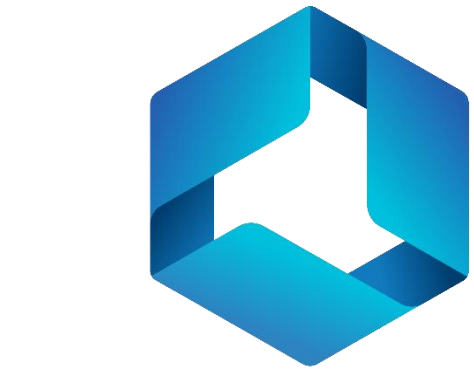


ALTIUM 365

Altium Designer

Essentials Course - Altium 365

Module 9: Making the Connections

Table of Contents

Module 9: Making the Connections

1.1 Purpose

In this exercise, you will complete the Processor_Interface, CAN_Interface and Digital_IO schematics by adding connectivity using wires, net labels, power ports and signal ports.

1.2 Shortcuts

Popular shortcuts when working with Module 9: Making the Connections

P » W:	**Place » Wire**
P » O:	**Place » Power Port**
P » R:	**Place » Port**
P » N:	**Place » Net Label**
P » B:	**Place » Bus**
Spacebar:	Change the angle of a wire corner during placement.
Shift+Spacebar:	Change the corner mode of a wire during placement.
Backspace:	Unwind an inadvertently placed segment during placement**.**
TAB:	During Placement of an Object, open Properties panel
Right-click or **Escape:**	End placement mode.

1.3 Preparation

1.3.1 General

1. **Close all existing projects and documents.**

1.3.2 Load the Existing Predefined Training Project.

Next, we will open a predefined training project located within the training workspace `Altium Designer Essentials Training Course - A365`. These predefined training projects are protected to prevent accidental modifications. You can open the project, but you are not allowed to save any modifications back to the server.

For the `Training Altium Designer Essentials Training with Altium 365`, we have implemented a specific workflow that enables you to Copy/Clone the Predefined Training Project, creating a new local copy that will be saved in your personal training folder within the workspace

1.3.2.1 Option One

2. As previously shown, we will first make a copy of the project into our own folder.
3. Select **File » Open Project…** to open the *Open Project* dialog, **Error! Reference source not found.**.
4. Navigate to the predefined training project: `Module 9 Making the Connection (Top\Projects\Altium Designer Essentials Training Course\...)`
5. Select **File » Open Project as Copy…** Open Project As Copy... .
6. At the new dialog *Create Project Copy,* as seen at **Error! Reference source not found.** .
   a) Add your name to the project: `Module 09 Making the Connection – [Your Name]`
   b) Add a description: `Altium Essential Training - Module 9 – [Your Name]`
   c) Open the *Advanced* section.

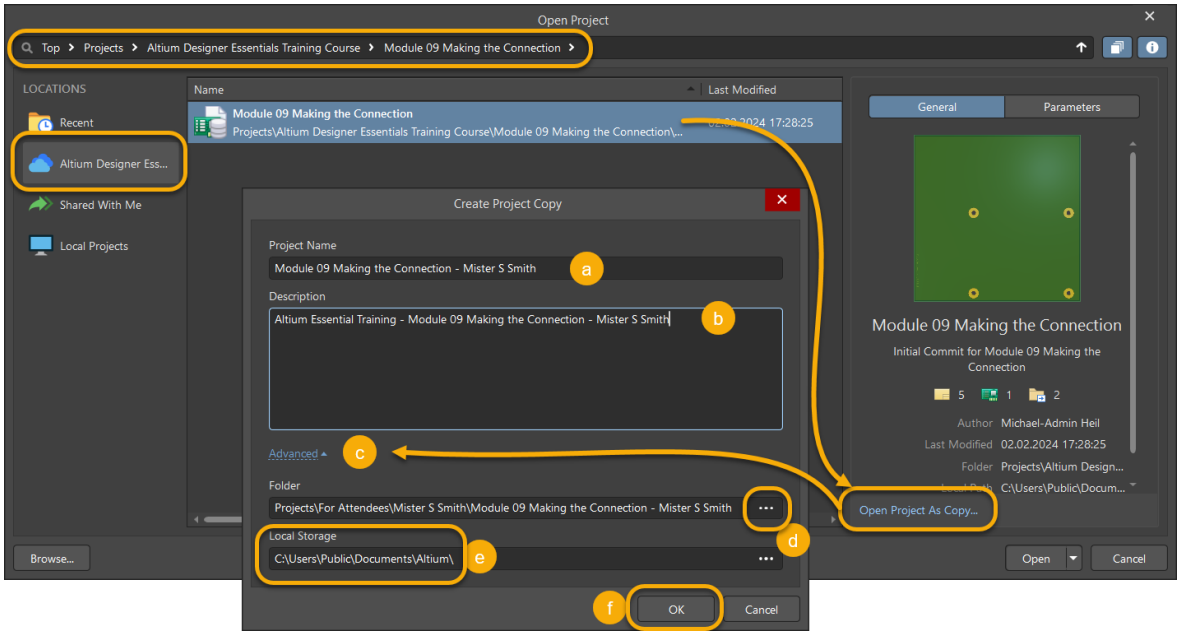


Figure 1. Copy existing Training Project

   d) Select the Ellipsis Button ..., as shown in Figure 1, to open the *Choose Folder* dialog; as shown in Figure 2
      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.

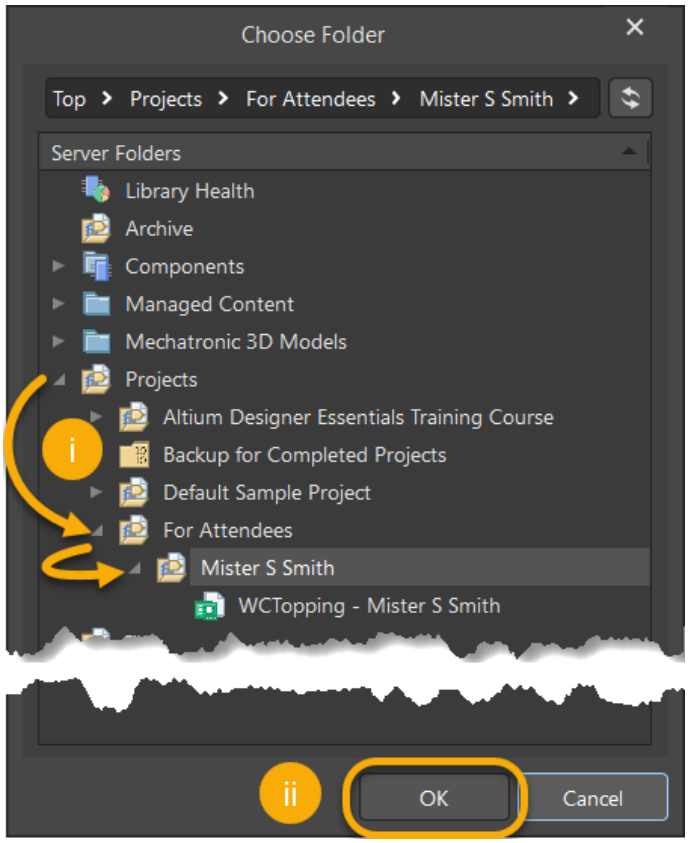


Figure 2. Step d - Choose Folder dialog

1.3.2.2 Option Two

8. Open the *Explorer* panel, **K » R**.
9. Navigate to the predefined training project `Module 9 Making the Connection (Top\Projects\Altium Designer Essentials Training Course\...)`
10. Select the Ellipsis Button ••• and the command **Clone** as seen at Figure 3.

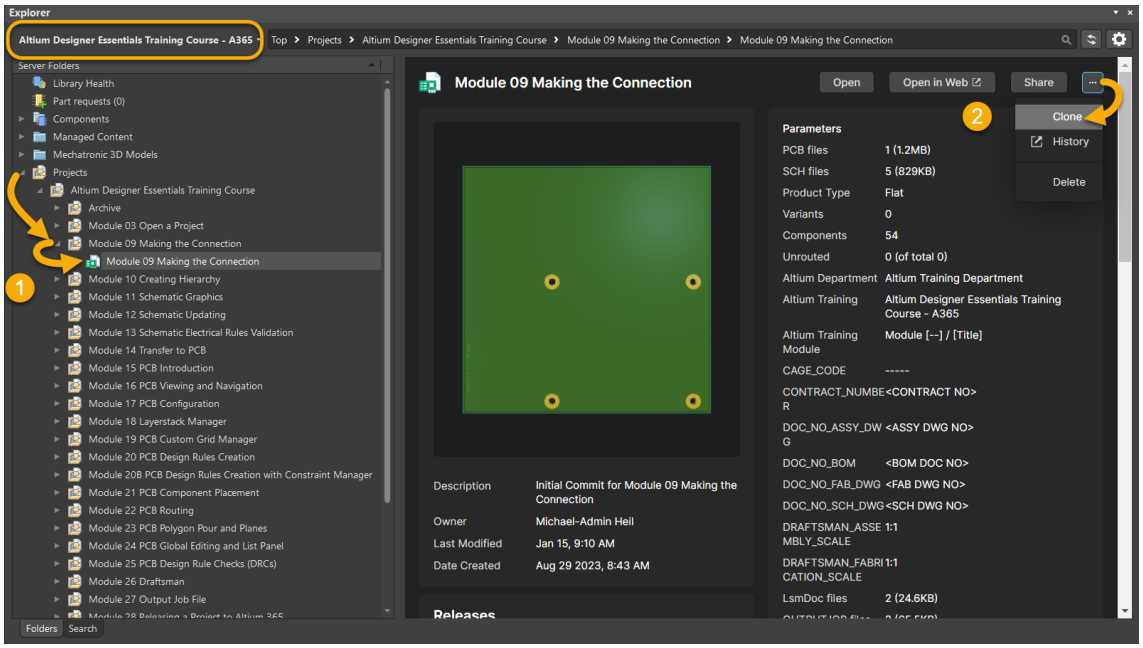


Figure 3. Copy Training Project

11. At the new dialog *Create Project Copy*, as seen at Figure 4.
    a) Change the project name to `Module 09 Making the Connection – [Your Name]`.
    b) Add a description: `Altium Essential Training - Module 9 – [Your Name].`
    c) Open the Advanced section if needed. Select the Ellipsis Button ••• from the **Folder** configuration to open the *Choose Folder* Dialog, Figure 5.
        i) Change the path to `Project\For Attendees\[Your Name]`
        ii) Select **OK**
    d) Change the *Local Storage* path if needed.
    e) Select **OK** to close the Dialog.
12. Please wait until Altium Designer creates a copy of the project and adds it to the Projects panel for you. This process may take up to 1 minute.

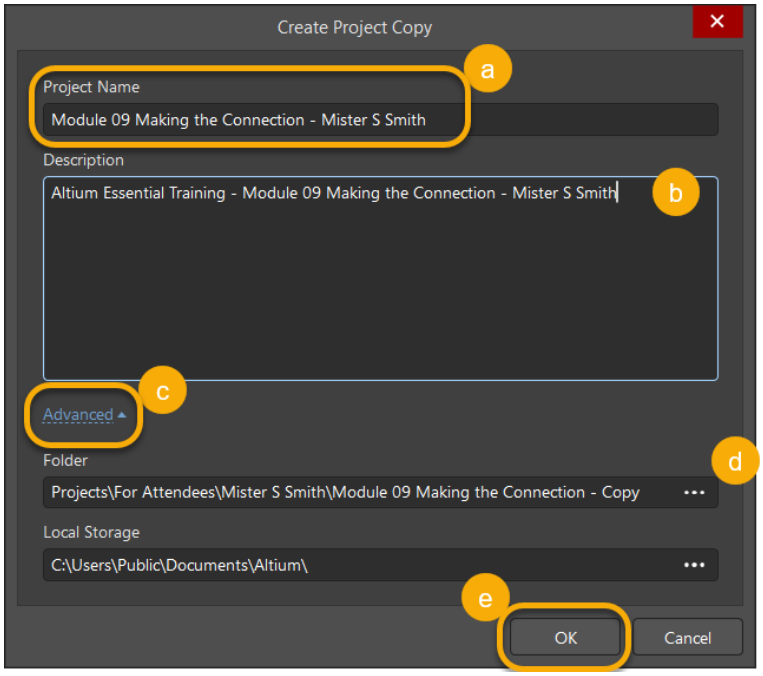


Figure 4. Create Project Copy

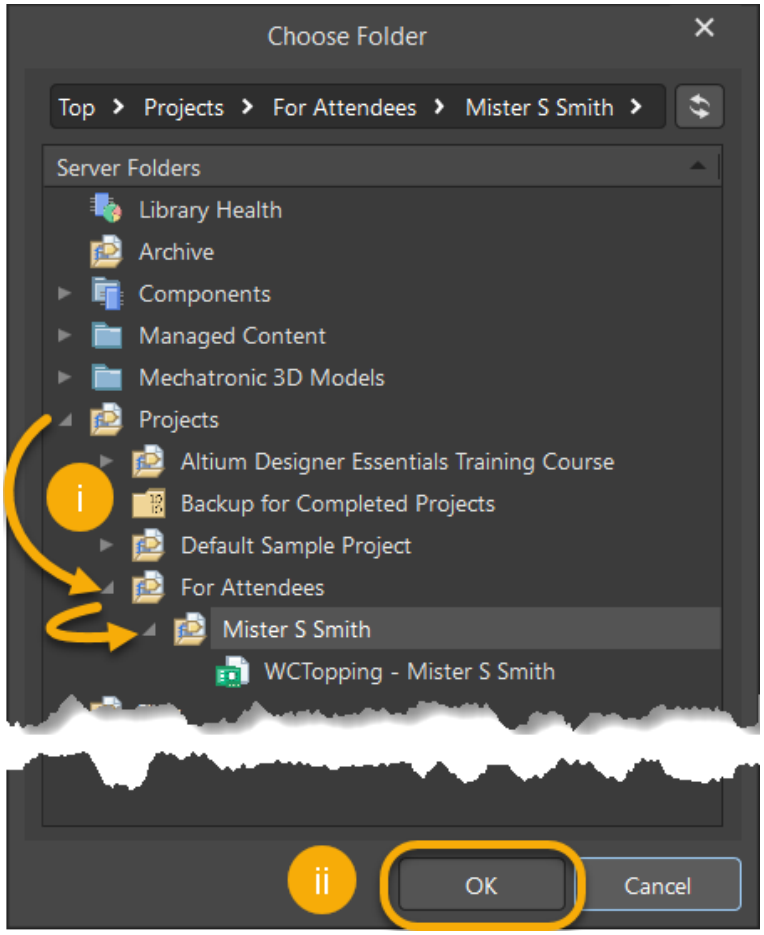


Figure 5. Last Image Step d - Choose Folder for Project Copy

1.3.2.3 New Project after Clone / Copy Process

13. As a result of the Copy/Clone process, you will now see a new project in your Project panel, similar to Figure 6.

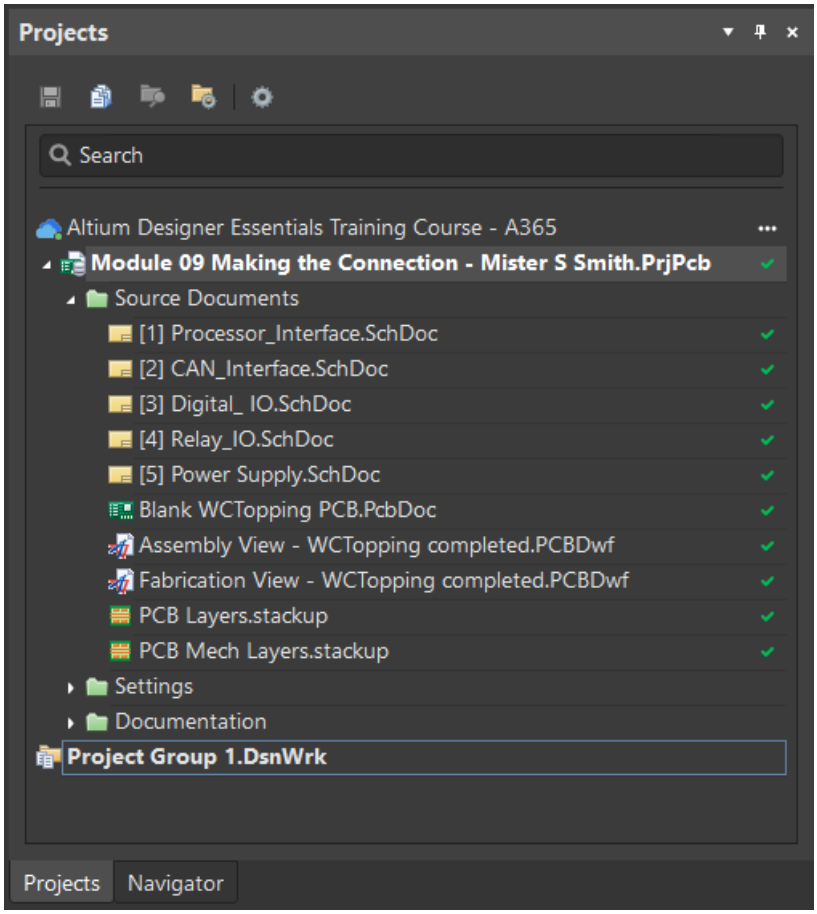


Figure 6. New Project based on the Predefined Training Project

1.4 Processor_Interface Schematic

1.4.1 Wire Connections

14. From the project `Module 09 Making the Connection - [Your Name]`, open the `Processor_Interface.SchDoc`. It contains a partially completed schematic as shown in Figure 7, in this exercise you will complete the remaining connections on this schematic by adding wires and net labels, as well as signal and power ports.

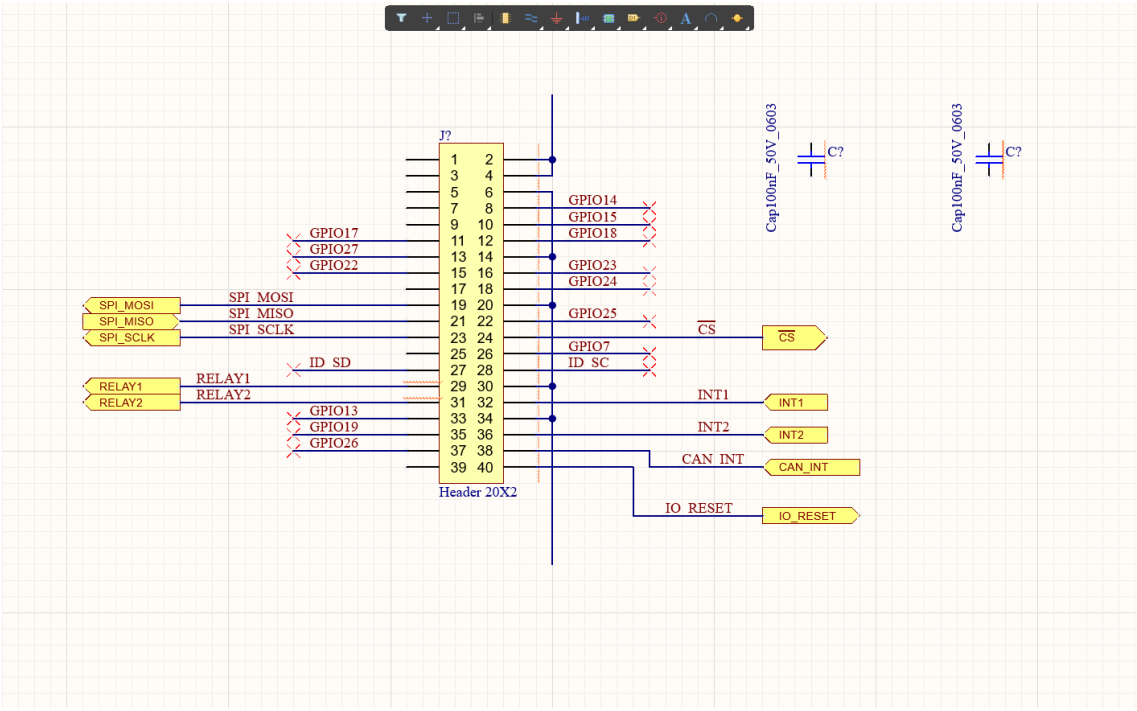


Figure 7. Processor Interface

The overscore, as shown on ports $\overline{CS}$ and IOexpand_$\overline{CS}$, can be created by entering a backslash "\" after the character to negate.

15. Let's start by connecting Pin 1 to Pin 17 of the header, and then place a power port. See *Figure* 8 as reference to how the completed schematic will look.

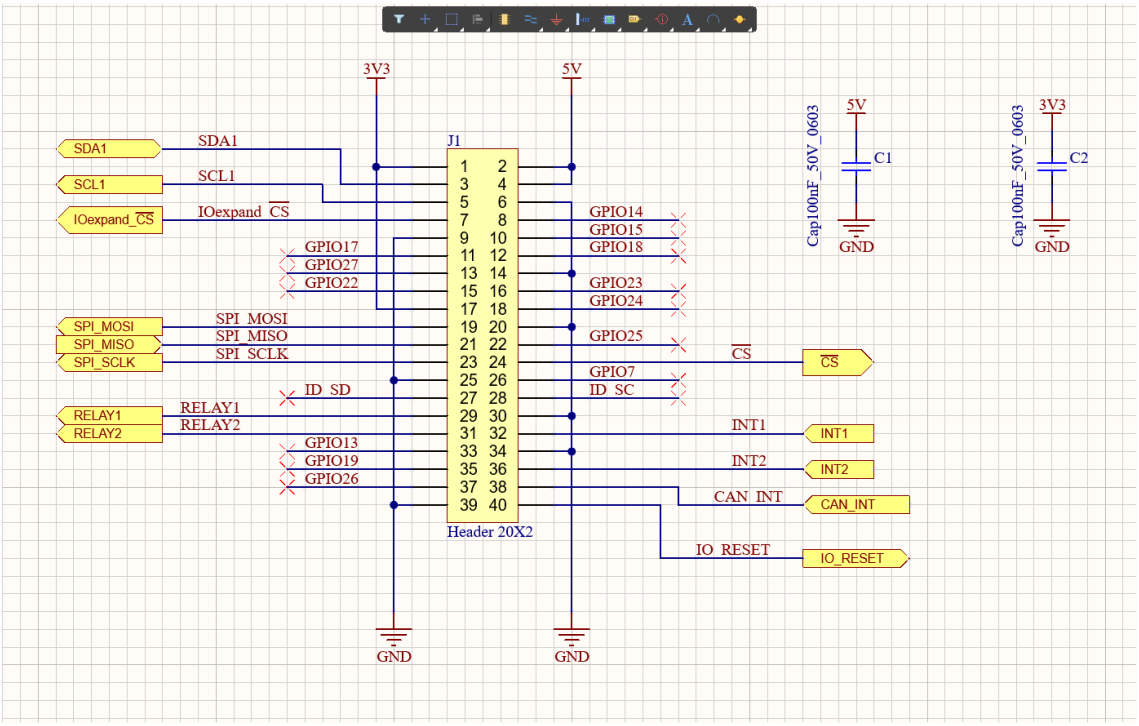


Figure 8. Completed Schematic as reference

a) Go to **Place » Wire** (shortcut **P » W**). This can also be done from the *ActiveBar* as shown Figure 9.

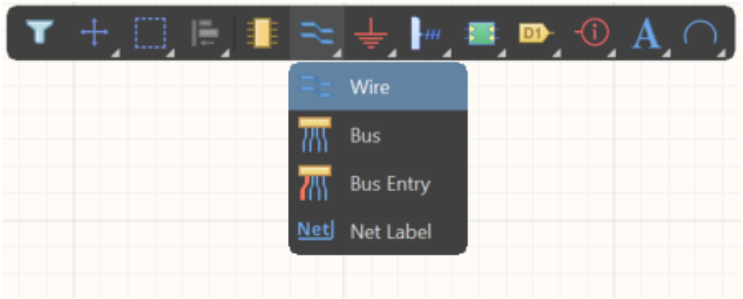


Figure 9. Place Wire from ActiveBar

b) Move the cursor to the tip of Pin 1 of component J4. You will see a cross appear, indicating you are on the pin's electrical hotspot as seen in Figure 10.

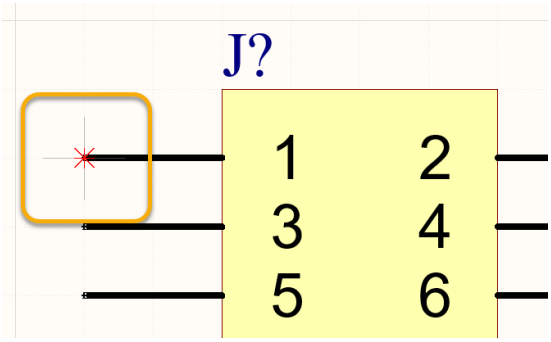


Figure 10. Electrical Hotspot of a pin for schematic connections

c) **Left-click** when the hotspot is active to begin placing a wire segment.

d) As you move the cursor away from the pin, you will notice the wire will be added to the pin. Press the **Spacebar** to flip the angle of the wire if needed.

e) A new segment will be added with each left-click until another electrical hotspot is selected (such as another pin) to complete the placement of this first wire.

f) Complete the connection from Pin 1 to Pin 17.

g) Place a small segment above pin as shown in *Figure 11*, to which you will add a power port later in this exercise.

5. During placement of a segment, **right-click** once to stop the current segment but remain in placement mode. If you've completed the wire connection, you will not need to right-click to cancel the command.
6. Using Figure 11 as reference, connect Pins 9, 25 and 39 together using the method described above.

Use Spacebar to change the angle of a wire corner during placement.
Use Shift+Spacebar to change the corner mode of a wire during placement.
Use Backspace to unwind an inadvertently placed segment during placement**.**
Use Right-click or **Escape** to end placement mode.

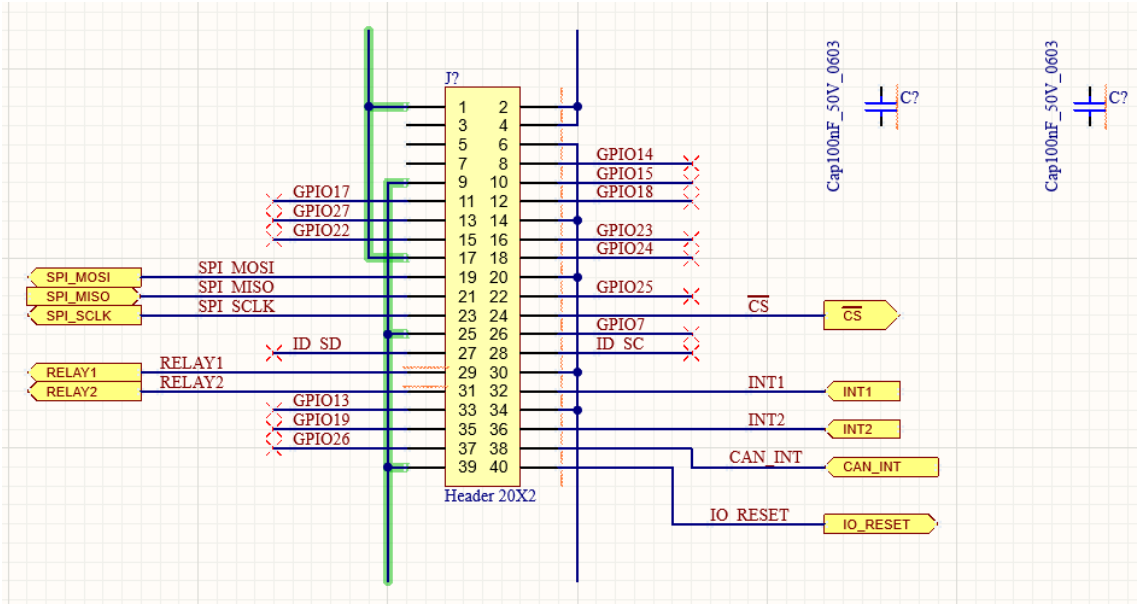


Figure 11. Wiring for component J4

1.4.2 **Power Ports**

7. Let's place a **GND** port from the **Active Bar** . (or from the **Place » Power Port** menu). You may need to Right Mouse Click on the Active Bar if the **GND** port is not visible.

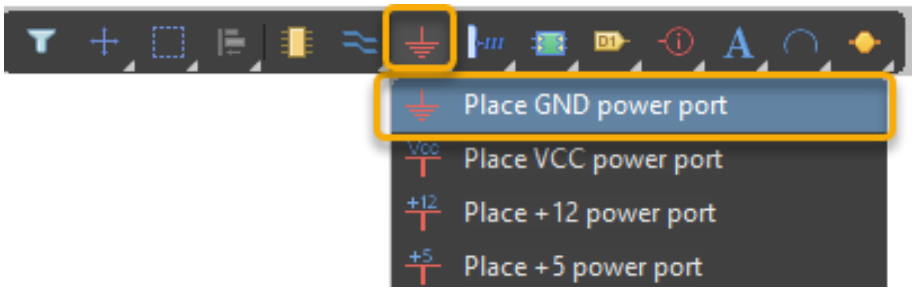


Figure 12. Active Bar GND Port

8. Place **GND** ports on the connections on `J4`, and directly on the pins of the two capacitors, using Figure 13 as a reference (Power Ports can be rotated by pressing the **Spacebar** while moving them).
9. When finished press **ESC** or Right mouse click to exit placement.

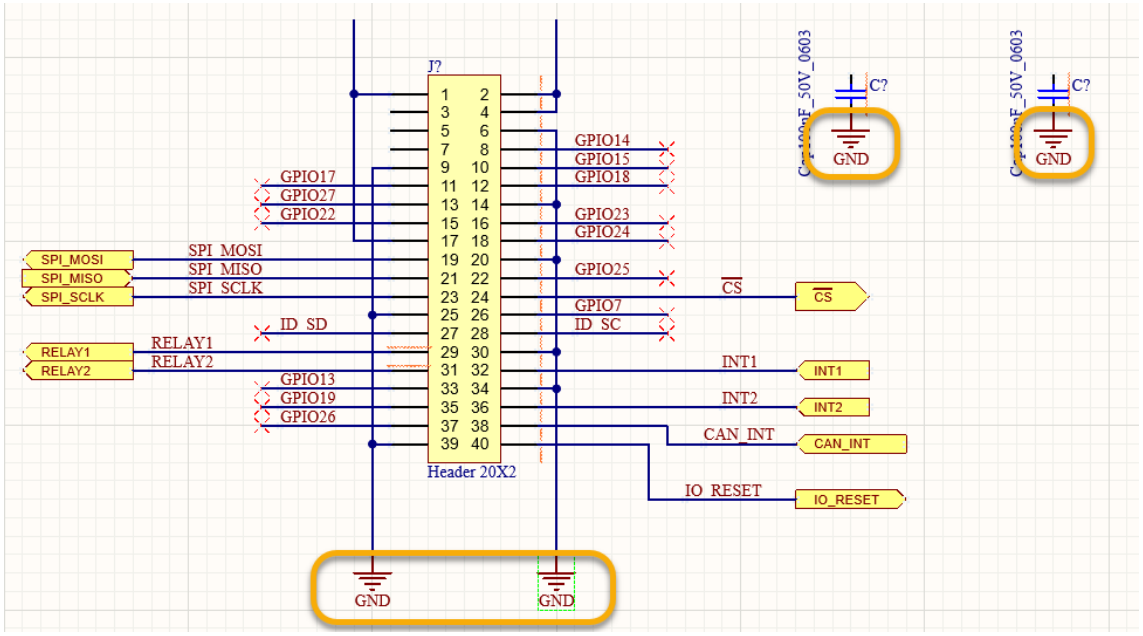


Figure 13. Power Ground ports below Header J4

10. Next place the remaining power ports, again using the **Active Bar** (or from the **Place » Power Port** menu). You may need to Right Mouse Click on the Active Bar to select the appropriate power port, as shown in Figure 14.

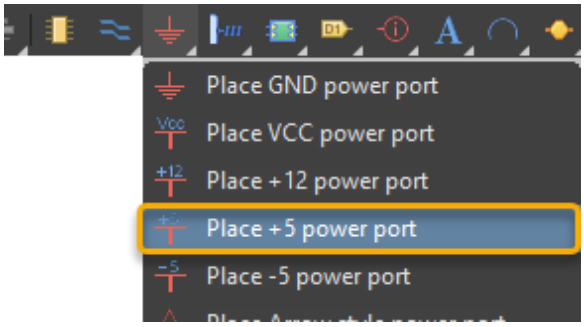


Figure 14. Power Port

11. With the Power Port on your cursor, hit the **TAB** key to change the properties, as shown in Figure 15.

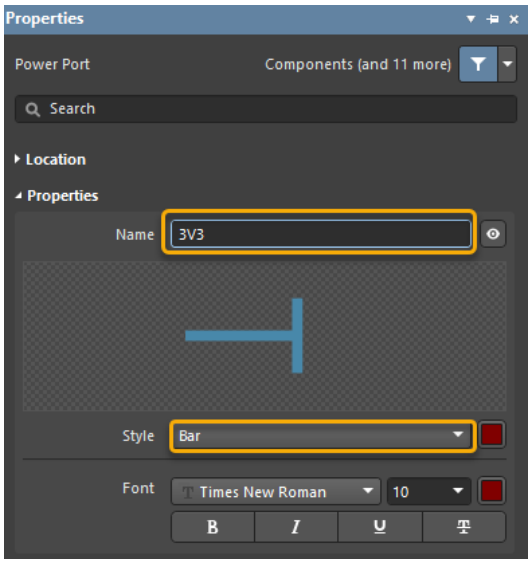


Figure 15. Power port

a) Change the **Name** to `3V3` and ensure the **Style** is set to **Bar**.

b) Press **Enter** or click on the Pause symbol to continue the placement of the Power Port.

c) Power Ports can be rotated by using the **Spacebar** while moving them.

d) Place the Power Port **3V3** on the connection to the connection above Pin 1, and directly onto one of the capacitors.

e) Repeat the process above and place the Power Port **5V** on the connections on **Pin 2** and **Pin 4,** and the remaining capacitor. When complete, the schematic should look similar to Figure 16.

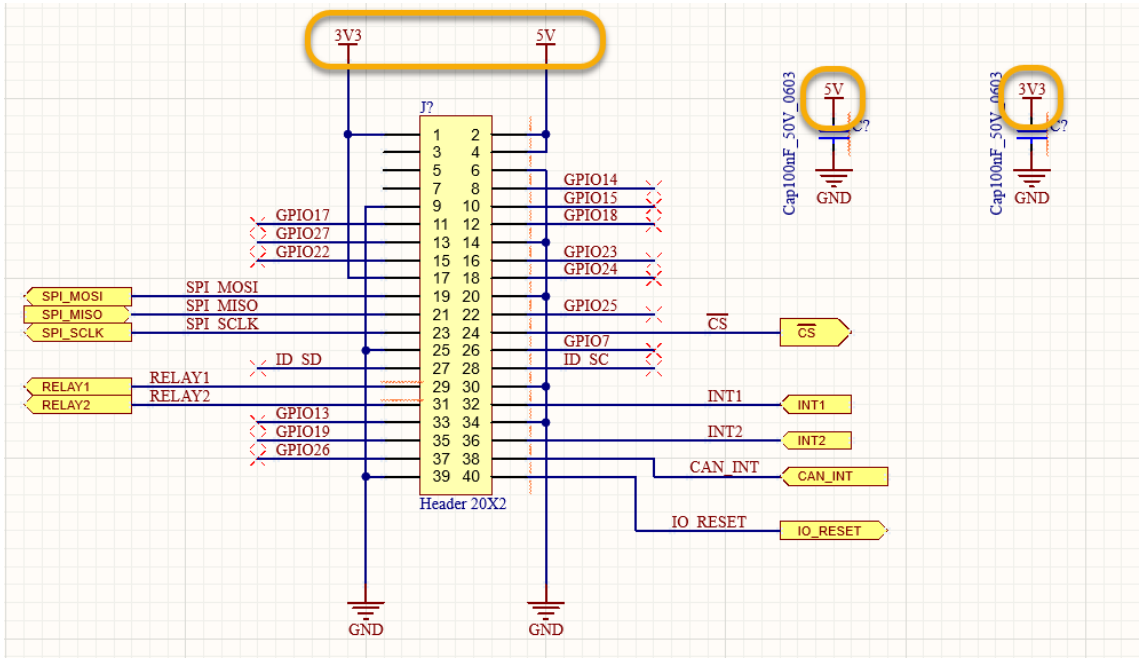


Figure 16. Header J4 with Power Ports

1.4.3 **Net Labels**

12. In the next steps, using *Figure 17* as reference, we will describe how to place **Net Labels** and **Signal Ports**.

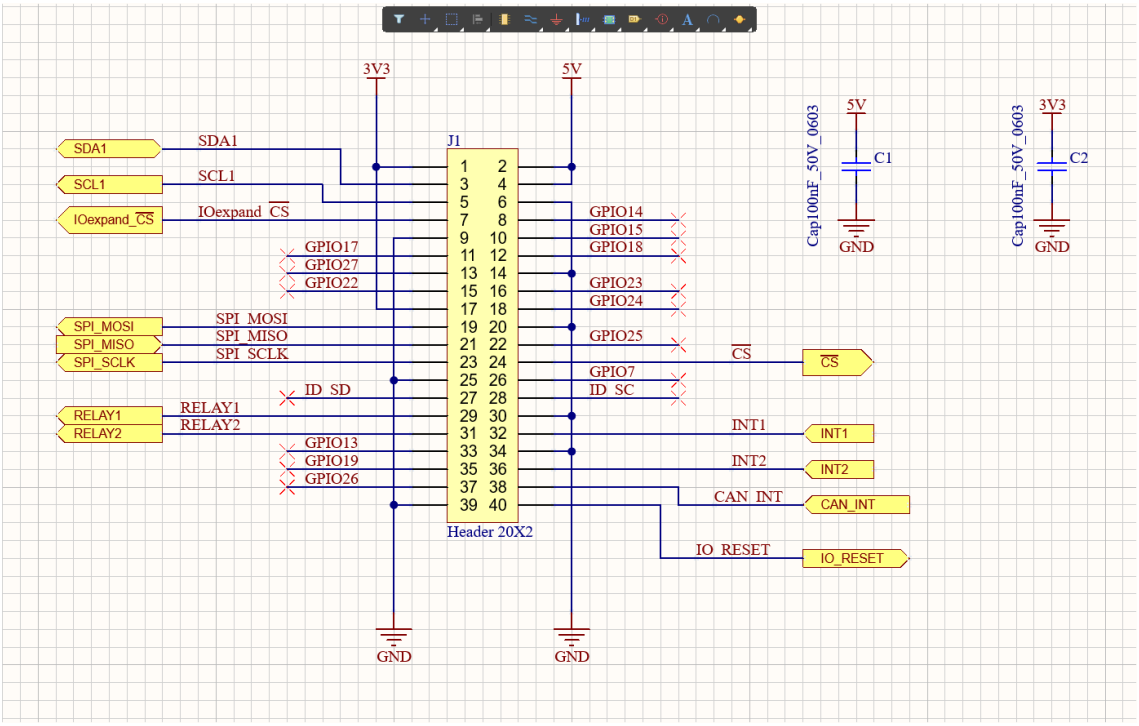


Figure 17. Completed Schematic wiring for Processor_Interface

13. Using the **Active Bar,** first place wires on J4 pins 3, 5 and 7, as shown in

14. *Figure* 18

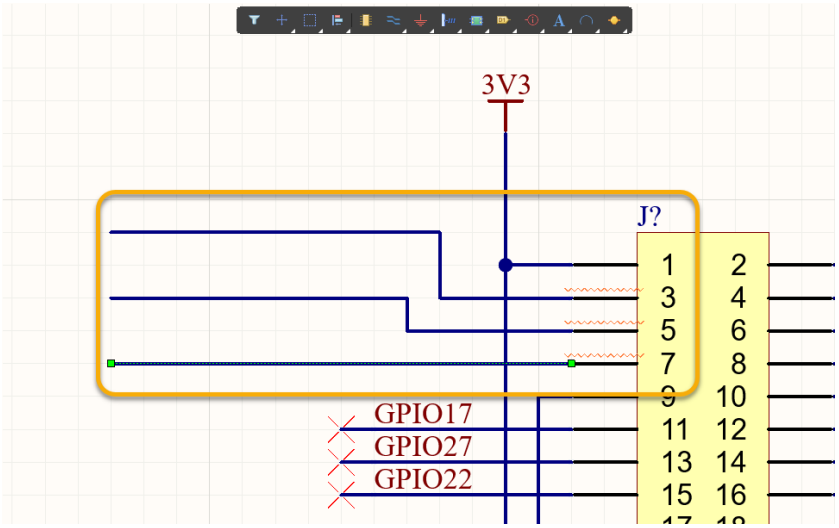


Figure 18. Place Wires

15. Now place **Net Labels** using the **Active Bar** (or by going to **Place » Net Label** or use the shortcuts **P » N**), see *Figure* 19.

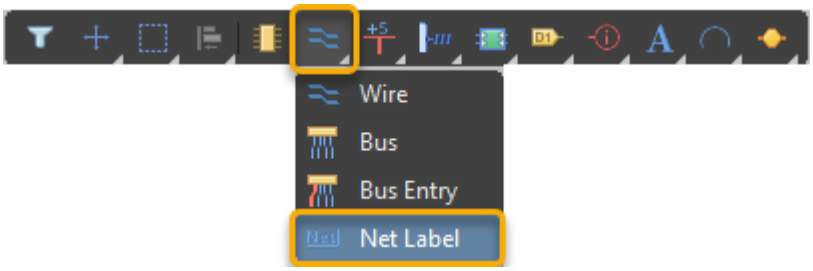


Figure 19. Net Label

16. With the **Place Net Label** command active, press the **TAB** key to open the *Properties* panel, to enter or change the name of the port.
17. Enter **Net Name** `SDA1` as shown in Figure 20.

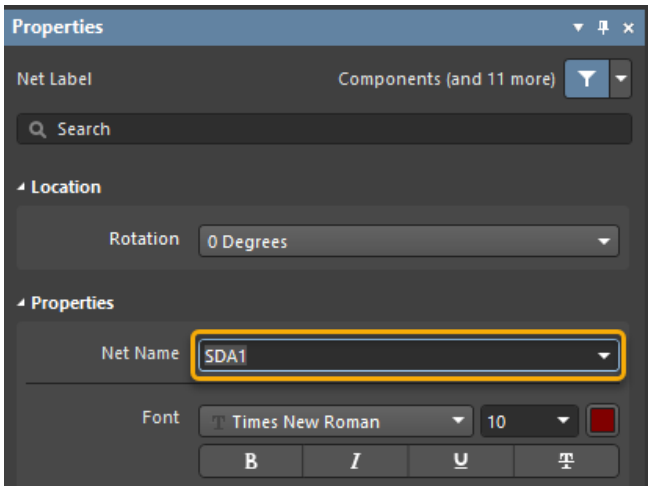


Figure 20. SDA1 Net Label Properties

18. Press **Enter** and place the Net Label on the wire on Pin 3 of the header using Figure 21 as reference.

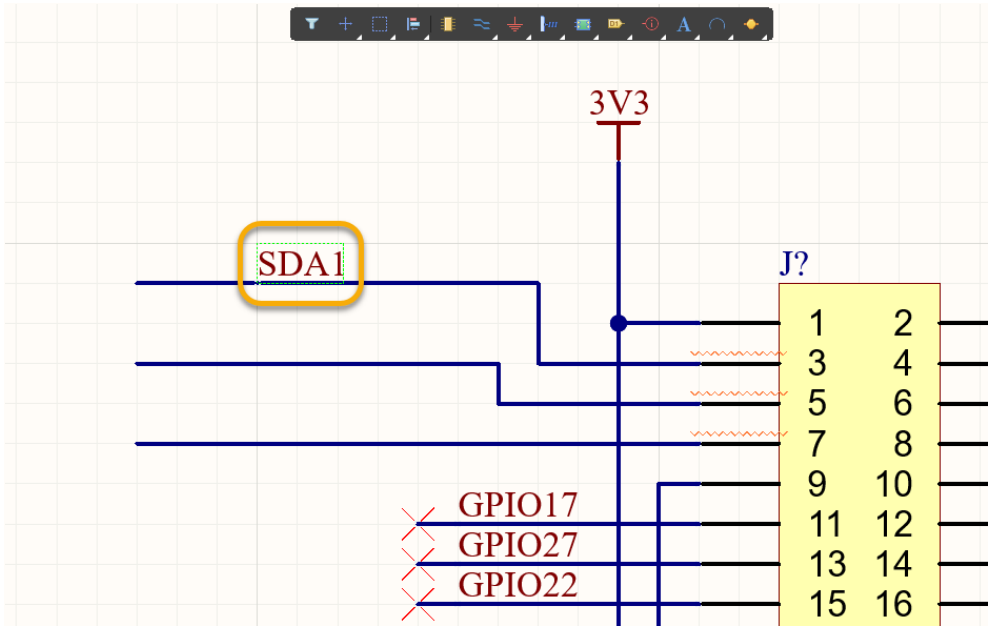


Figure 21. Placing the SDA1 Net Label onto Pin 3

19. Continue to place the remaining *Net Labels* using Figure *22* as reference. Notice the numeric values will automatically increment if the *Name* is the same. The properties of a *Net Label* can always be changed by hitting the **TAB** key during placement, or by selecting the *Net Label* and changing its name manually in the *Properties* panel.
20. To place a negation overbar on a N*et Label*, place a \ (backslash) after the character requiring the overbar, in this case, **IOexpand_C\S**.

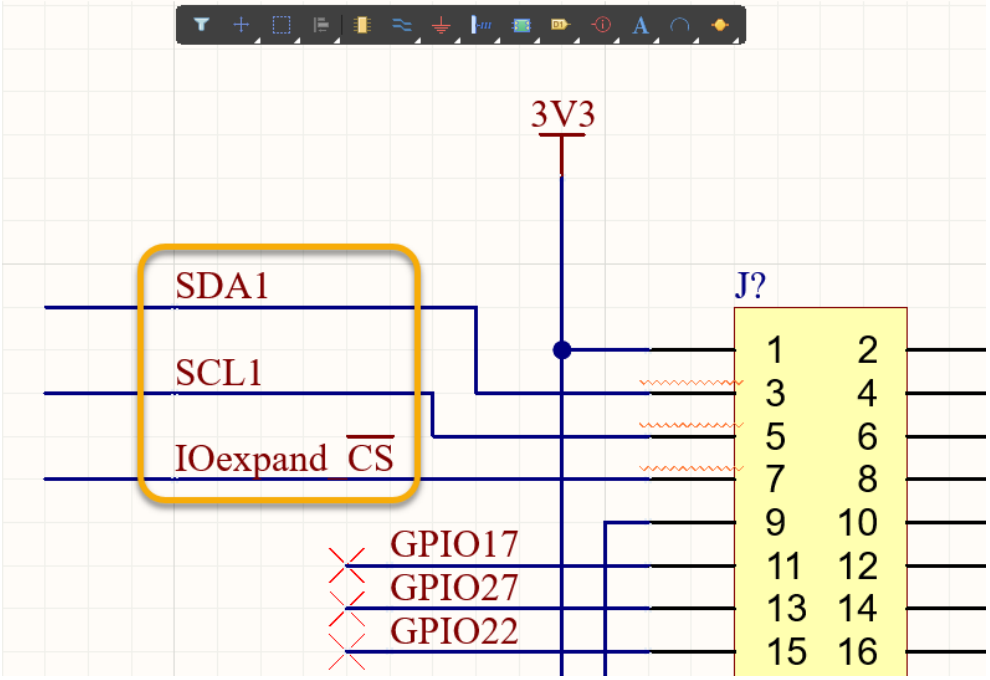


Figure 22. Net Label Placement

Some Net Labels contain an **Underscore** that could be mistaken as a space, and not clearly seen if placed on a Wire. These Net Labels include the **SPI_xx**, **ID_xx** and **IO_RESET** group of Net Labels. This naming syntax is necessary for proper net connectivity.

Moving Net Labels after they are placed on wires can be difficult because moving the Net Label also drags the attached wire. Selecting the Net Labels, then going to **Edit » Move » Move Selection** can be helpful when moving Net Labels. Also, if **Always Drag** is enabled in Preferences, you can use the **Ctrl+Left mouse button** combo to drag a net label off of a wire.

1.4.4 **Signal Ports**

21. Using the **Active Bar**, see *Figure* 23**;** place the three remaining signal ports. Ports enable connections across project schematic sheets in a design. Ports can also be placed using the **Place » Port** command as well as the **P»R** shortcut keys

Figure 23. Active Bar Signal Port

22. Press the **TAB** key with the port placement active at the curser and change the name of the port to `IOexpand_`$\overline{CS}$ in the *Properties* panel. As a reminder, to place a negation overbar on a N*et Label*, place a \ (backslash) after the character requiring the overbar, in this case, **IOexpand_C\S**.
23. Set the **I/O Type** to *Output* as shown in Figure 24.

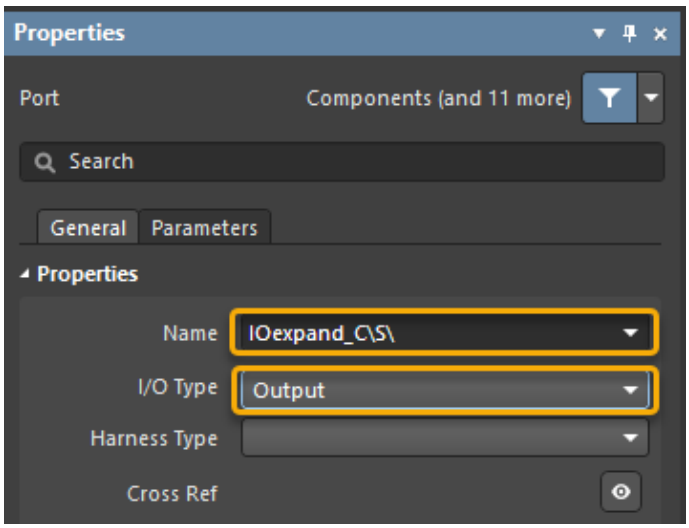


Figure 24. Port Properties dialog

24. Press **Enter** or click on the Pause symbol to the continue placement of the port.
25. Place the **IOexpand_**$\overline{CS}$ port near the wire for net label **GPIO4** on **Pin 7** of the header as shown in *Figure* 25

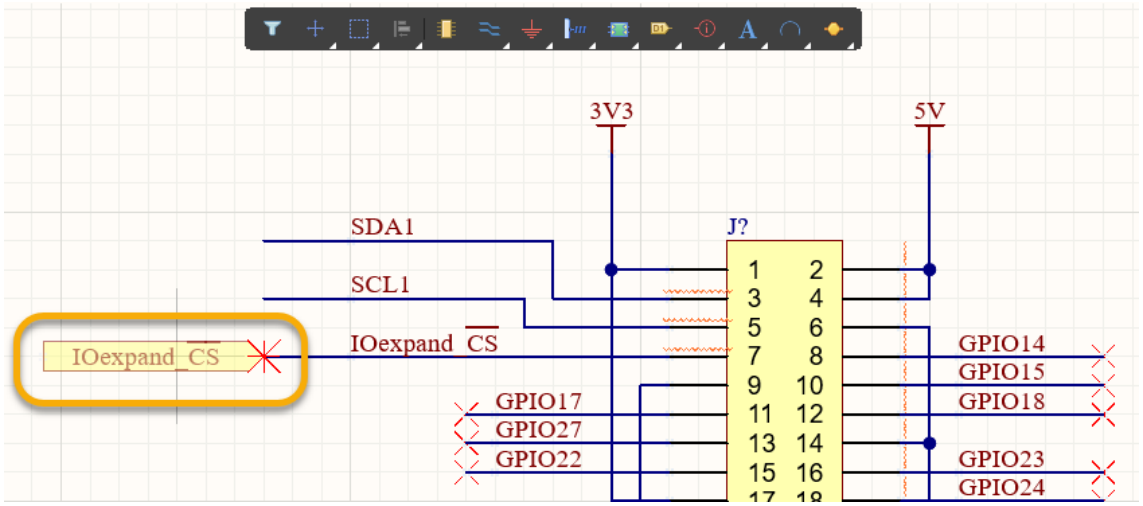


*Figure 25. Signal Port IOexpand_*$\overline{CS}$

26. The port will require two **left-clicks**. The first left-click defines the left side of the port, and the second left-click defines the right-side of the port and its length. After placement, change the height of the port to 150mil, so that the bar above the text is not obscured by the port. See *Figure* 26

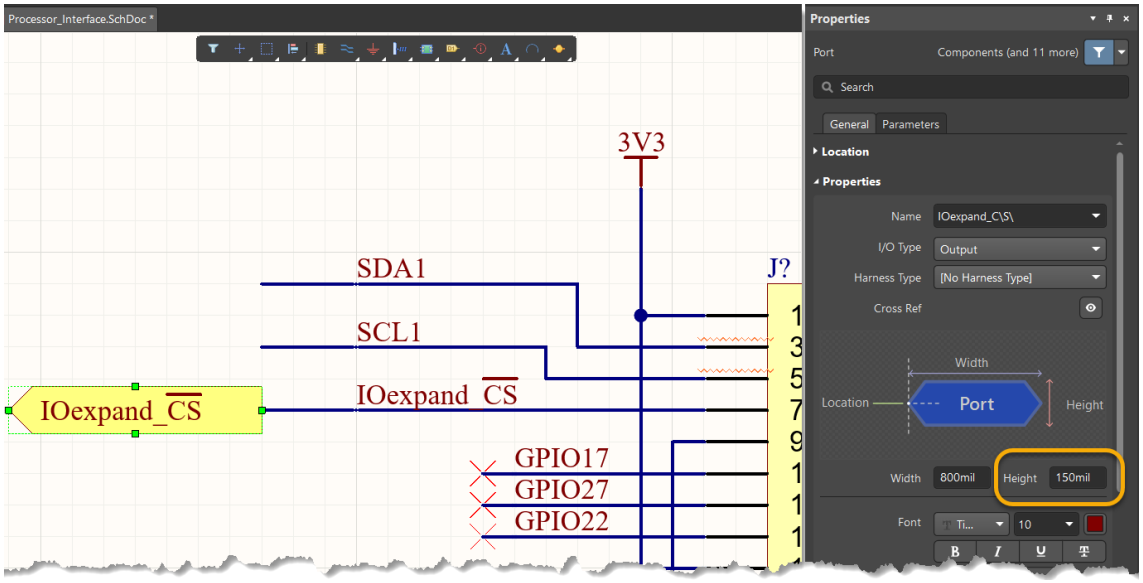


Figure 26. Port Properties

27. Continue to place the remaining two ports as shown in *Figure* 27. Port **SDA1** should be set as **Bidirectional**, and port **SCL1** should be set as **Output**. Ports will automatically show their I/O type direction (such as Input, Output) once the placement is complete. The best way to place ports, is to set the I/O Type in the Properties panel during initial placement, although it is still possible to change port properties after placement.

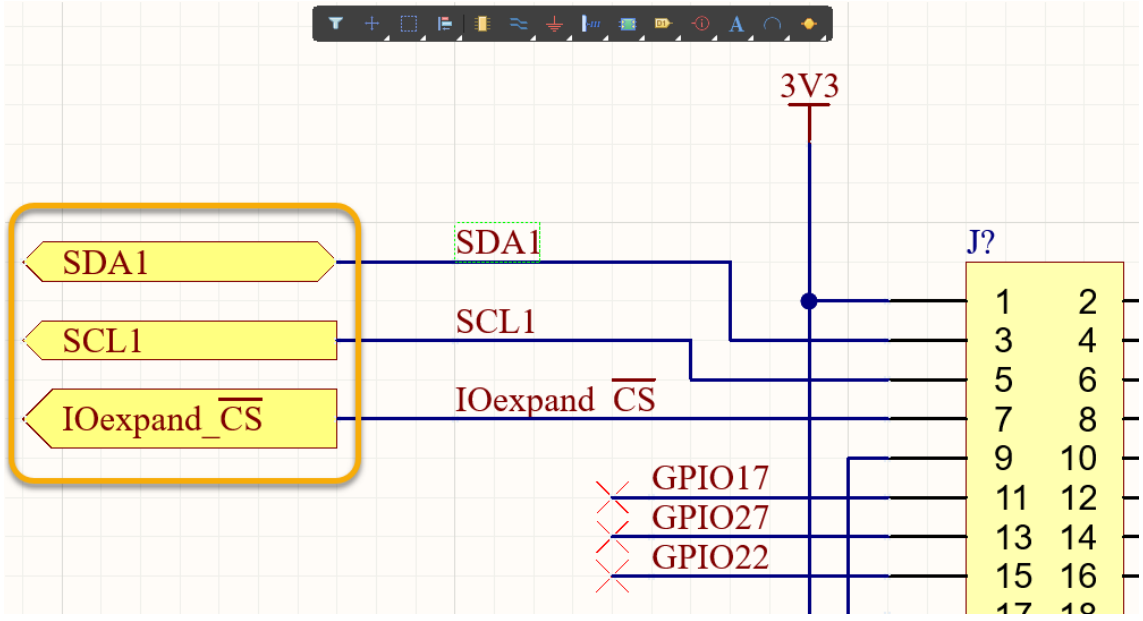


Figure 27. Signal Ports

28. **Save** the schematic changes made so far.

1.5 CAN_Interface Schematic

1.5.1 Net Labels

29. Open the `CAN_Interface.SchDoc` from your *Projects* panel.
30. Go to **Place » Wire** and draw a short wire segment from R2 to the left, as shown in Figure 28 below.
31. Go to **Place » Net Label** and press the **TAB** key to set the **Net Name** to `CAN_RXD`.
32. Hit **Enter** to place the *Net Label* between the two resistors attached to `UC2-4` (`Pin 4` of `UC2`) as shown in Figure 28 below.

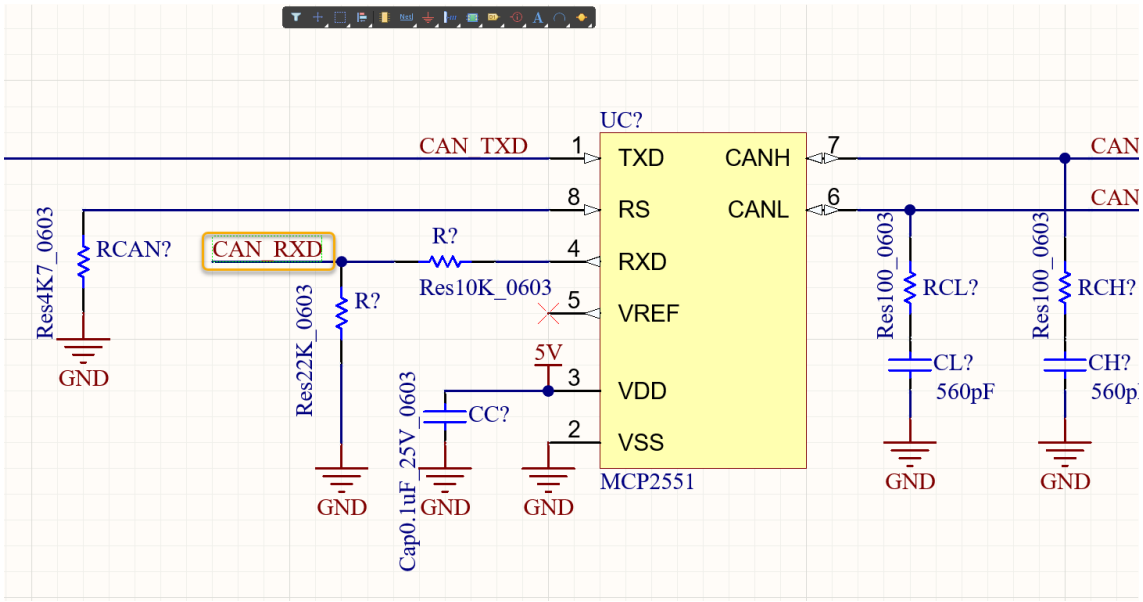


Figure 28. CAN_RXD Net Label placement

33. Place another instance of the `CAN_RXD` *Net Label* on the wire attached to `UC1-2` to create a logical connection between `CAN_RXD`, as shown in Figure 29.

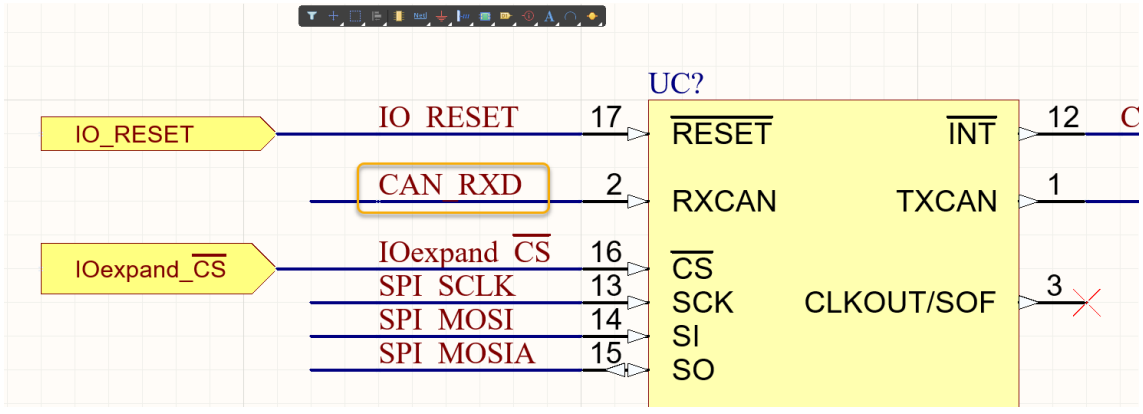


Figure 29. CAN_RXD Net Label placement

34. To see the logical connection press and hold **ALT** key and select net label CAN_RXD with mouse left click, this highlights the net labels showing connections within the schematic, as shown Figure 30.

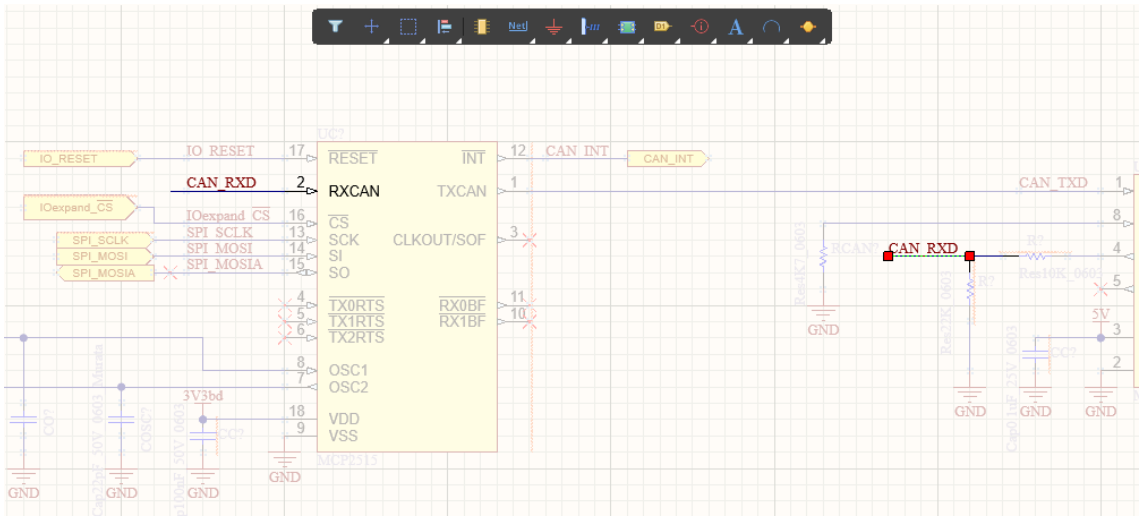


Figure 30. Active Masking to see a logical connection

35. Use **Shift+C** to clear the masking.

1.5.2 Signal Ports

36. Place ports using methods described earlier, on `CAN_RXD`, `SPI_SCLK`, `SPI_MOSI` and `SPI_MOSIA` wires as shown in Figure 31 below.

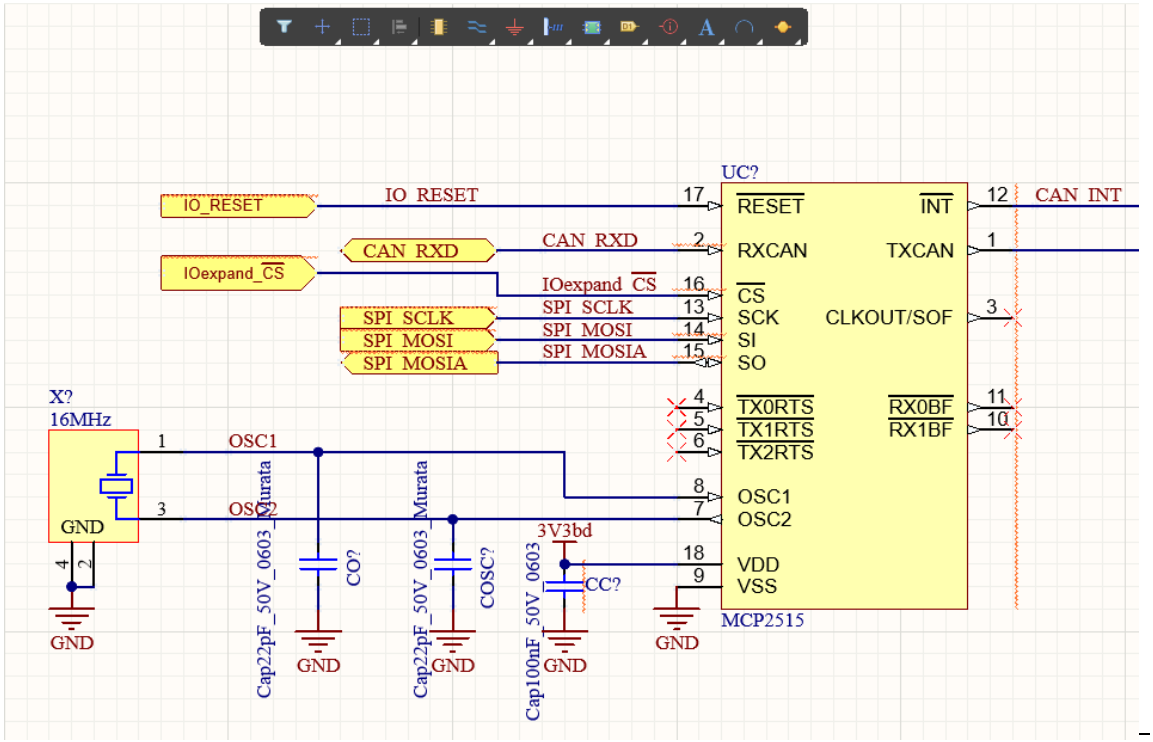


Figure 31. Finalized section of the CAN_Interface schematic

1.6 Digital_IO Schematic

1.6.1 Auto-Generated Wires

37. Open the `Digital_IO.SchDoc` schematic from the *Projects* panel.
38. Select the first `Header8`.
39. Hold the left-click and drag the header over to MCP23S17 that pins of the `IOS?` header bump up against Pins 21-28 of MCP23S17 as shown in *Figure* 32 below.
40. Let go of the left-click so that the component is set in place.

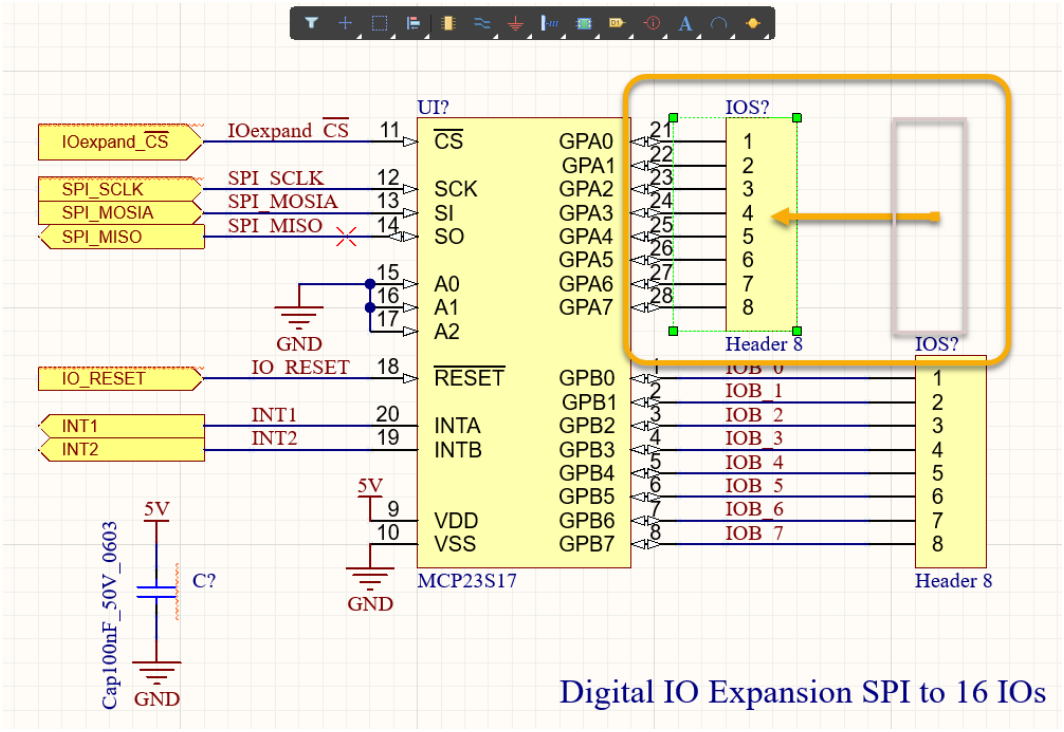


Figure 32. Connecting pins for MCP23S17 and first Header 8

41. Drag `Header 8` away from MCP23S17 to create wire connections between the pins automatically, as shown in Figure 33 below.

If **Always Drag** is not enabled in Schematic Preferences, holding the **CTRL** key will be required for automatically creating connections.

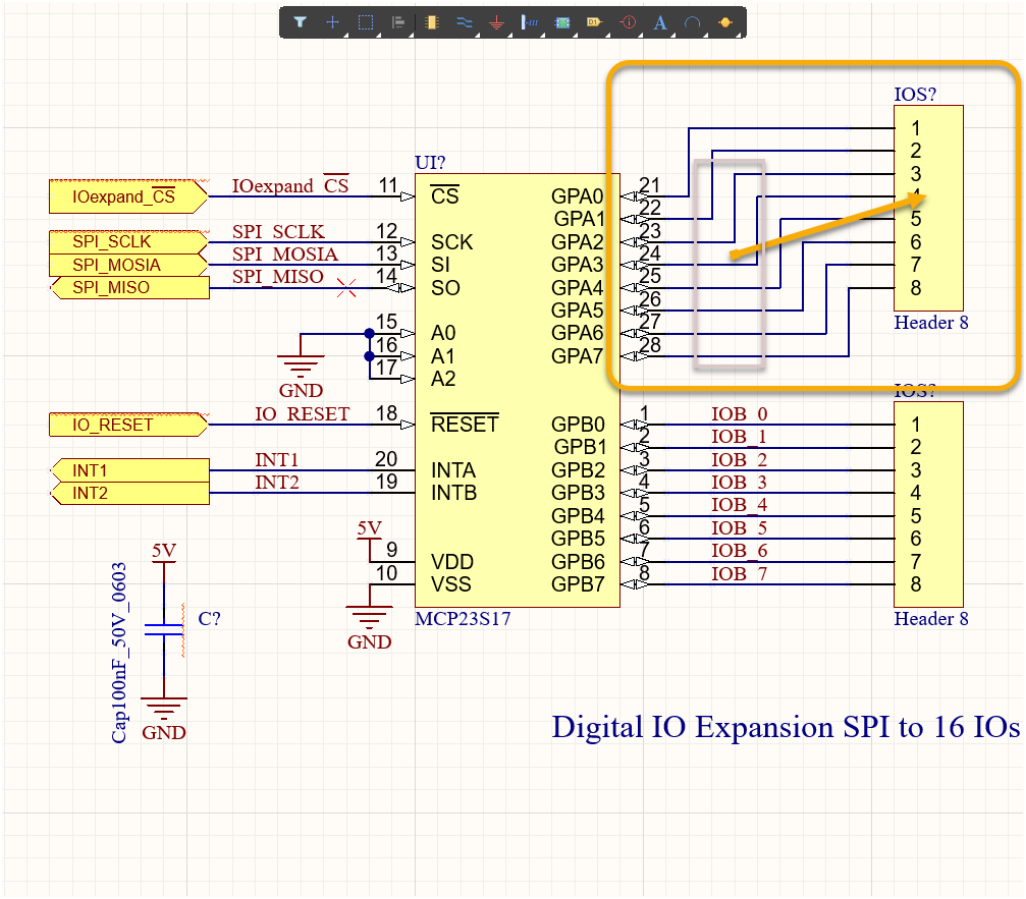


Figure 33. Auto-Generated wires when pulling the header away

42. Go to **Place » Net Label** and press the **TAB** key to set the **Net Name** to `IOA_0`
43. Place the netlabel on to the first wire, MCP25S17 Pin21 – Header 8 Pin1.
44. Continue placing the netlabel, automatically Altium increment the last number `IOA_1, IOA_2,…n.`

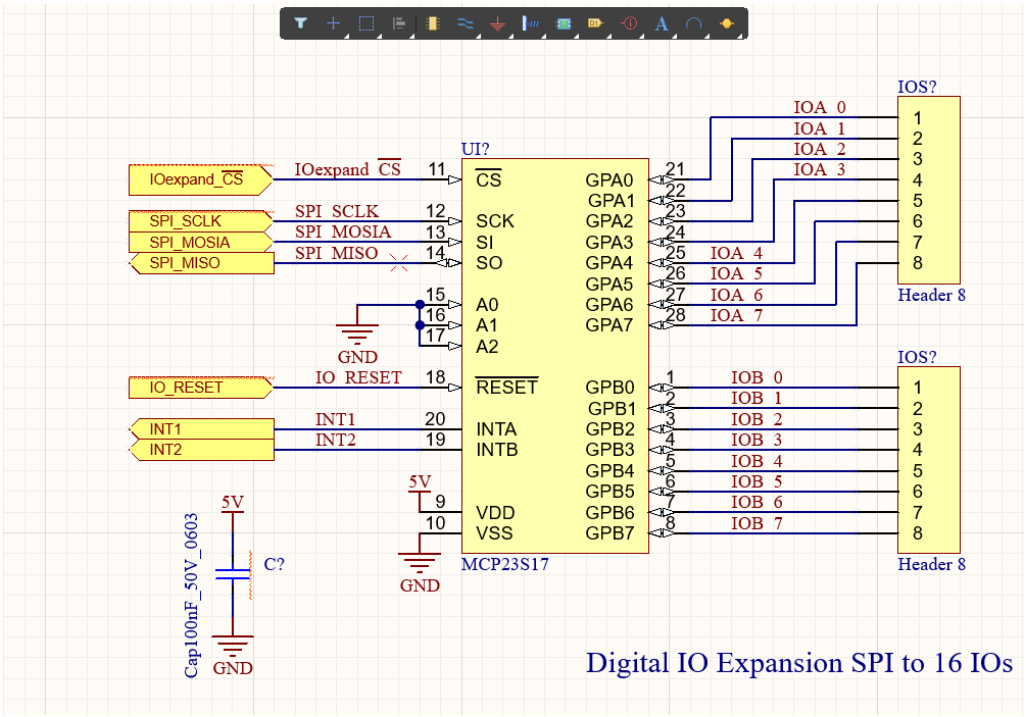


Figure 34. Completed Connections

1.7 Create a Bus Connection

45. Open the `Relay_IO.SchDoc` schematic from the *Projects* panel.
46. Create a BUS connection for the Signals Relay1 and Relay2, use Figure 35 as reference.

The bus naming follows the Syntax **Name[LSB..MSB]** e.g. Data[0..7].
It is also possible to use the Syntax **Name[MSB..LSB]** e.g. Data[7..0].

47. Use the following menu commands to create the bus structure. Feel free to use the Toolbar commands or shortcut keys instead.
    a) **Place » Wire**
    b) **Place » BUS**
    c) **Place » Netlabel:** *Relay[1..2]*
    d) **Place » Port:** *Relay[1..2]*

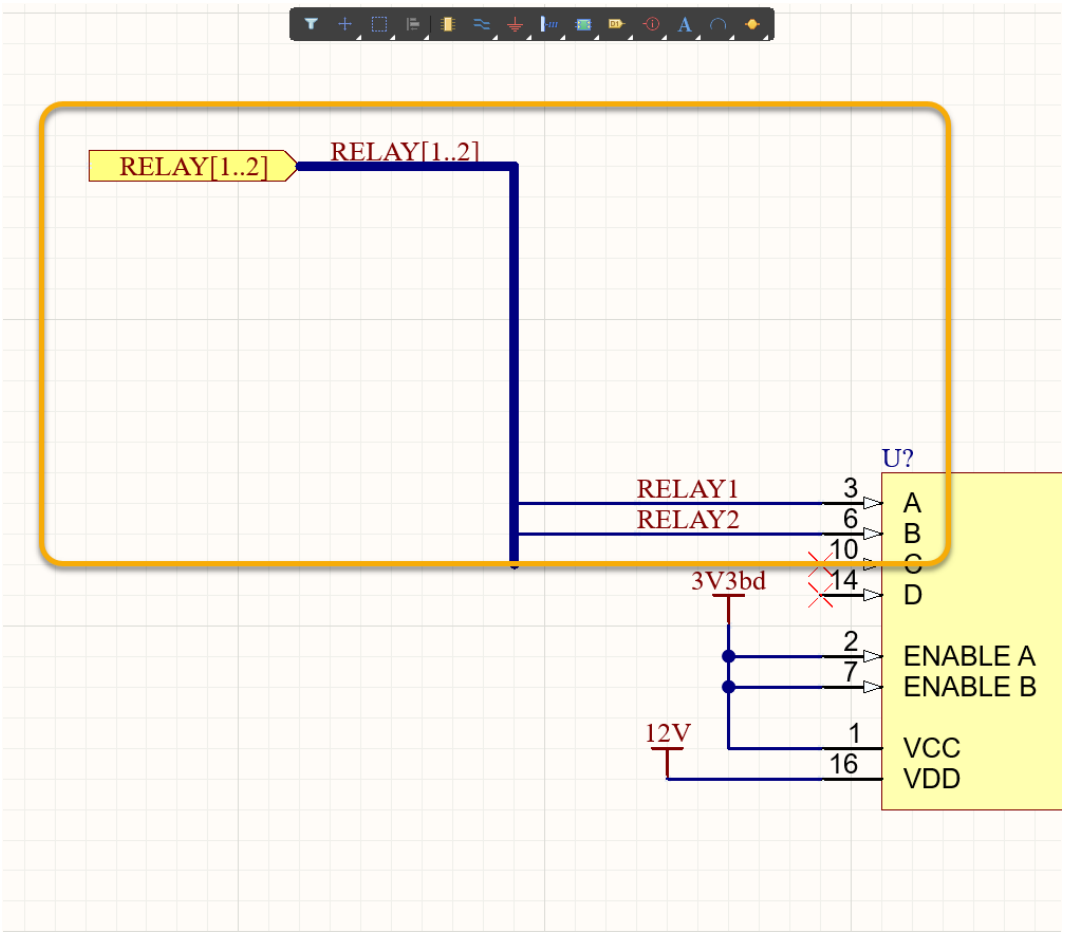
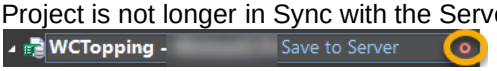


Figure 35. Relay_IO with BUS

1.8 Save Modifications for Project

Now that you updated the predefined training project you will save the modifications, and update the Workspace.

48. After we modified the Project, the Project is not longer in Sync with the Server, shown with a red icon next to the project name 

49. Select **File » Save ALL** from the main menu to save all modifications to your local hard disk.
50. At the *Project* panel, next to the project name you find the command **Save to Server** 
.
51. Select **Save to Server.**
52. At the dialog *Save [Project Name],* as seen at Figure 36
    a) Activate the checkboxes for the files that are not under version control, skip the file `*.PrjPcbStructure`.
    b) Add the comment `Module 9: Making the Connections - [Add Your Name] finished.`
    c) Click on **OK,**
    d) **Wait until Altium Designer finished the Commit and Save to Server.**

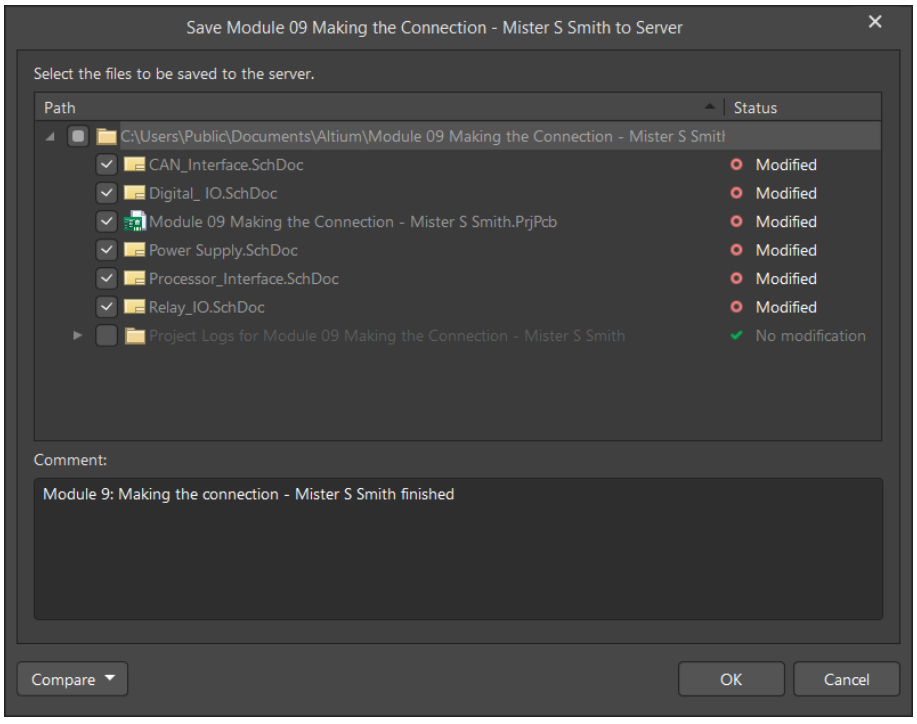


Figure 36. Save to Server

53. The red circle next to the file names will be replaced with a green check mark to indicate the files are up to date.
54. **If you are ready, close the project and any open documents (Windows » Close All).**

Congratulations on completing the Module!

Module 9: Making the Connections

from the

Altium Designer Essential Course
with Altium 365

Thank you for choosing Altium Designer