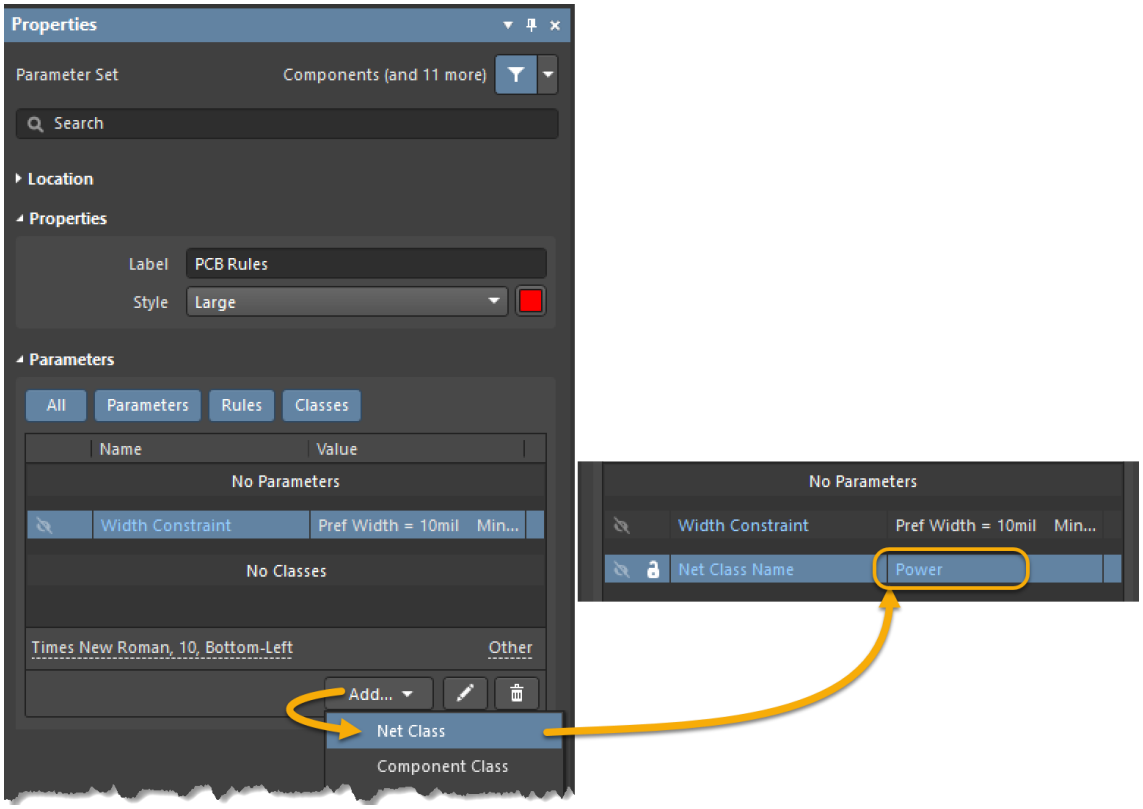


Figure 10. Adding Net Class from the schematic

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

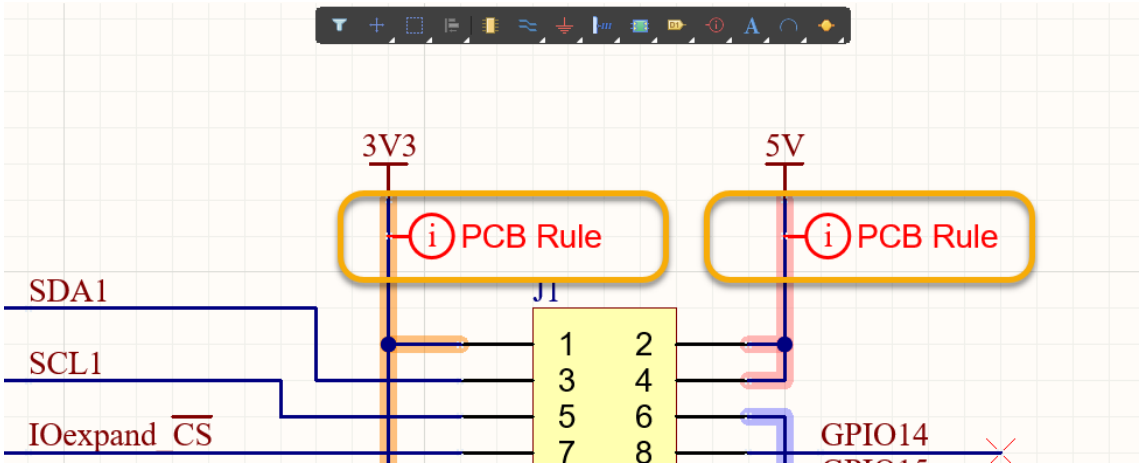


Figure 11. Placing the PCB Rule directives in the schematic

k) Right-click to exit the command.

1.8 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.

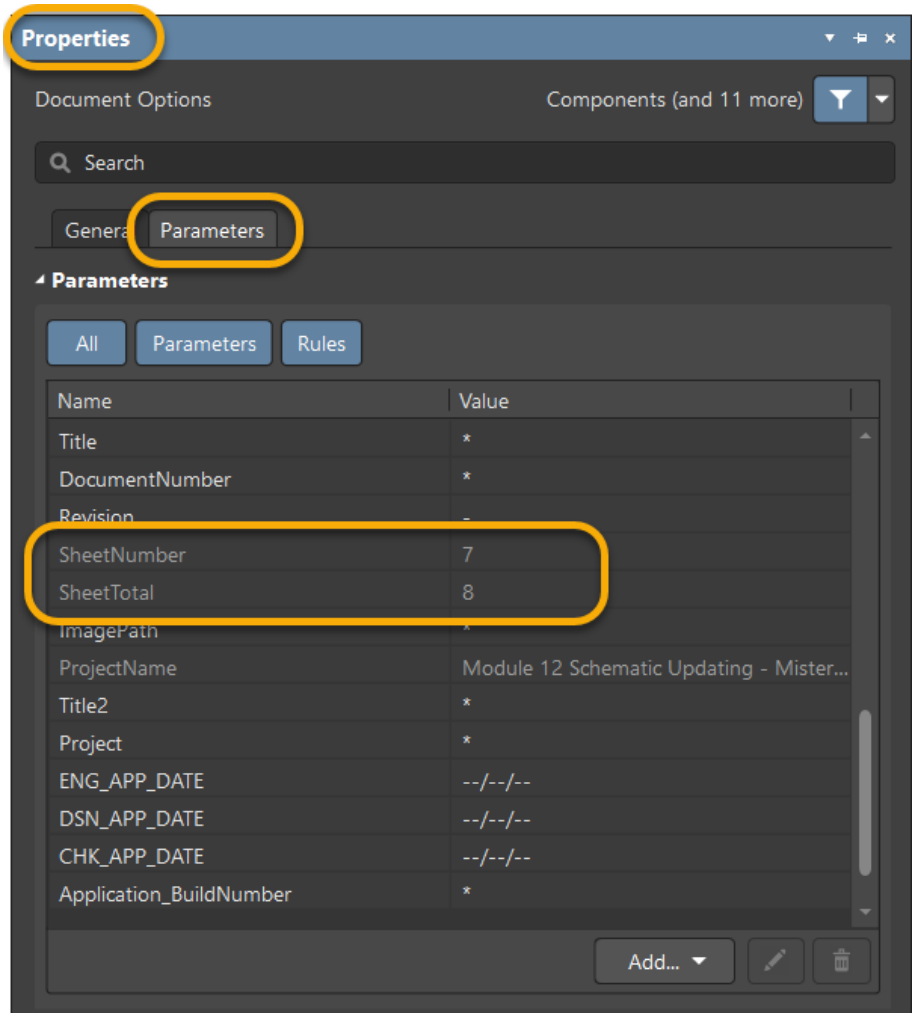


Figure 12. Sheet Number Parameters

25. From the **Projects** menu, select **Project » Project Options...** .
    The *Options for PCB Project* dialog will appear as shown at Figure 13 below.
    a) Select the Tab *Options*.
    b) Notice that the **Automatic Sheet Number** is active to assign numbers to the sheets.

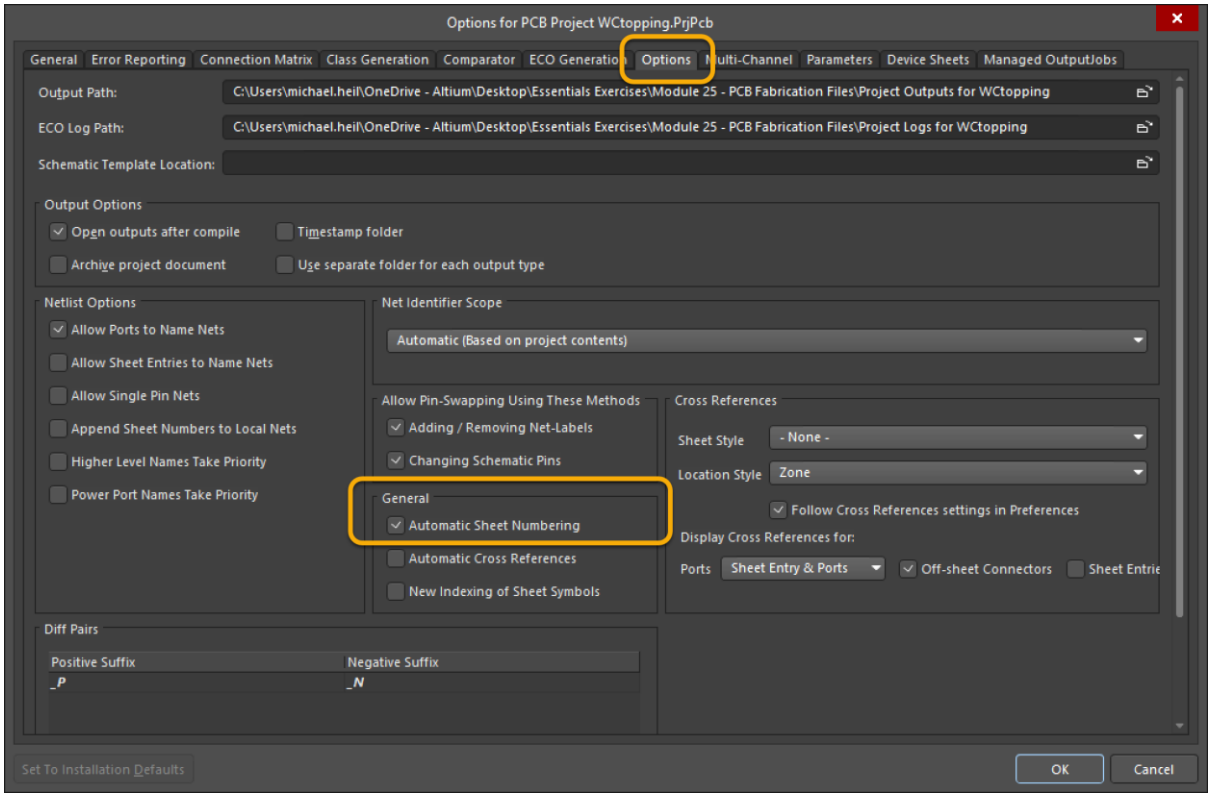


Figure 13. Sheet Annotation dialog with Sheet Numbering

26. 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.

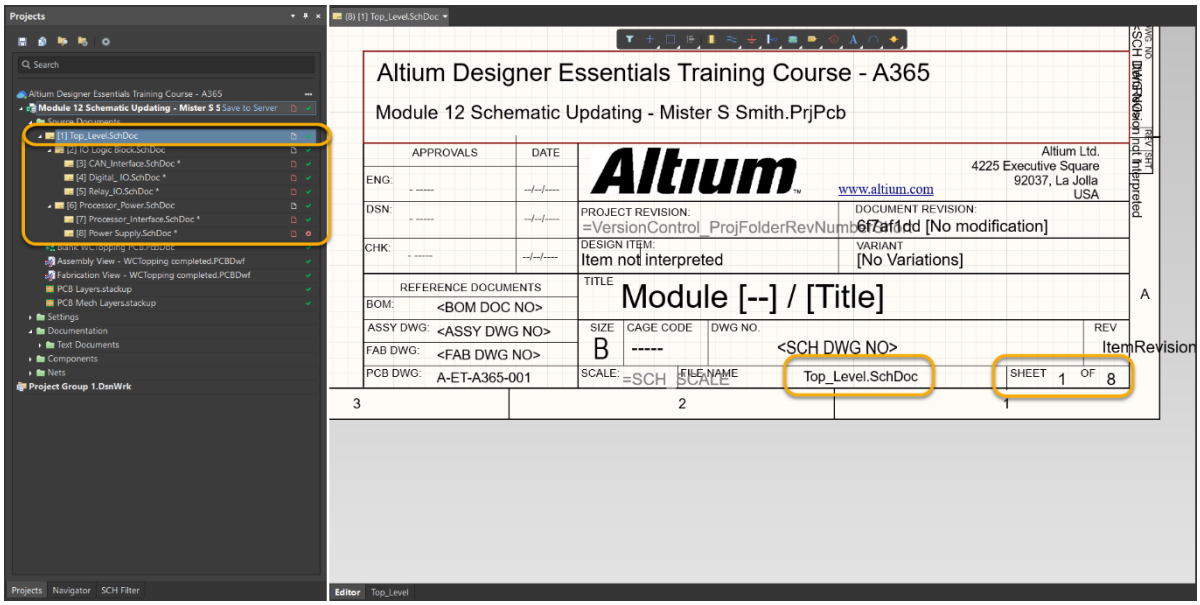


Figure 14. Title Block with Sheet number

27. Dragging a sheet in the *Project* panel to a new position and the Sheet Number will be automatically updated.

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.

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… .**

28. Select **File » Save All** to save all modifications.
29. 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) Activate the checkboxes for the files that are not under version control.
        ii) Add the comment `Module 12: Schematic Updating - [Add Your Name]-Finished`
        iii) Click on **OK**
30. **When ready, close the project and any open documents, Window » Close All.**

Congratulations on completing the Module!

Module 12: Schematic Updating

from the

Altium Designer Essential Course
with Altium 365

Thank you for choosing Altium Designer