



# Altium Designer

## Essentials Training with Altium 365

### Module 11: Schematic Graphics

**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>Module 11: Schematic Graphics</b>	<b>3</b>
<b>1 Purpose</b>	<b>3</b>
<b>2 Shortcuts</b>	<b>3</b>
<b>3 Preparation</b>	<b>3</b>
<b>4 Adding Text String</b>	<b>4</b>
<b>5 Adding Rectangle</b>	<b>6</b>
<b>6 Formatting Net Labels</b>	<b>8</b>
6.1 Selecting All Net Labels	8
6.2 Apply Formatting to Net Labels	9





# Module 11: Schematic Graphics

## 1 Purpose


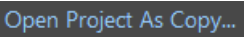

In this exercise you will add text and graphics to the schematics to provide documentation and organization. You will also number schematic sheets.

## 2 Shortcuts

Shortcuts used when working with Module 11: Schematic Graphics

P » T	Place Text
P » F	Place Text Frame
P » D » R	Place Rectangle
Shift+C	Clear masking
Shift+F	FSO – Find Similar Objects

## 3 Preparation


1. Close all existing projects and documents.
2. Next, create a Copy / Clone of the Training Project `Module 11 Schematic Graphics`.
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 11 Schematic Graphics` (`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 11 Schematic Graphics - [Your Name]`.
  - b) Add a description: `Altium Essential Training - Module 11 - [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 on how to Copy / Clone the predefined training project, see Module 9, Making the Connection, Step 3 Preparation.





## 4 Adding Text String

9. Open the `CAN_Interface.SchDoc` from the *Projects* panel.
10. Place a text string by selecting **Place » Text String**. This can also be done from the **ActiveBar** .
11. Before placing it, press the **Tab** key to edit its properties as shown in Figure 1.
  - a) In the Text field, enter `3.3V Logic Levels`.
  - b) Change the Font size to 16.
  - c) Press **Enter** or hit the **Pause** icon to continue with the text placement.

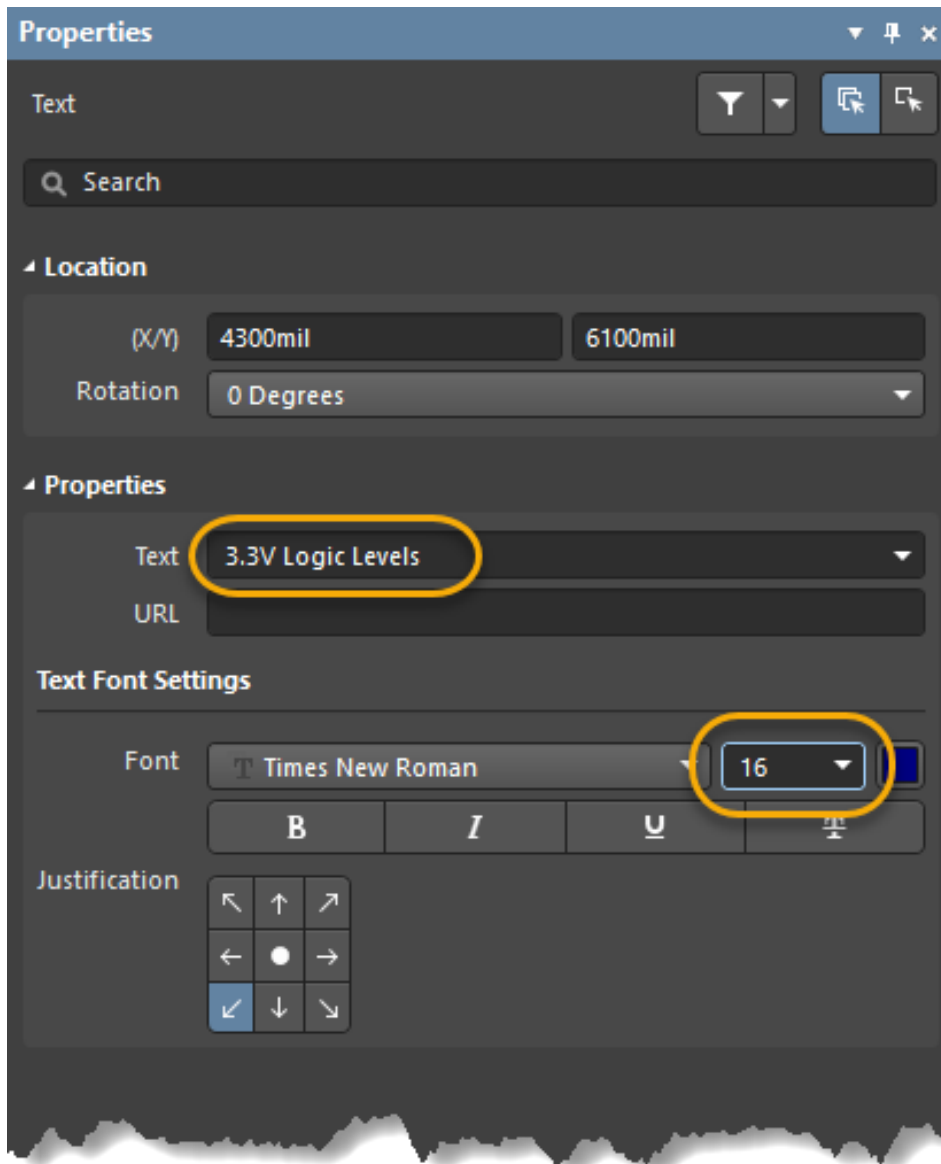


Figure 1. Font dialog in the Properties panel



12. Left-click to place the string above component MCP2515 as shown in Figure 2 below.
13. With the place string command still active, we will place another text string.
  - a) Press the **Tab** key to edit the properties.
  - b) In the Text field, enter 5V Logic Levels.
  - c) Press **Enter** or hit the **Pause icon** to continue with the text placement.
14. Place the 5V Logic Levels above of component MCP2551 as shown in Figure 2 below.

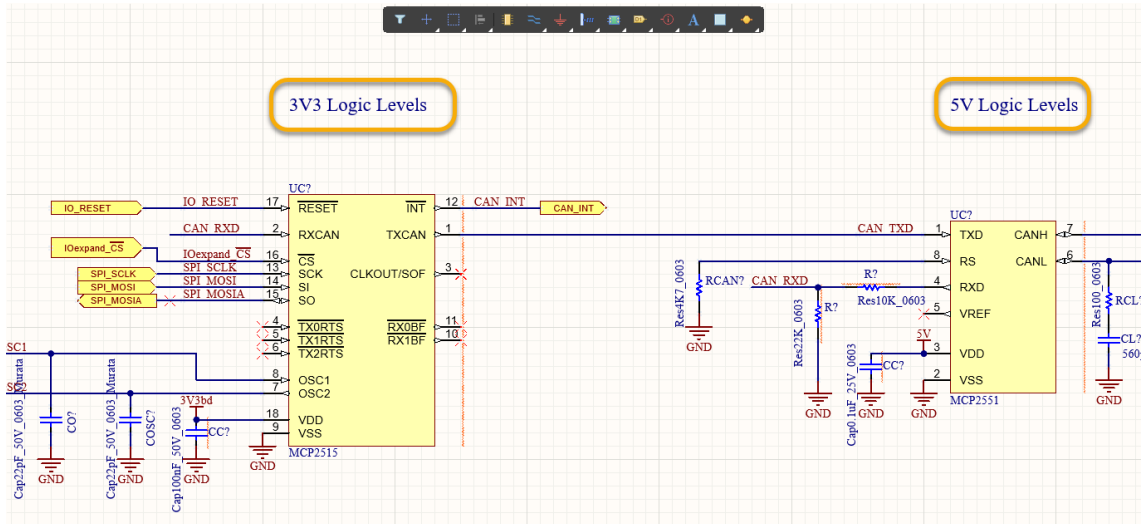



Figure 2. Placing a Text string

## 5 Adding Rectangle

15. Place a Rectangle by selecting **Place » Drawing Tools » Rectangle** or use the **Active Bar** .
16. Left-click to place the top left corner of the rectangle similar to what is shown in Figure 4 below.
17. Press the Tab Key, open the *Properties* panel and activate the option **Transparent**, Figure 3.
18. Move the mouse in a diagonal manner to cover the components and wire as shown in Figure 4 below.
19. Left-click place the lower right corner of the rectangle.
20. Right-click to escape the command.

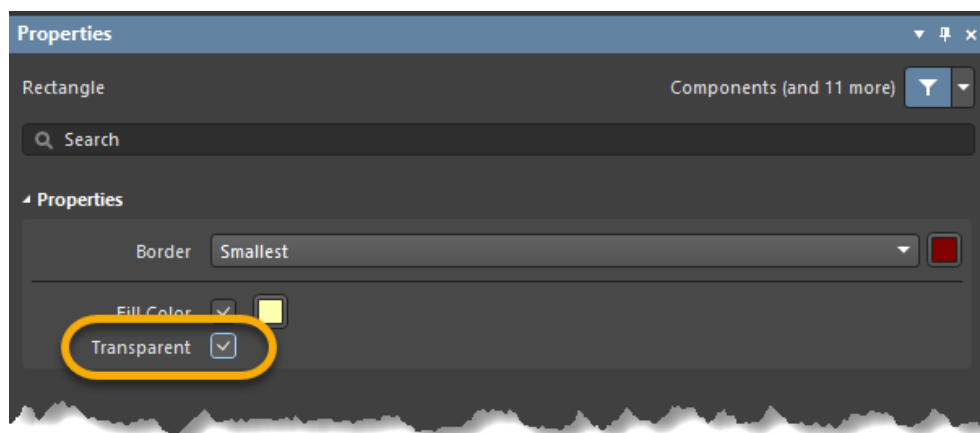


Figure 3. Transparent Option for Rectangle

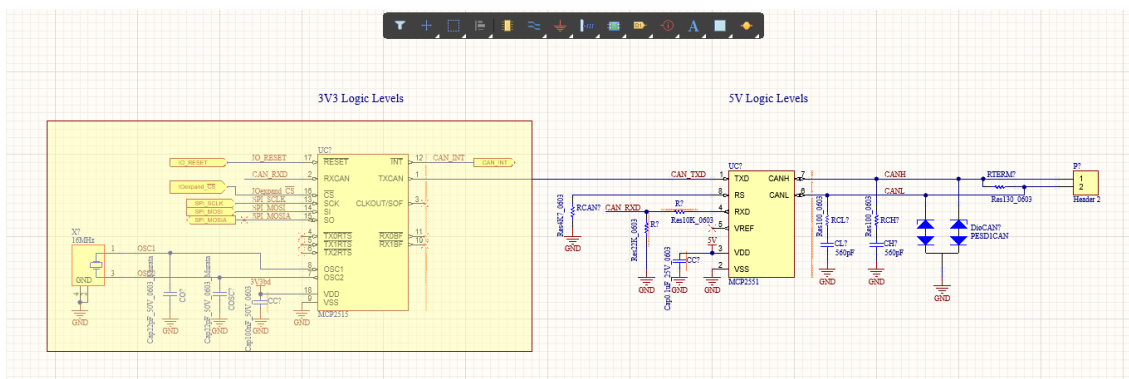


Figure 4. Rectangle with transparent mode

21. Using the same method as above, place another rectangle for the right part of the schematic, see Figure 5.

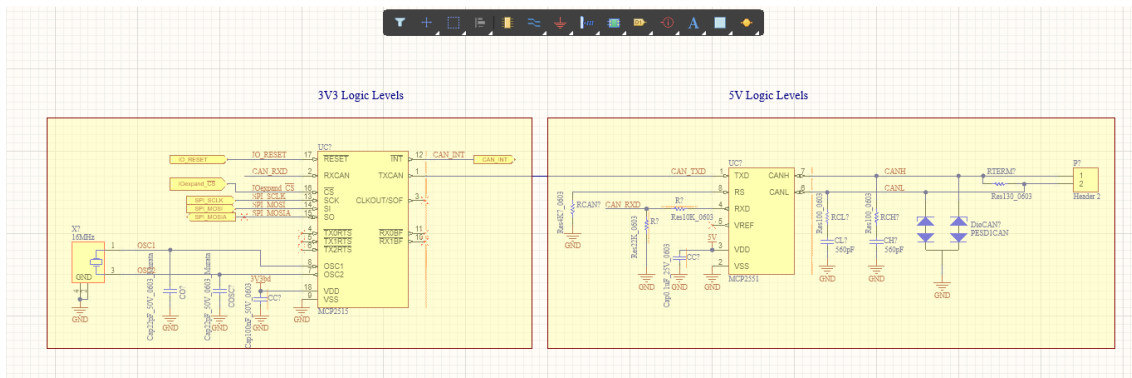


Figure 5. Placing a second rectangle

22. Select both rectangles using **Shift+Left-click**. Then, open the *Properties* panel.
23. Click on *Fill Color* field and set the fill color to **light blue** as shown in Figure 6 below.

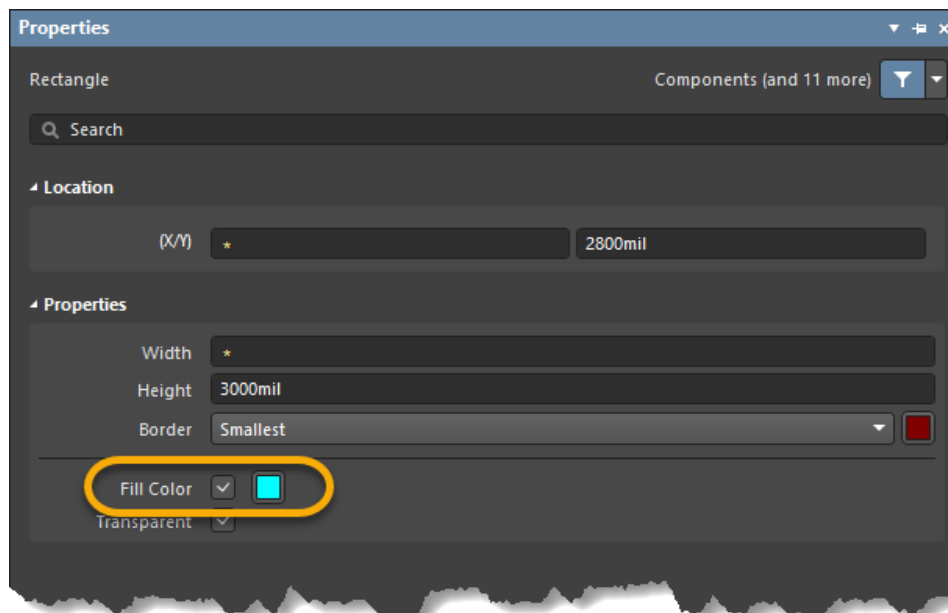


Figure 6. Select Light Blue for the Fill Color

24. Once you've changed the properties, the schematic should look similar to Figure 7 below.

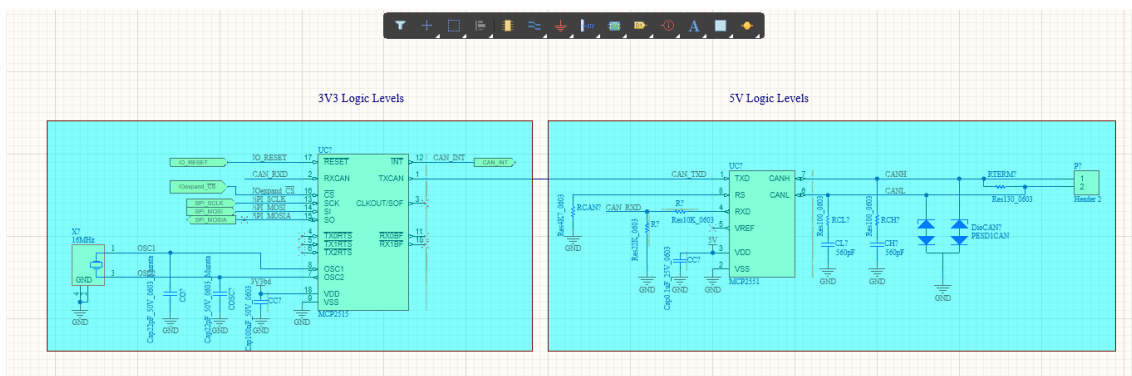


Figure 7. CAN\_Interface schematic with text and rectangle placement





## 6 Formatting Net Labels

### 6.1 Selecting All Net Labels

In this section, we will modify all the Net Labels in the project to ensure they have the same font, font size, and color.

25. Before we change the Net Label, let's configure the Selection Filter. By configuring the Selection Filter it is possible what is selected.
26. Open the selection filter from the **Active Bar** and deselect **Drawing objects**, as seen at Figure 8.

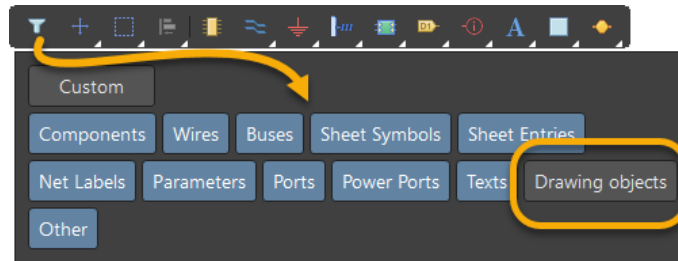


Figure 8. SCH Filter for Objects


27. After applying a filter selection, note the filter menu will have a green dot indicating that a filter selection is active .
28. Right-click on one of the net labels such as `IO_Reset`. In the right-click menu, click on **Find Similar Objects...** (also referred to as "FSO" for short). This will open the *Find Similar Objects* window.
29. Set the selection options as defined in Figure 9 below.



Figure 9. Find Similar Objects configuration for Net Label selection

30. Click **OK**. Notice that all of the Net Labels in the schematic are now selected.
31. Without clicking anywhere on the schematic, open the `Digital_IO.SchDoc` schematic. Notice that the Net Labels in this schematic are also selected.
32. Return to the `CAN_Interface.SchDoc`.





## 6.2 Apply Formatting to Net Labels

33. While the Net Labels remain selected, open the *Properties* panel if it is not already open.
34. Change the text font to `Verdana` and change the text size to 9.
35. Click on the color box within this panel and click on a dark green color.
36. Then, click on **I** to italicize the net labels shown in Figure 10 below.

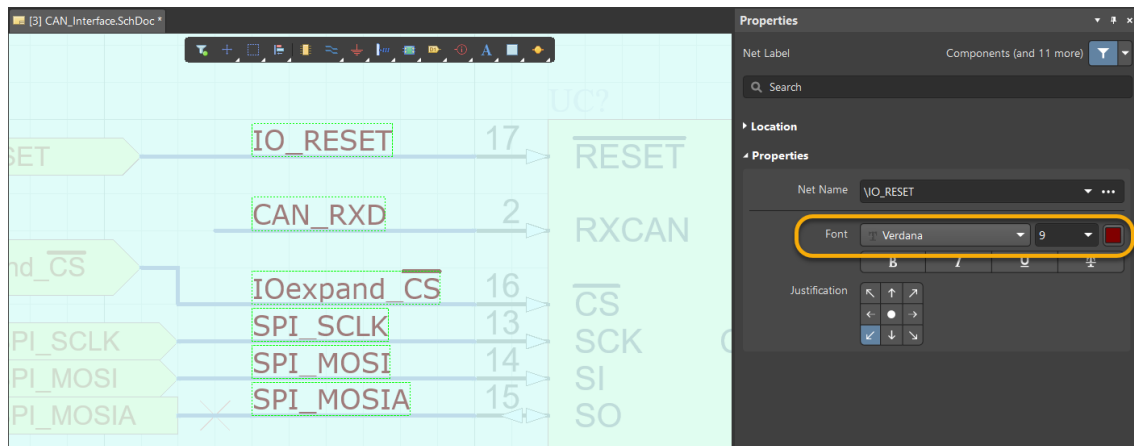

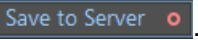


Figure 10. Net Label formatting defined in the Properties panel

37. The formatting defined in this panel will be applied on all the Net Labels selected in the project. Left-click once in the main workspace and hit **Shift+C** to clear the highlight and mask.
38. Change the Selection Filter back to default .
39. Select **File » Save All** to save all modifications.
40. 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 11: Schematic Graphics - [Add Your Name] - Finished
    - ii) Click on **OK**
41. When ready, close the project and any open documents.





**Congratulations on completing the Module!**

Module 11: Schematic Graphics

from

**Altium Designer Essentials Training  
with Altium 365**

Thank you for choosing **Altium Designer**

