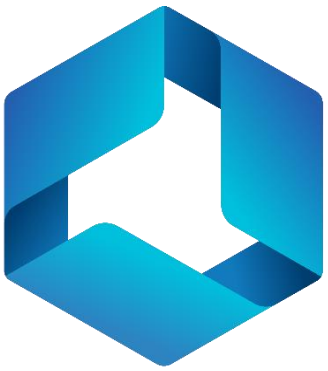

ALTIUM 365

Altium Designer

Essentials Course - Altium 365

Module 5: Schematic Preferences

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.
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 5: Schematic Preferences 3

1.1 Purpose 3

1.2 Shortcuts 3

1.3 Preparation 3

1.4 General Preferences 4

1.4.1 Display Cross-Overs 4

1.4.2 Components Cut Wires 5

1.5 Graphical Editing Preferences 8

1.5.1 Always Drag 8

1.5.2 Convert Special Strings 9

1.6 Grid Preferences 11

Module 5: Schematic Preferences

1.1 Purpose

In this exercise, you will learn about schematic preferences. These preferences allow you to customise your schematic environment and ensure document clarity and standardization.

1.2 Shortcuts

Popular shortcuts when working with Module 5: Schematic Preferences

or **T » P:**	Preferences
V » D:	View » Fit Document
V » F:	View » Fit All Objects
Mouse Wheel:	Page up - Page down
Mouse Wheel+Shift:	Page left - Page right
Mouse Wheel+Ctrl:	Zoom in - Zoom Out
Page Up:	Zoom in
Page Down:	Zoom out
CTRL during Drag**:**	Change to Move Mode
G:	Toggle Grid

1.3 Preparation

1. If you have closed the project `SL1 Xilinx Spartan-IIE PQ208 Rev1.02.PrjPcb` from the last module, `Module 4 Design Environment`, please reopen it. The project may also be available in the **File** menu, under **Recent Projects**.

There are three hierarchical levels of configurable settings in Altium Designer:

Preferences: these affect all projects
Project Options: these affect the open, selected project
Document Options: these are configured in the *Properties* panel with no objects selected

1.4 General Preferences

1.4.1 Display Cross-Overs

2. Open the `SL1 Xilinx Spartan-IIE PQ208 Rev1.02.SchDoc` schematic. Observe how the wires are currently displayed on the left side of component `U2,as` displayed in Figure 1 below. For document clarity, it is usually recommended for *Cross-Overs* to be displayed. This can be enabled in the Schematic Preferences.

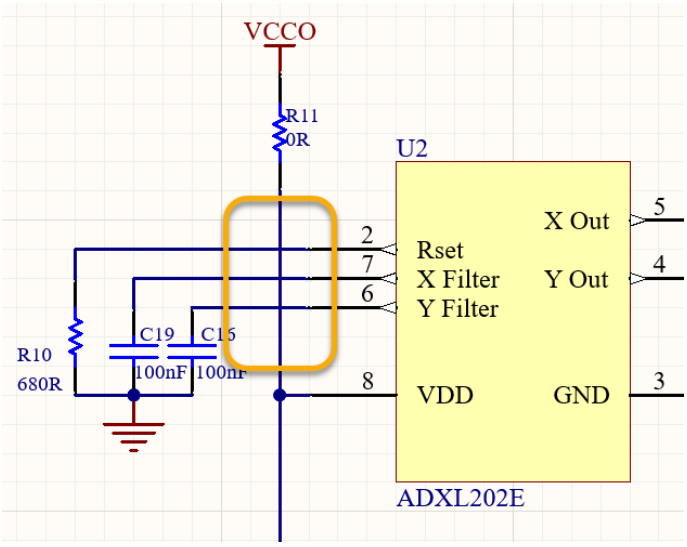

Figure 1. Schematic Connections for X1

3. Click Preferences, in the upper right corner of Altium Designer. Expand the *Schematic* branch and open the *General* page.
4. Enable **Display Cross-Overs** as shown in Figure 2 below.
5. Click **OK** to exit the Preferences**.**

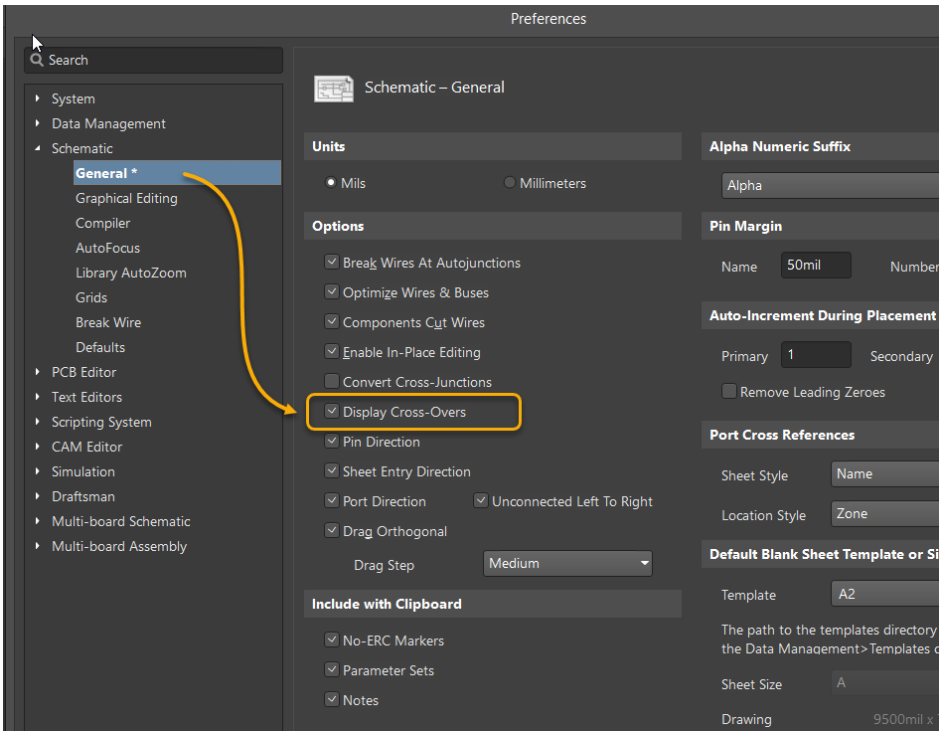

Figure 2. Display Cross-Over Preference

6. You'll notice that the cross-overs are now displayed as shown in Figure 3. If needed, press **Shift+C** or close the schematic document and re-open it to see the Cross-Overs.

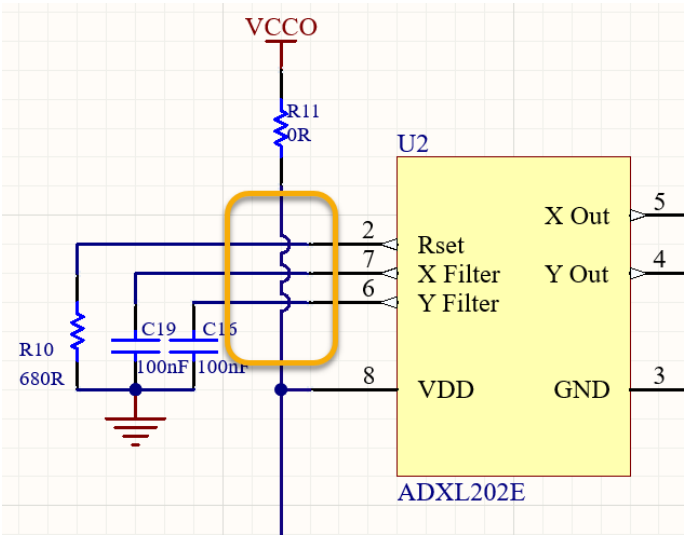

Figure 3. Cross-Overs being displayed after enabling it in Preferences

1.4.2 **Components Cut Wires**

In this section we will observe the effect of the *Component Cut Wires* feature. Go to the Preferences in the upper right corner of Altium Designer.

7. Expand the *Schematic* branch and open the *General* page. Verify that **Components Cut Wires** is enabled as shown in Figure 4.

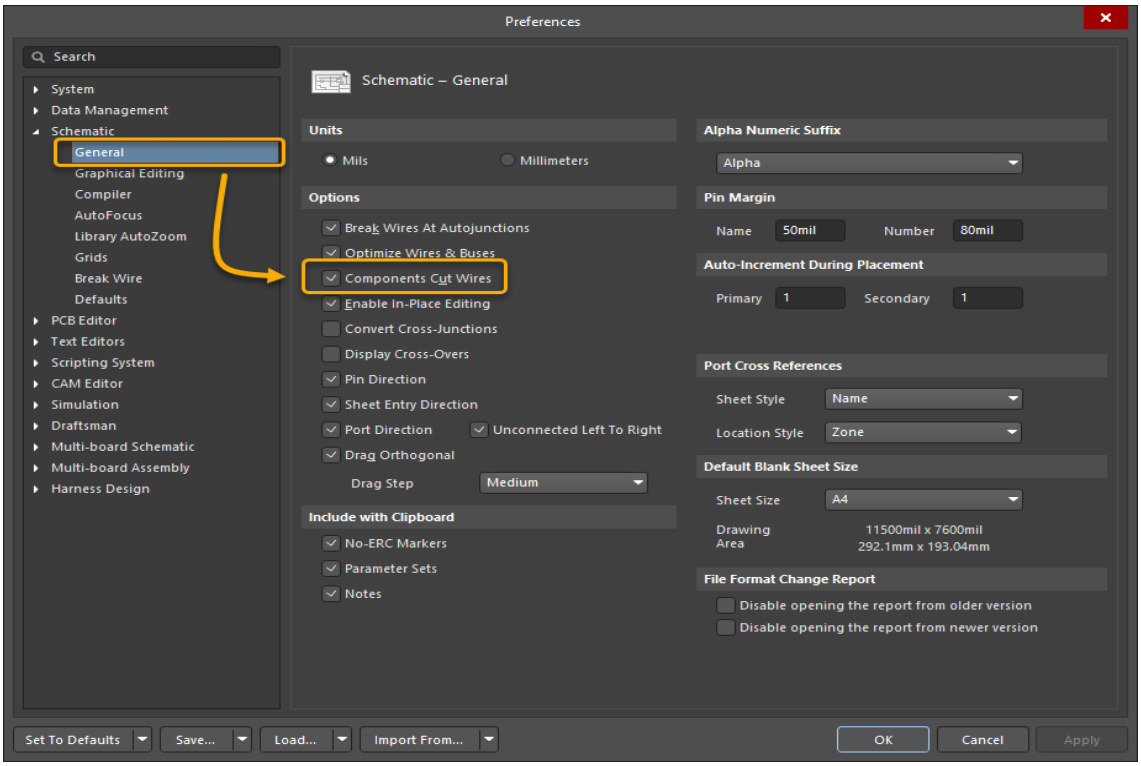

Figure 4. General and Graphical Editing Preference Settings

8. Click **OK** to save and close Preferences.

9. In the *Projects* panel, double-click on `SL1 Xilinx Spartan-IIE PQ208 Rev1.02.SchDoc` to open it as shown in Figure 5.

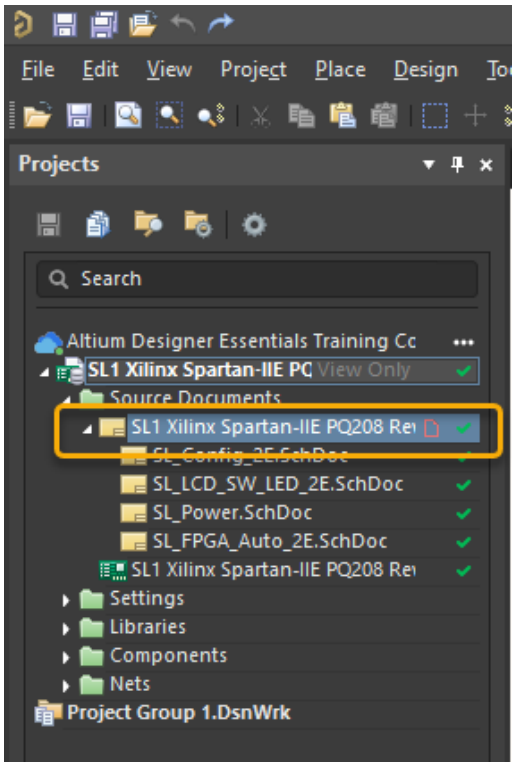

Figure 5. Projects Panel

10. Zoom into the area around U2, as shown in Figure 6.
    a) Select resistor R11 and copy from the right mouse menu (RMB) and select **Copy** or use **CTL+C.**
    b) Using the paste option either from the RMB or **CTL+V** paste the new resistor as shown in Figure 6.

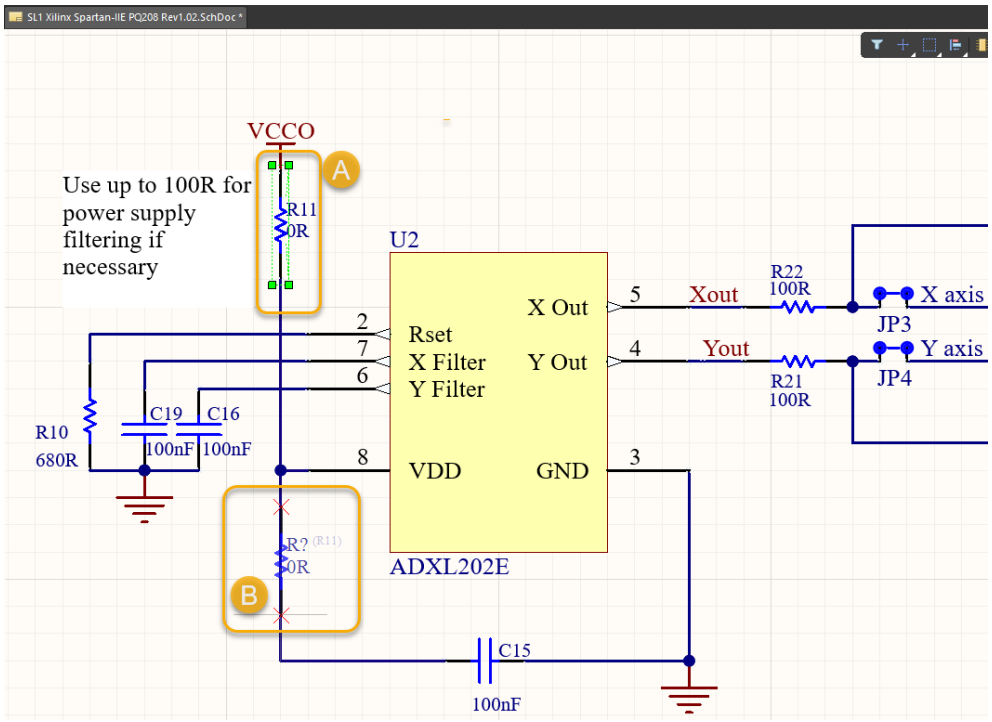

Figure 6. Copy and Paste

11. Notice how the wire has been removed or cut-away between the resistor pins after pasting. See Figure 7

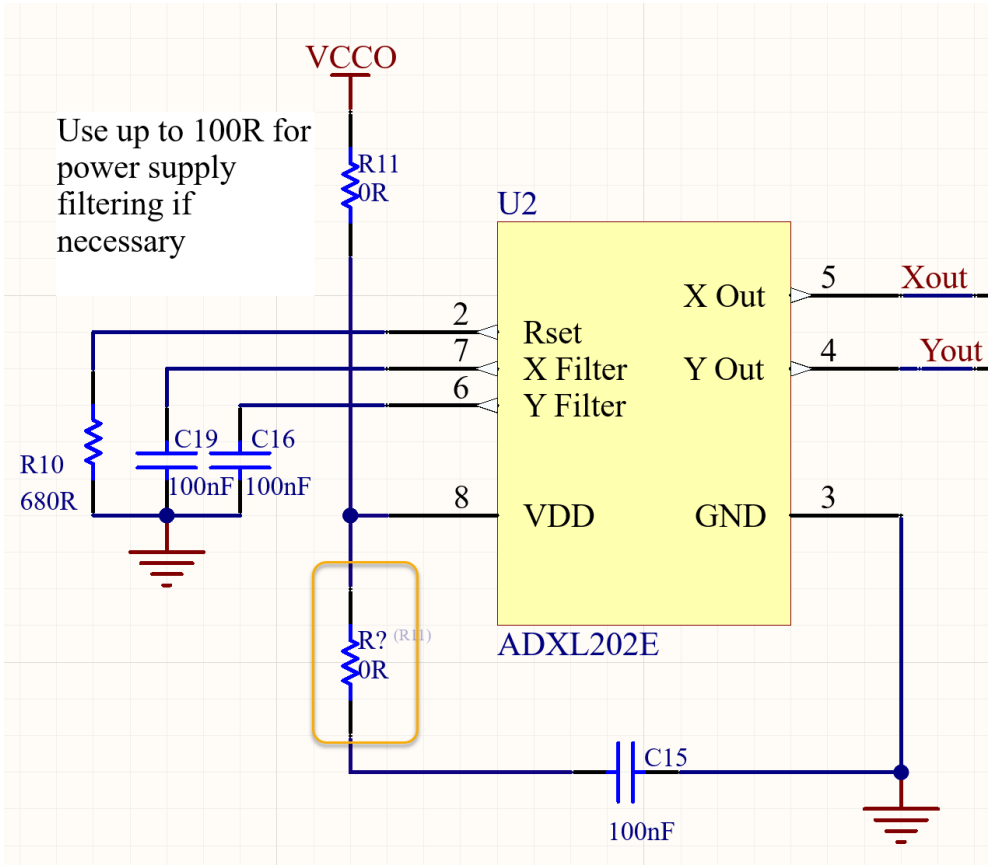

Figure 7. Placed Component

1.5 Graphical Editing Preferences

1.5.1 Always Drag

12. Make the schematic file `SL1 Xilinx Spartan-IIE PQ208 Rev1.02.SchDoc` the active document.
13. Open the Preferences in the upper right corner of Altium Designer. Expand the *Schematic* branch and open the *Graphical Editing* page. Ensure that the **Always Drag** option is enabled as shown in Figure 8. Click **OK** to close the Preferences dialogue.

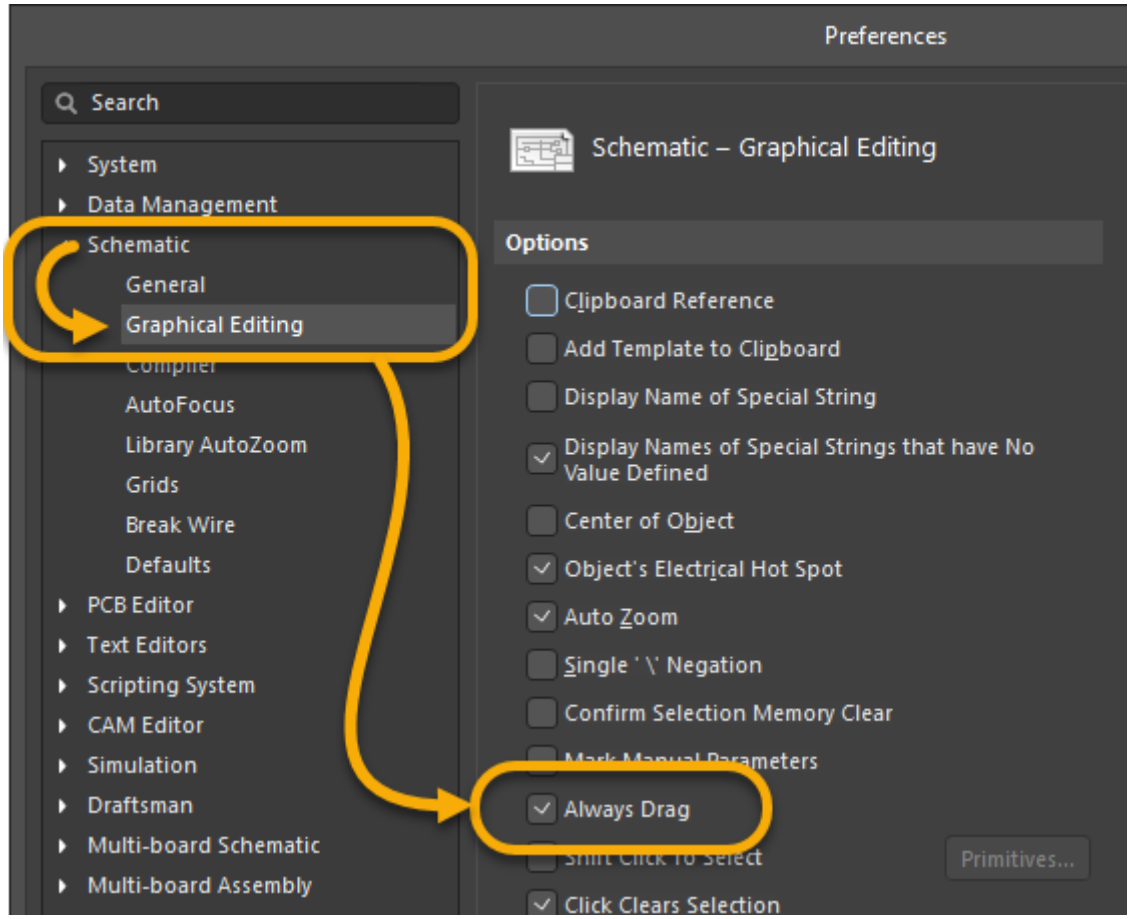

Figure 8. Always Drag Preference

14. Left click and hold, on *R10*.
    a) While holding the left-click, move the cursor to drag the component around. Notice the behavior of the connections where the wires will remain connected to the components.
    b) Hit **Escape** or **Right-Click** to abort the drag.
15. To move the component without maintaining connections, hold the **Ctrl** key, then left-click and drag *U2*.
    a) The component should now move independently of the previously connected tracks.
    b) Hit **Escape** or **Right-Click** to abort the drag.
16. Go to **Edit » Undo** or hit **Ctrl+Z** to undo any changes if necessary.

When dragging a component, ensure to check the connections that it is about to create. Before releasing the left mouse button, an indicator will show if there is a netlist modification or not.

1.5.2 **Convert Special Strings**

17. Open the Preferences .
18. Expand the *Schematic* branch and open the *Graphical Editing* page.
    a) Ensure that the option **Display Names of Special Strings that have No Value Defined** is enabled as shown in Figure 9.
    b) Feel free to toggle the Option **Display Names of Special Strings**.
    c) Click **OK** to exit the Preferences.

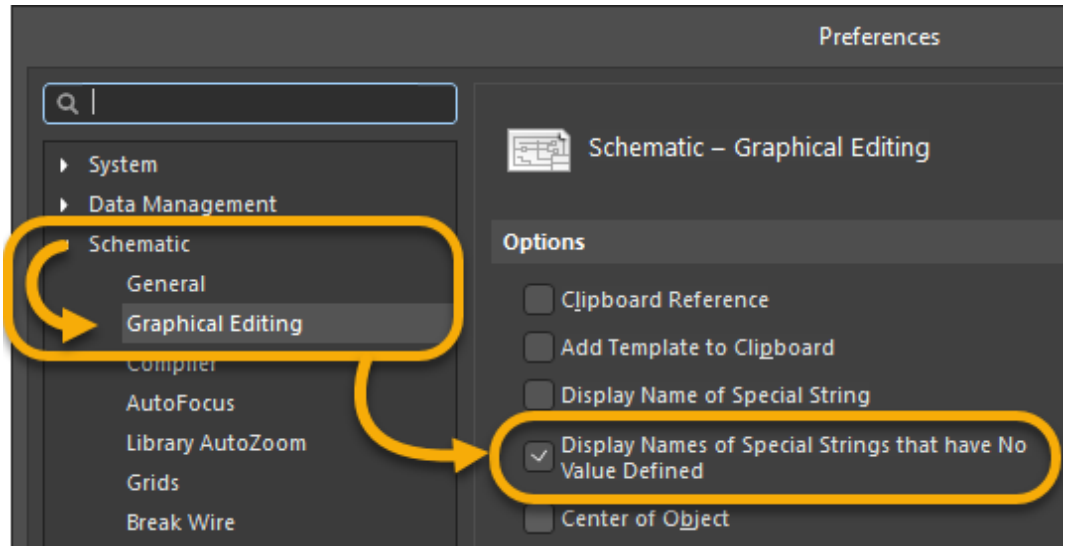

Figure 9. Display Names of Special Strings Preference

19. Navigate to the title block at the bottom right of the schematic sheet and notice the placed text strings. Some of these strings are pointing to parameters which currently do not have values yet. Hence, the parameter names are shown, instead of their values (**Display Names of Special Strings that have No Value Defined**) Figure 10.

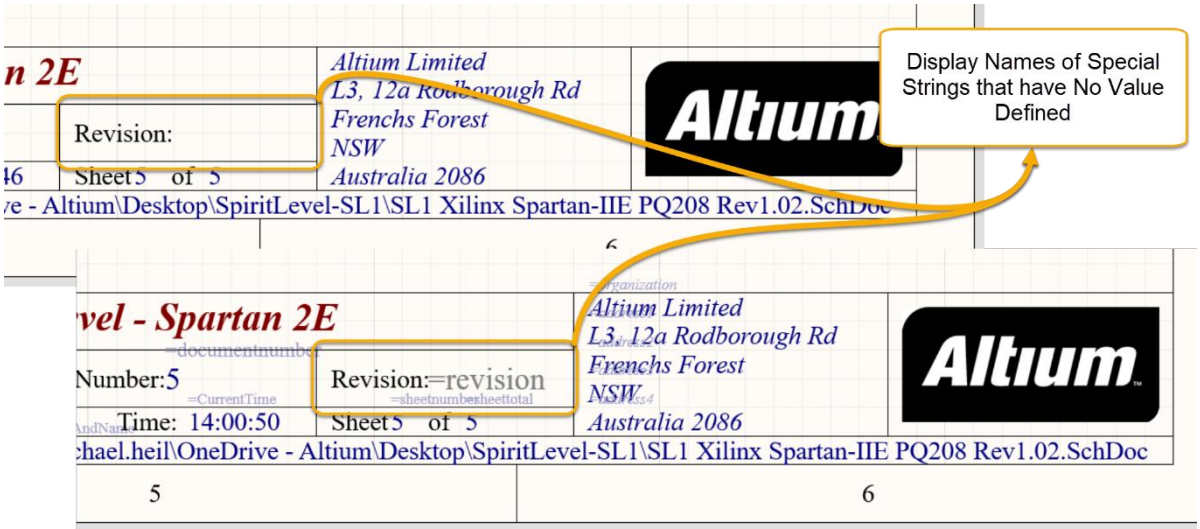

Figure 10. Visibility of Special Strings

20. We will make some changes to the Parameters as shown in Figure 11 below.
    a) Open the *Properties* panel from the **Panels** button and open the **Parameters** tab.
    b) Try changing the `Revision` parameter value to `1.0` and hit **Enter**. Notice the change on the title block.

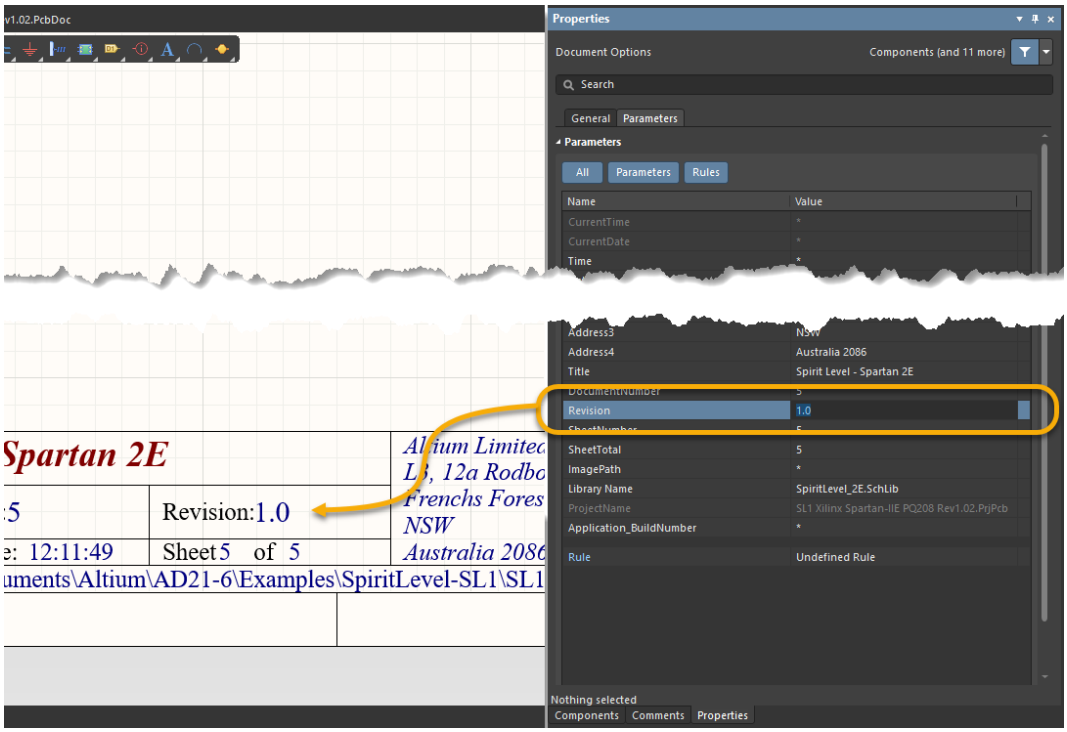

Figure 11. Document Parameters

21. Feel free to change the `Title` parameter value to `Spirit Level - Spartan 2E - Training`.

Special strings of the title block are defined & modified within the Schematic Template file *.SchDot. In general, the Schematic template creation and configuration is part of the Workspace configuration.

1.6 Grid Preferences

During the Training we use the Imperial System of measurement (mil) for the schematic grid. This is because the Training Libraries - SCHLIB - are defined using imperial unit of measurement.

22. Click Preferences and expand the *Schematic* branch and open the *Grids* page.
23. Under the *Imperial Grid Presets* section, click **Altium Presets** and select **Coarse (3 settings)** as shown in Figure 12 below.
24. Click on the color box to the right of **Grid Color** if you wish to change the color of the grid.
25. Click **OK** to save and close the dialogue.

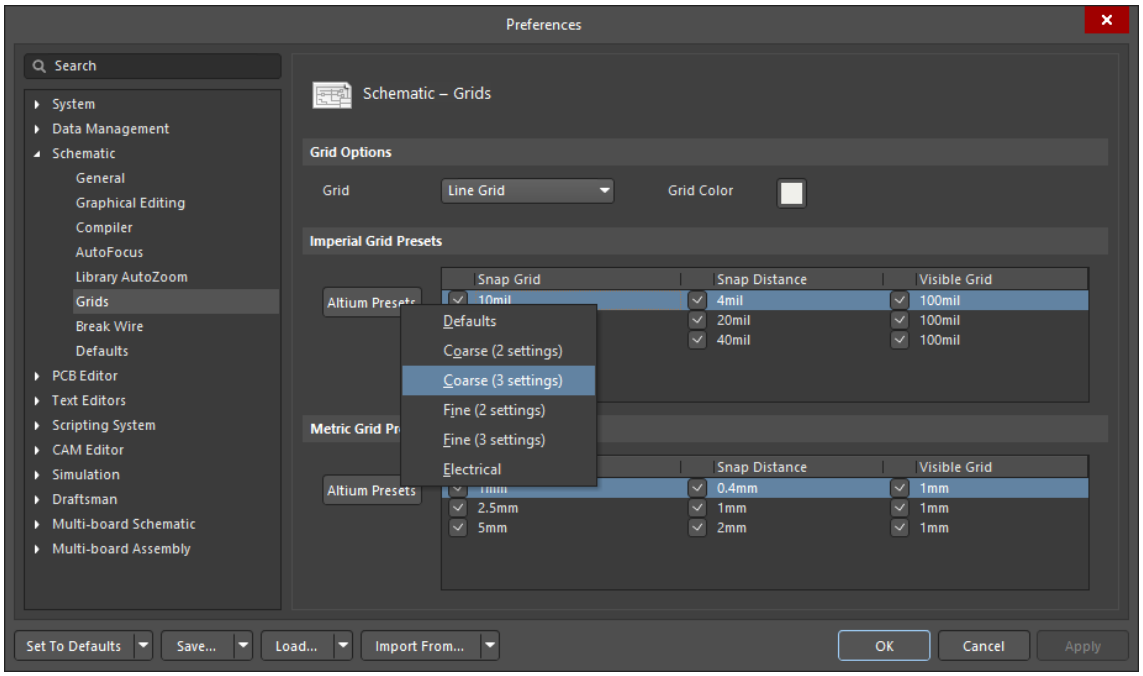

Figure 12. Grid Settings in Preferences

26. In the schematics, toggle the snap grid through your new presets by hitting the **G** key. You can see the grid change between 50mil, 100mil, and 200mil on the *Status Bar* as shown in Figure 13 below.

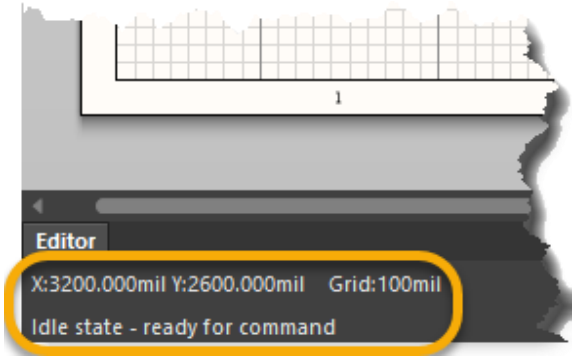

Figure 13. Grid Information from the Status Bar

We can hit the F1 key while in a Preferences page or any general menu to open the corresponding online documentation regarding the active window or menu.

27. **Do not close the project, we will use this project in the next module as well.**

Congratulations on completing the Module!

Module 5: Schematic Preferences

from the

Altium Designer Essential Course
with Altium 365

Thank you for choosing Altium Designer