Altium Designer

Essentials Course - Altium 365

Module 8: Open an Existing Project and Schematic Capture

Software, documentation and related materials:

Table of Contents

Module 8: Open an Existing Project and Schematic Capture

1.1 Purpose

In this module, you'll learn how to open a project from the Altium 365 Workspace. You learn to add components in the schematic editor, along with learning the best practices of schematic component placement. You will also learn how to place predefined sections of Circuits – Reuse Blocks.

In this exercise, you will continue to use the project WCTopping you created in the workspace in the previous module. In this module you will learn how to place to components and continue to capture the schematic design intent. The Project you will be creating during the training is a daughter board, providing an expansion of a typical controller board via a 40-pin connector.

1.2 Shortcuts

Shortcuts when working with Module 8: Open an Existing Project and Schematic Capture

F1:	Help
P » P:	Place Part - Components Panel
SPACE:	Rotate Counter Clockwise
Shift+SPACE:	Rotate Clockwise
X:	Flip X-Axis Reference
Y:	Flip Y-Axis Reference
G:	Grid
CTRL+S:	Save Document

1.3 Preparation

1.3.1 General

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

1.3.2 Load an existing Project

Below are three methods for opening projects, please select one of the options to access your project. For more information on options two and three, refer to Module 3.

1.3.2.1 Option One

2. Open the list of recently used projects, **File » Resent Projects**, and open `WCTopping – [Your Name]`.

1.3.2.2 Option Two

3. Open the *Open Project* dialog with **File » Open Project**…
   a) Select **Workspace** `Altium Designer Essentials Training Course - A365` on the left side**.**
   b) Browse to the `WCTopping – [Your Name]` project, located at the folder `Projects » For Attendees » [Your Folder]`.
   c) Use **Open** to open the `WCTopping – [Your Name]`.

1.3.2.3 Option Three

4. Open the *Explorer* panel with **K » R,** with the Panels `Panels` button, or any other way you prefer.
   a) Navigate to the section `Projects » For Attendees » [Your Folder ] » WCTopping – [Your Name]`.
   b) Select **Open** to open your project.

1.3.3 Project Panel with Project WCTopping

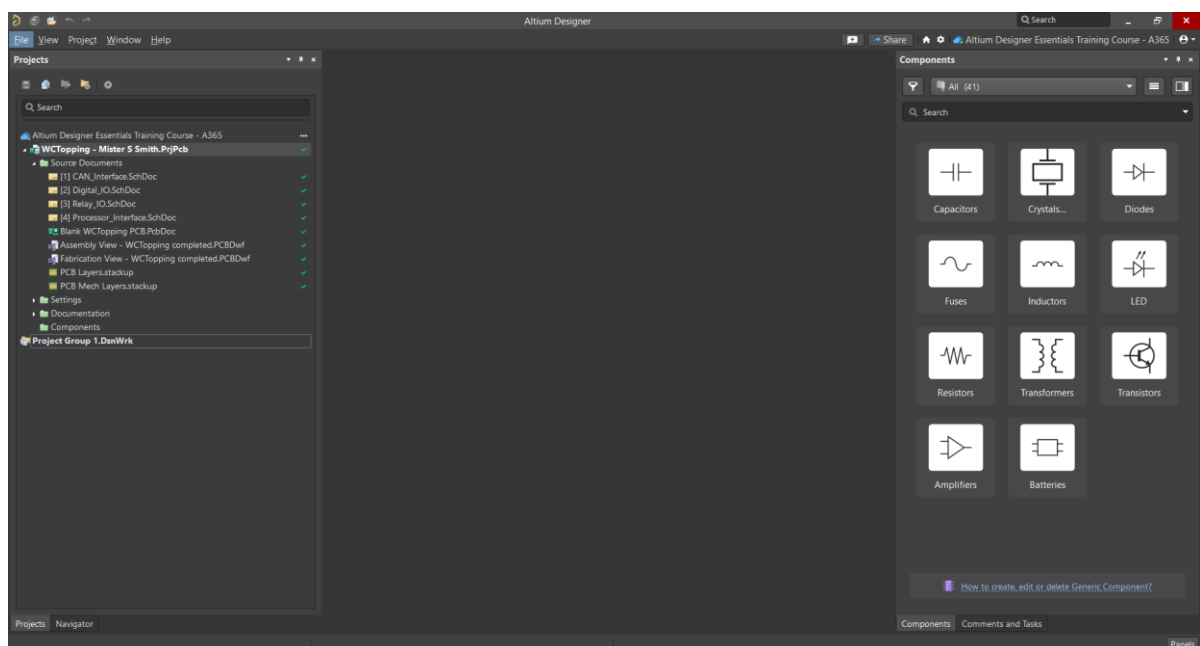5. With the project opened in Altium Designer, it should look similar to Figure 1.

Figure 1. First Project loaded

1.4 Grid Setting

1.4.1 **Setting the grid**

6. Start by double clicking on the `Processor_Interface.SchDoc` schematic to make it the focused schematic.

    a) Open the *Properties* panel from the **Panels** button `Panels`.
    b) In the *General* tab, check that **Snap Grid** value is `100mil`, as shown in Figure 2.
        If the **Snap Grid** Value is different, change the value to `100mil`.
    c) Repeat these steps later, if needed, for the other schematic pages.

> If you are using your own templates during the training and the template you selected was in Metric units you can change to Imperial units using the **CTRL+Q** shortcut keys in the *Properties* panel.

Figure 2. Set the Snap Grid to 100mil

1.5   Panels for Component Placement

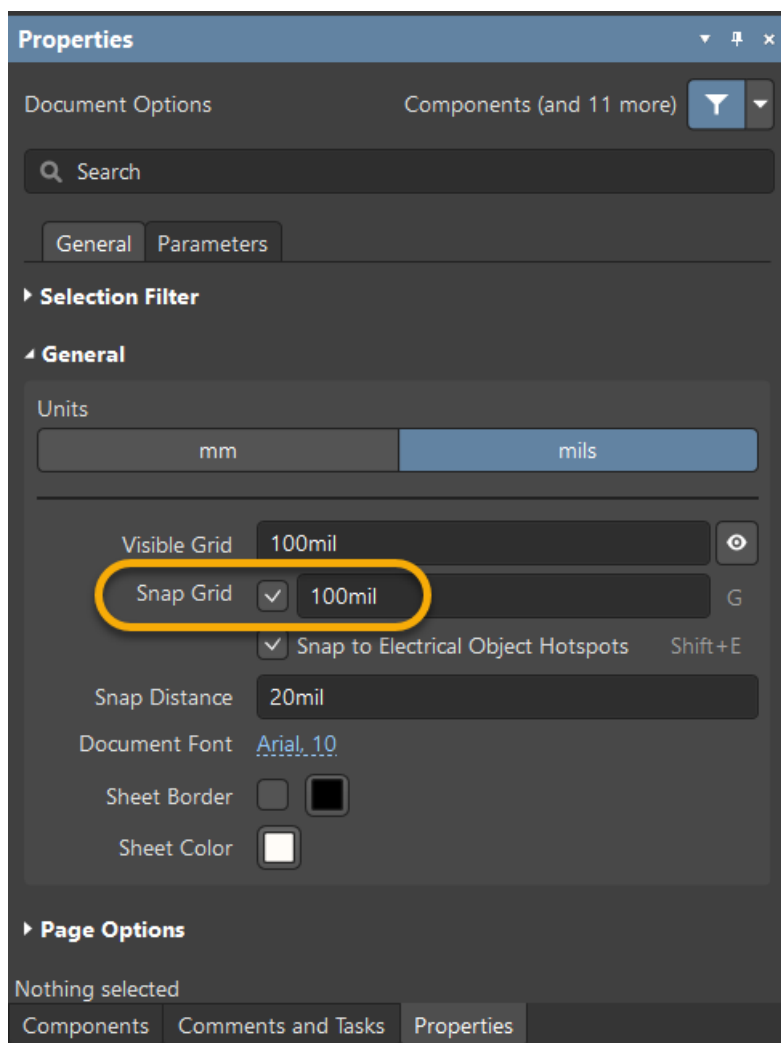Below, you will find an overview of two panels: the Explorer panel and the Components panel. We will begin the exercise on component placement in section 1.6 Component Placement.

1.5.1 Explorer Panel

The *Explorer* panel provides capability to view the Item categories and Items stored in the Workspace, what you see will depend on folder permissions. Among the Item categories available, you will find the 'Component Definition' category

7. Open the Explorer Panel using **K»R**, you will see a panel similar to Figure 3.

    a) Path field for the Item category

    b) Tree View for the Items category

    c) Items from the selected category

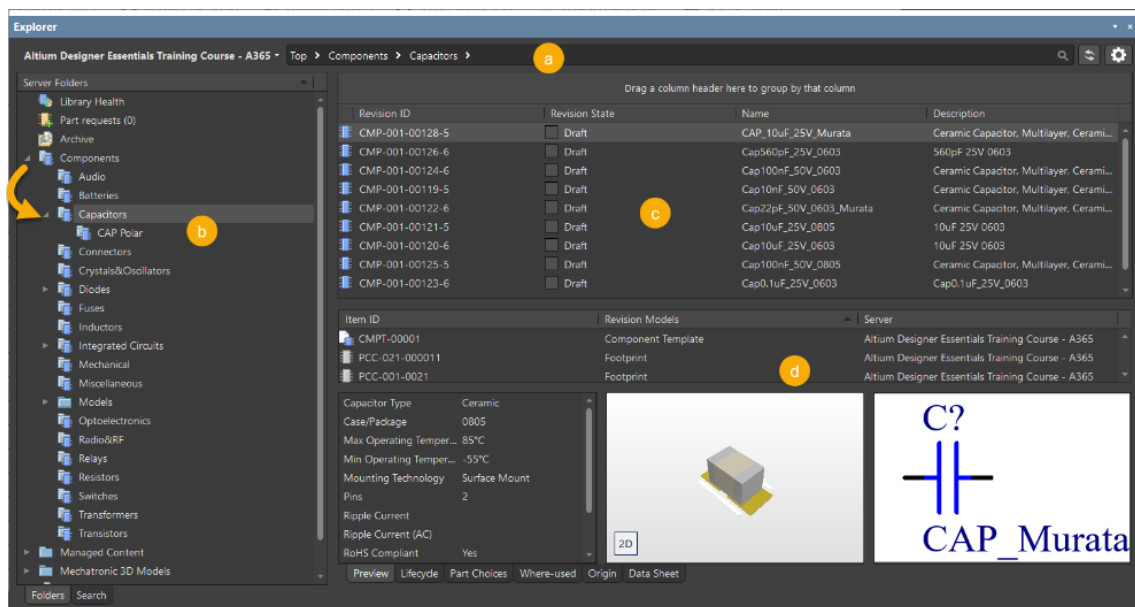   d) Details for the selected Item

Figure 3. Explorer Panel - Folder View

8. It is possible to place a component directly from the Explorer panel, by simply using mouse right-click, and then choose the 'Place' command.

9. A second view allows you to search for specific components, as shown in Figure 4.
   a) Activate the Search tab to see the second view
   b) Search Command Line
   c) Predefined Search Filter configurations (might be that no filter is predefined at your installation).
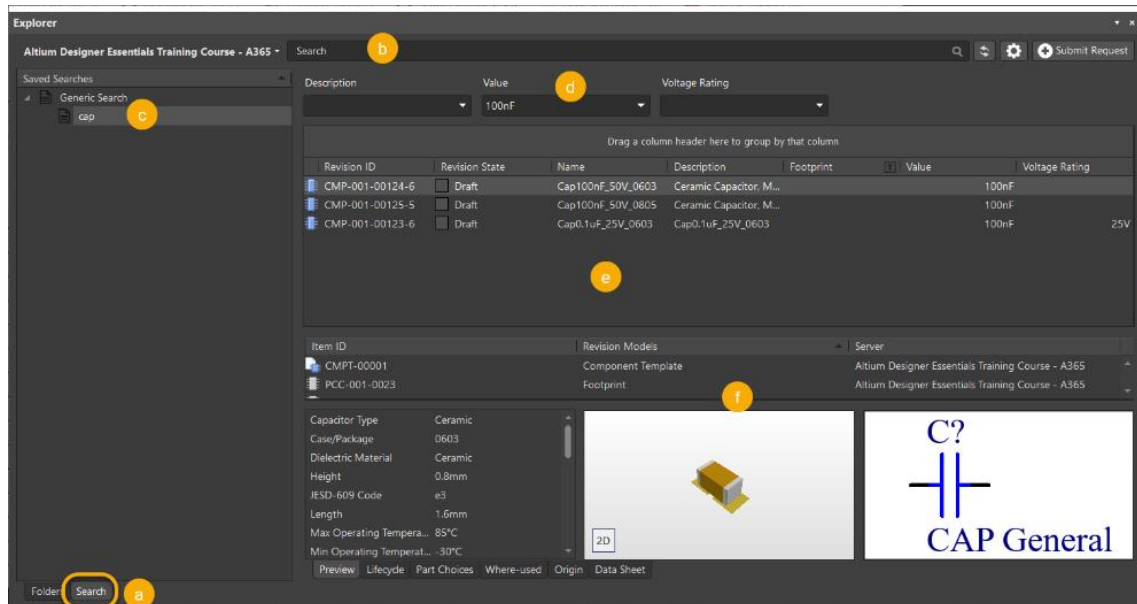   d) Filter
   e) Filter result
   f) Component Details

Figure 4 Explorer Panel - Search Component View

1.5.2 **Components Panel**

The Components panel provides direct access to available components. These components may be stored in various locations, such as a Workspace (the library concept used during this training), a database library, or a file-based library. Depending on the defined view and the active/inactive panel options, you can have different displays within the Components panel.

1.5.2.1 Components Panel – Compact View

10. The compact view starts with an overview **1** for different component categories.

   a) You can use the drop-down list and/or the search.

   b) Or you can select a category to see all components from that category **2**

   c) The button Switch Mode ▯▮ allows you to switch between Compact and Normal View for the *Component* panel.

   d) The Operations menu Button opens configuration options.

   e) You can expand / minimize the detail area [CMP-001-00060-3] (≫) for selected components.

   f) Preview of schematic symbol and pcb footprint
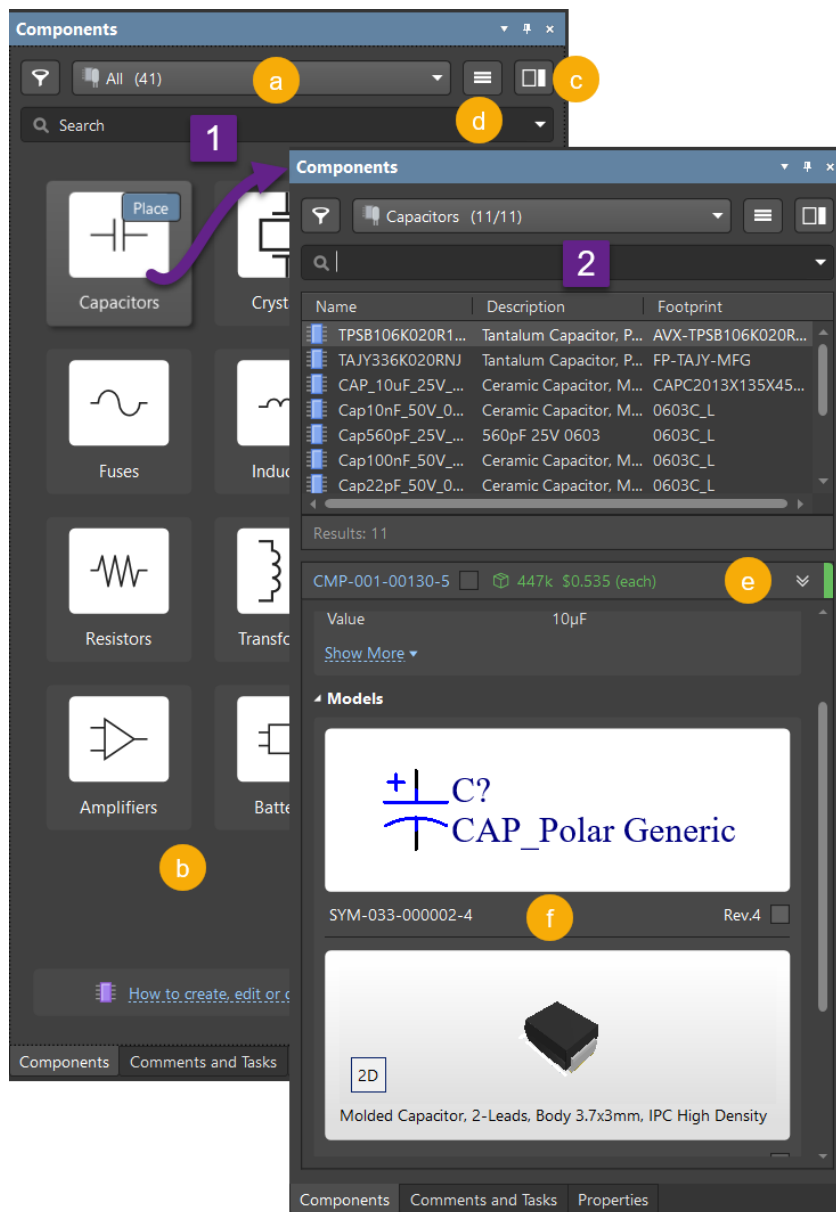

Figure 5. Components Panel – Compact View

Altium

1.5.2.2 Components Panel – Normal View

11. The Normal View from the *Components* panel can include:

    a) The Categories of components, as defined at the workspace.

    b) A filter for component search.

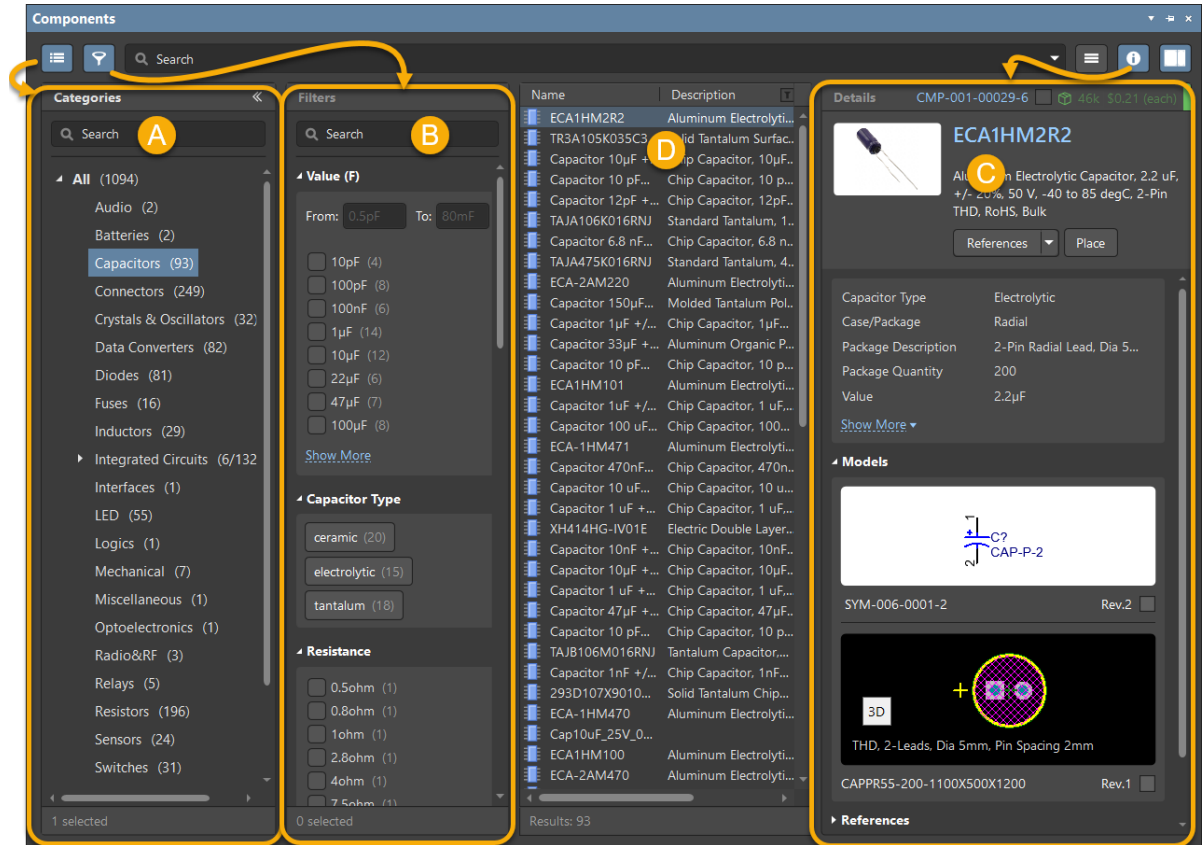   c) Details about the selected components

    d) The component list.

Figure 6. Components Panel - Normal View

1.6 Component Placement

Next, we will place components using the Components panel.

1.6.1 Document Processor Interface - Schematic Component Placement

12. Ensure that the `Processor_Interface.SchDoc` is still the active schematic document. Figure 7 below will be used as a reference for the following section.
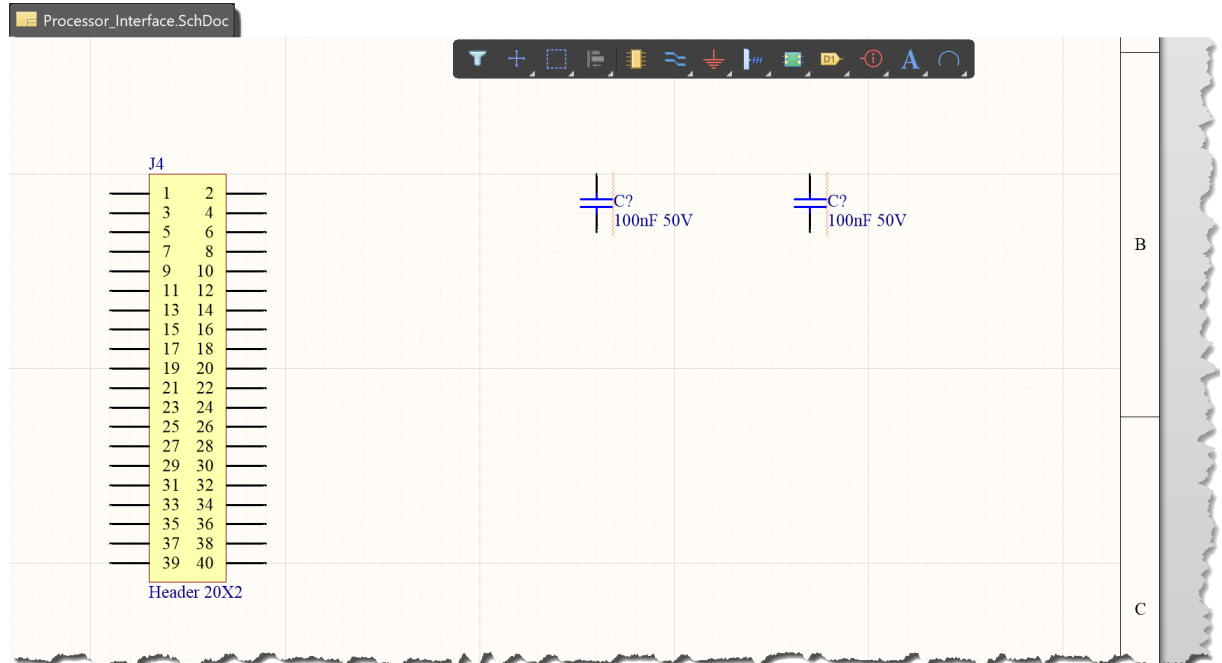

Figure 7. Processor Interface Schematic

13. To access the *Components* panel, if not already open, you can use the **Place » Part…** command from the main menus, the **Place » Part...** command from the right-click menu of the schematic, the Panels button [Panels] at the lower right of your workspace, or the part command in the Active Bar [⬛].

14. Select the category `Connectors` from the drop-down list of libraries at the *Components* panel, as shown in Figure 8 below.

Figure 8. Select the Miscellaneous Connectors library

15. At the Search Field – type `Header` to search for the first component

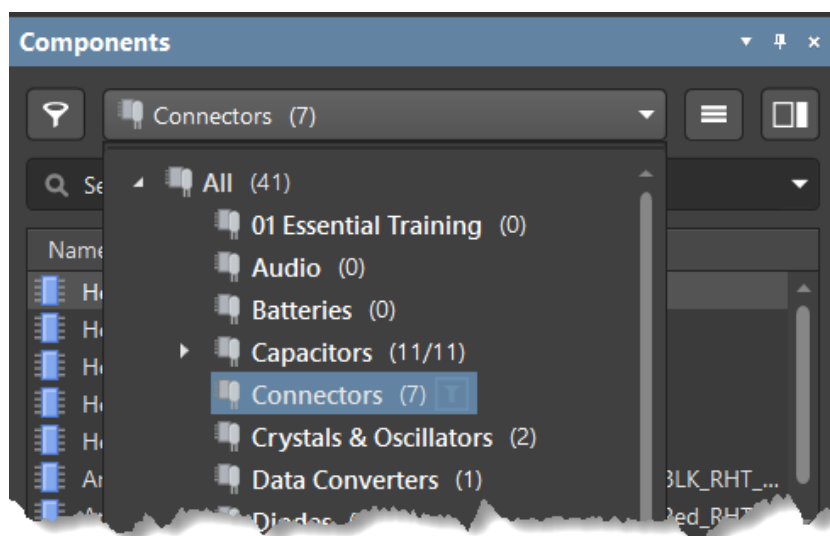16. If needed, scroll down the list of components to find the 40-pin I/O Connector called `Header 20x2,` as shown in Figure 9Figure 8.
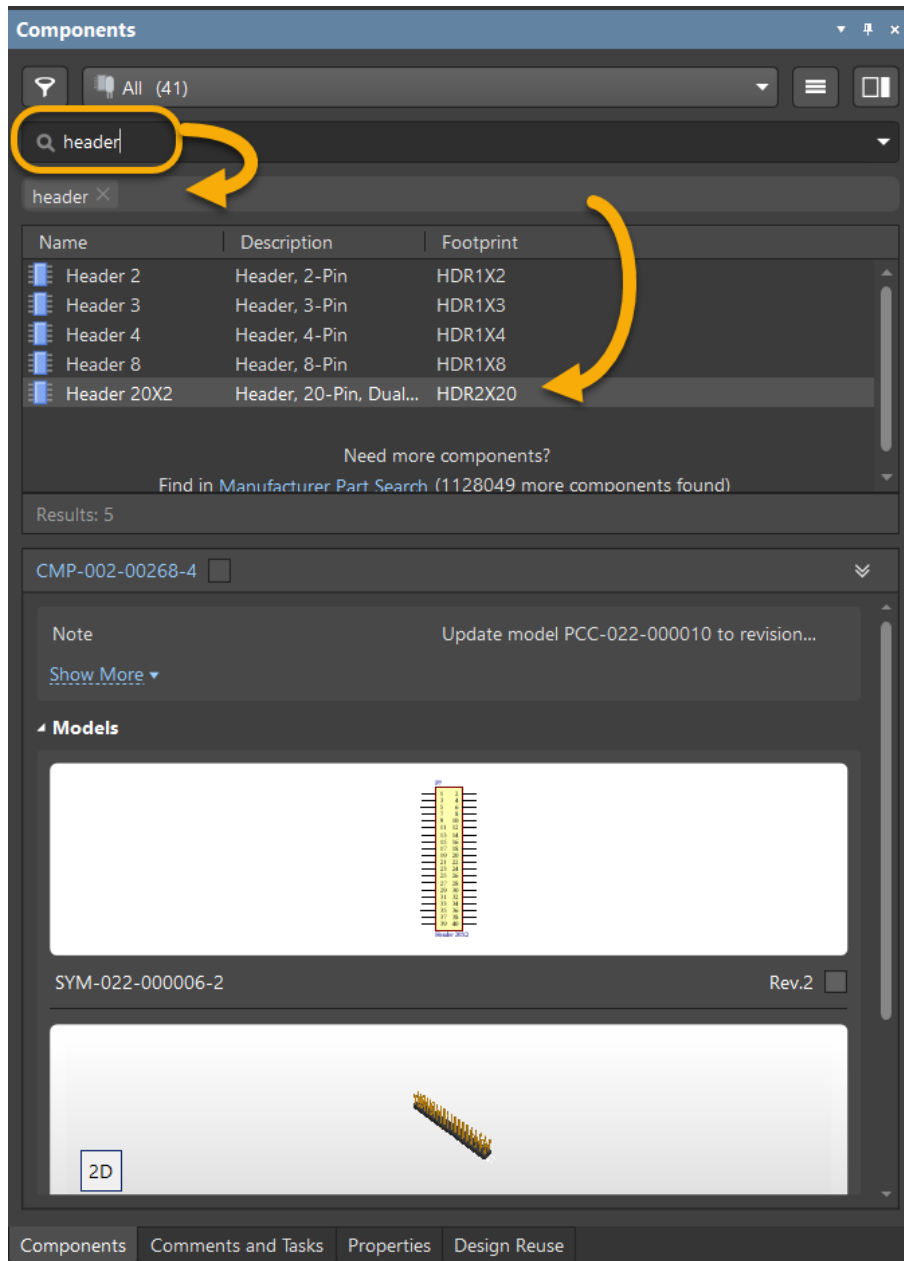

Figure 9. Header 20x2

17. Right-click to **Place Header 20x2** and move the cursor with the attached symbol inside the schematic page.

18. Before placing the connector on the schematic, press the **TAB** key:

a) In the *Properties* panel that opens, any changes to the properties will be applied to subsequently placed objects during this command.

b) Change the reference designator to J1 as shown in Figure 10.

c) To apply the changes press, **Enter** or click the **Pause** icon ⏸ and continue with the placement.

> The **TAB** key is used to pause the command that you are currently in. This allows you to make changes to an item's properties. To resume the command, hit the **Enter** key, or the **Pause** ⏸ icon in the middle of the editor to continue with the command.

Figure 10. Component Properties dialog

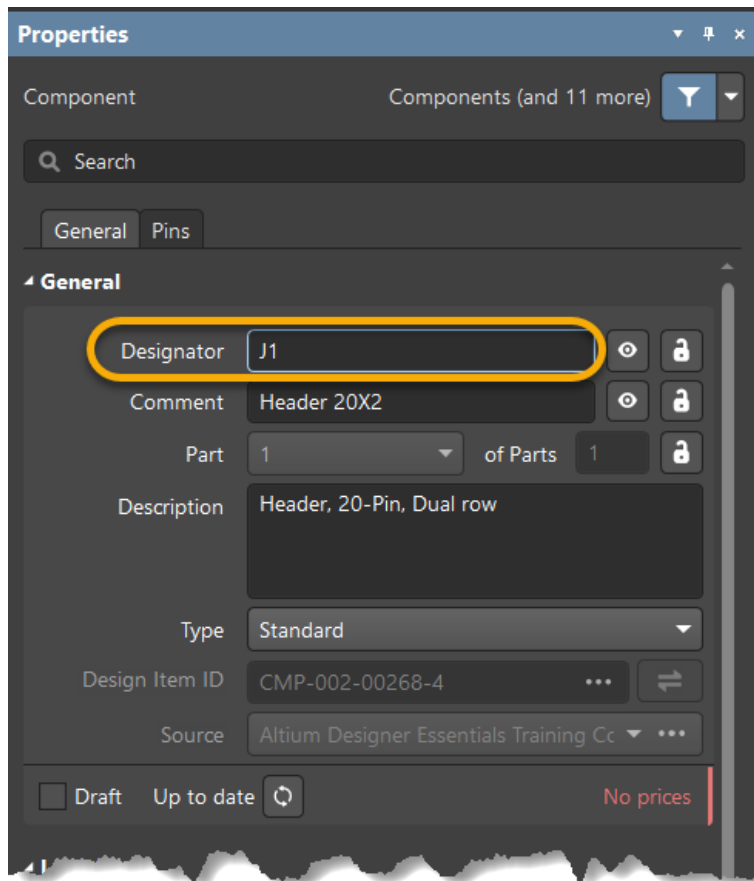19. While still in placement mode with the connector attached to the cursor, experiment with the following commands to orient the component.

   a) **Spacebar** to rotate counterclockwise 90°

   b) **Shift+Spacebar** to rotate clockwise 90°

   c) **X** to flip along the X-axis

   d) **Y** to flip along the Y-axis

> If a placed component needs to be rotated, select the component and press **Spacebar** to rotate counterclockwise 90° or **Shift+Spacebar** to rotate clockwise 90°**.**

20. Left-click anywhere in the schematic to place the component.

21. Going back to the Components panel, from the drop-down list, select the Category `Capacitors`, use the library search (100nF), or simply navigate down the list of components, and place two `Cap100nF_50V_0603` capacitors, into the `Processor_Interface.SchDoc.` Making the *Components* panel larger may help you see the full name of the component you wish to select and place, Figure 11.

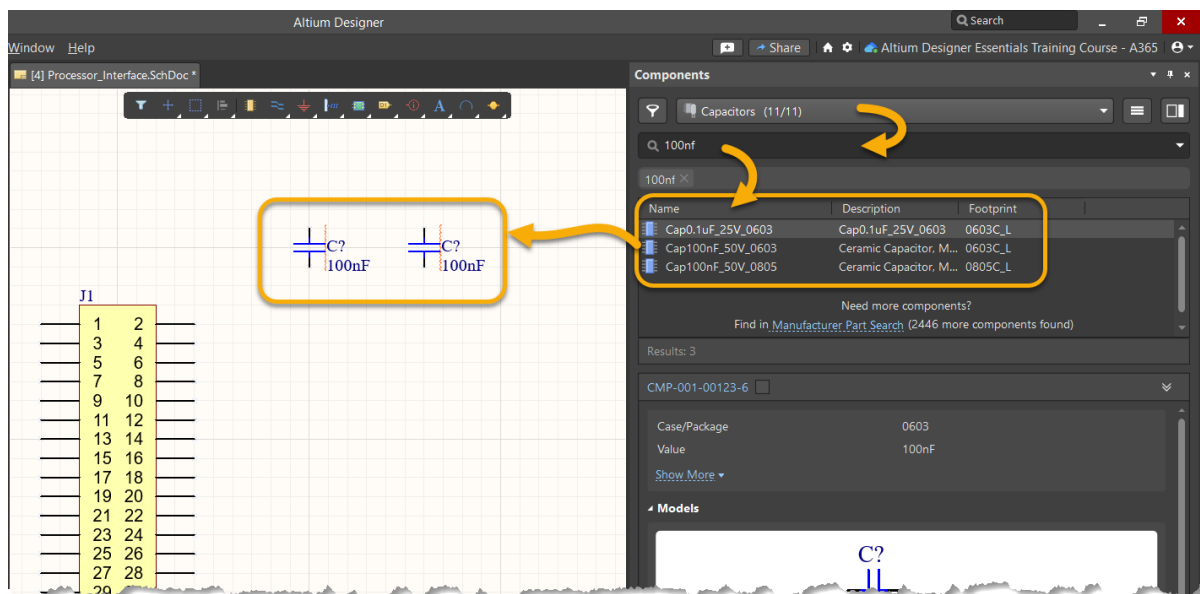22. **Right-click** to end the command when finished.

Figure 11. Placing two CAPS with 100nF 50V 0603

23. Save the schematic by right-clicking on the schematic in the *Projects* panel, and click **Save.**

Altium.

1.6.2 CAN_Interface Schematic Component Placement

Next let's place some components for the CAN_Interface, as shown in Figure 13.

24. Open the `CAN_Interface.SchDoc.`

25. Select the Components panel.

26. From the integrated Section, see Figure 12, place the following components, as shown in Figure 13.

    a) MCP2515-E/SO

    b) MCP2551-E/SN
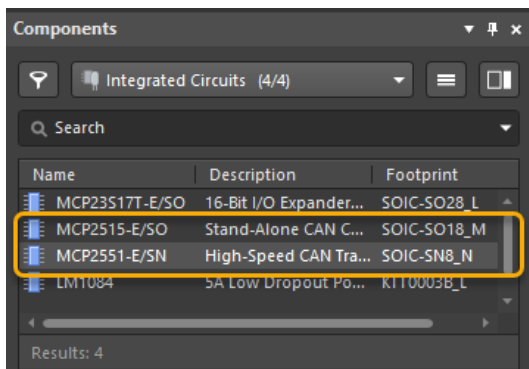

Figure 12. Integrated Circuits
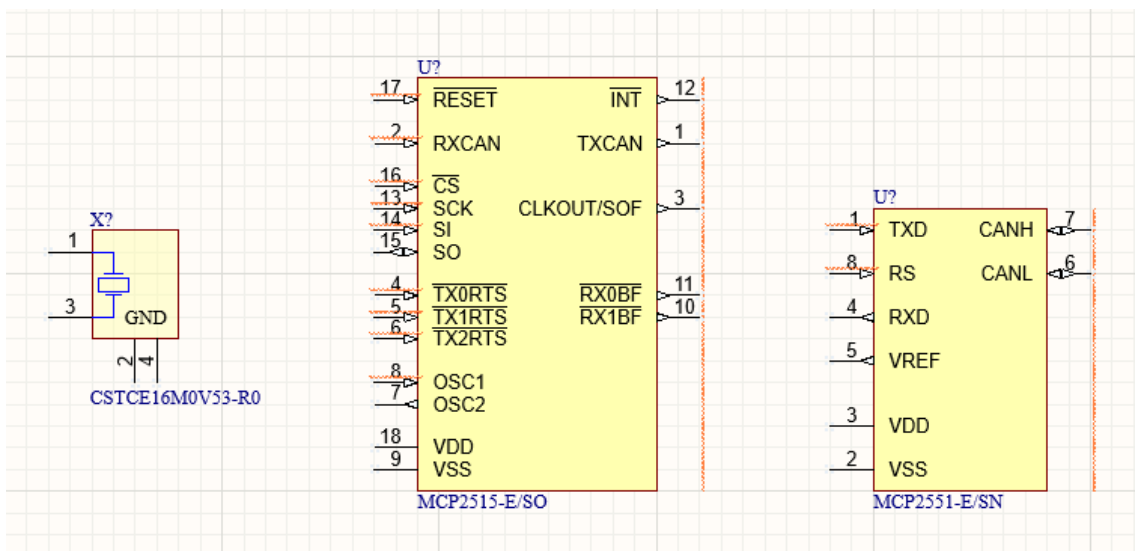

Figure 13. CAN_Interface Schematic

27. Next find and place the crystal CSTCE16M0V53-R0, by using the search option in the Components panel, see Figure 14.
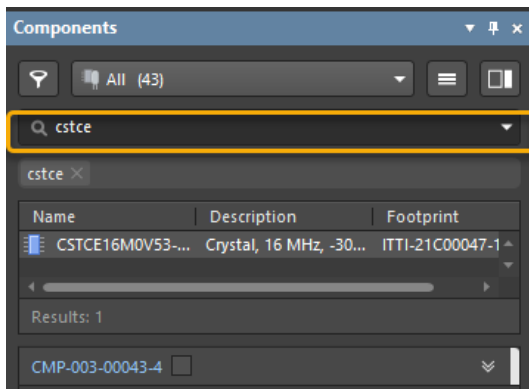

Figure 14. Place Crystal

1.6.3 Relay_IO Schematic Component Placement

28. Open the `Relay_IO.SchDoc`. Use Figure 15 as reference for component placement.
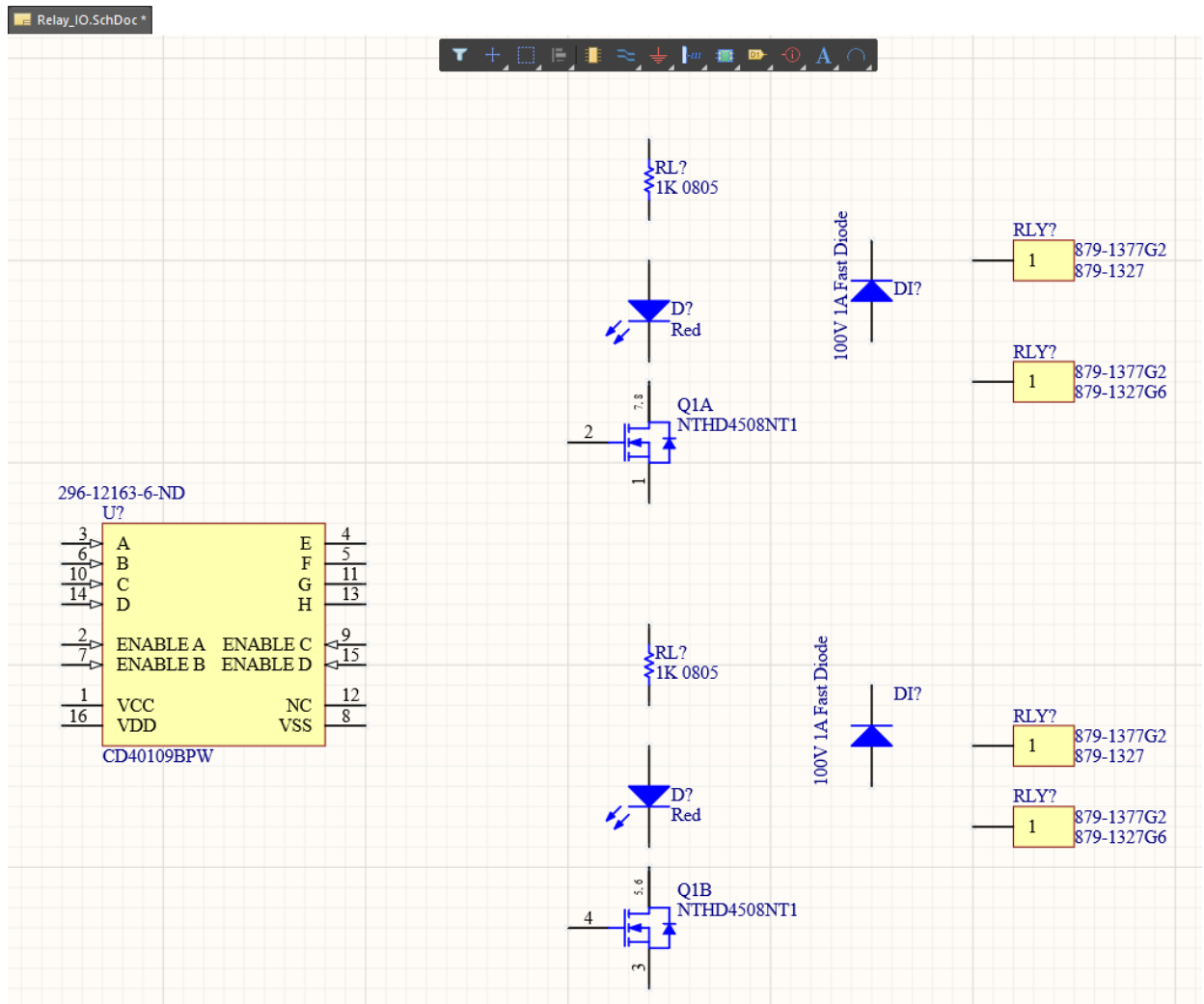

Figure 15. Final Relay I/O Schematic we create with the next steps.

29. From the *Components* panel, select the category `Transistors`.

30. Place NTHD4508NT1. Navigate to the component to find it as shown in Figure 16.

    a) This is a multi-part component as indicated by the part information below the Symbol preview as shown in Figure 16. A multi-part component is represented by multiple schematic symbols. Expand the *Models* pane if necessary to see this.
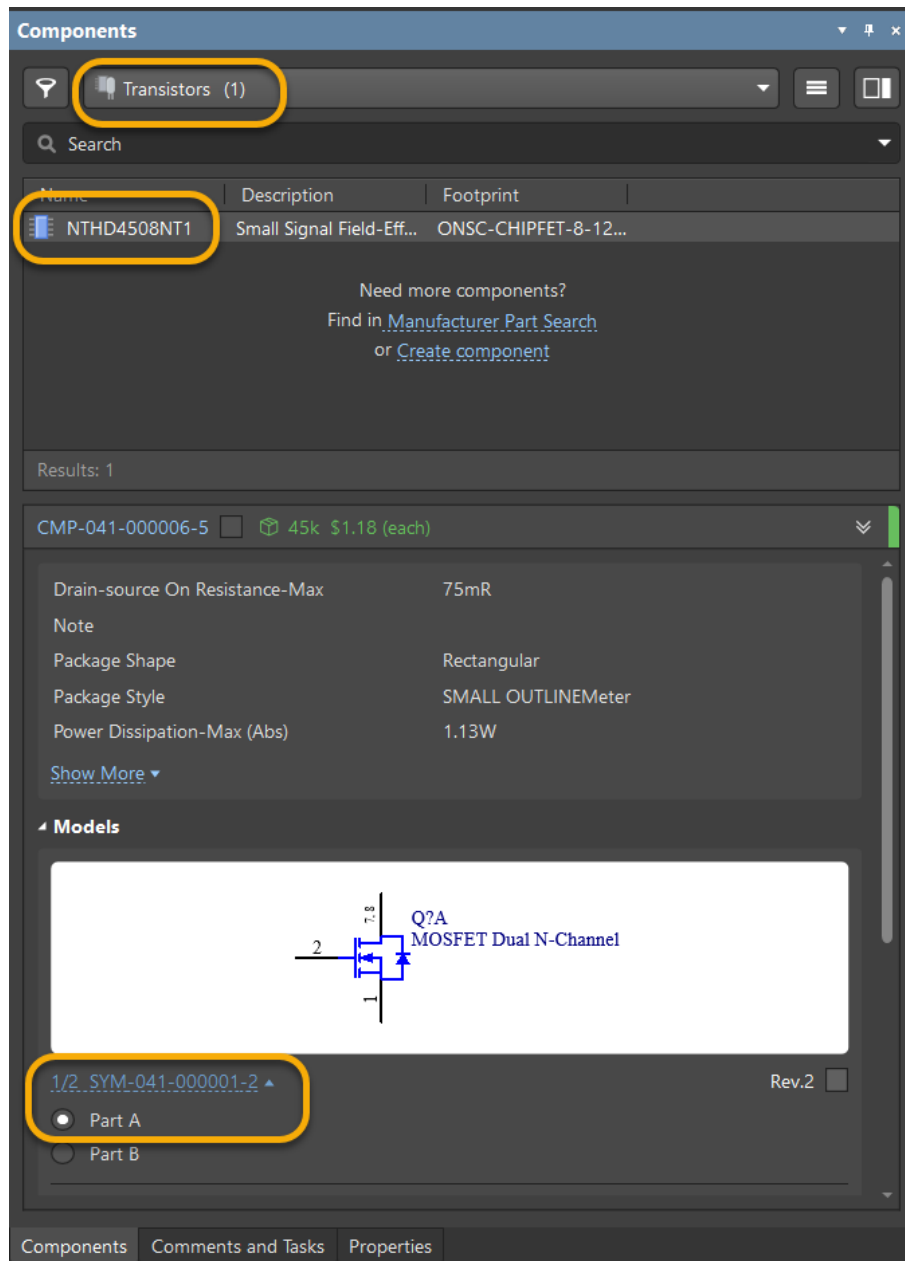
Figure 16. Placing NTH Multi-Part into the Relay I/O schematic

    b) Select `Part A`.

    c) Right-click and place the component. During placement, press the **TAB** key to change the designator to `Q1`, hit **Enter** and place the component.

    d) Place `Part A`, and now `Part B` will be on your cursor. Place that part as well. This is considered a multi-Part component that is represented by more than 1 schematic symbol.

    e) Right-click to end the command after placing `Q1A` and `Q1B`.

31. Search and place the remaining components for the `Relay_IO.SchDoc`. Refer to Figure 17 to reference the component placement.

a) `Res1K_0805` (2)

b) `LED_Red0603` (2)

c) `Diode 1N4148` (2)

d) `Anderson-1Pin BLK_PCB_RHT` (2)

e) `Anderson-1Pin Red_PCB_RHT` (2)

f) `CD40109BPW`
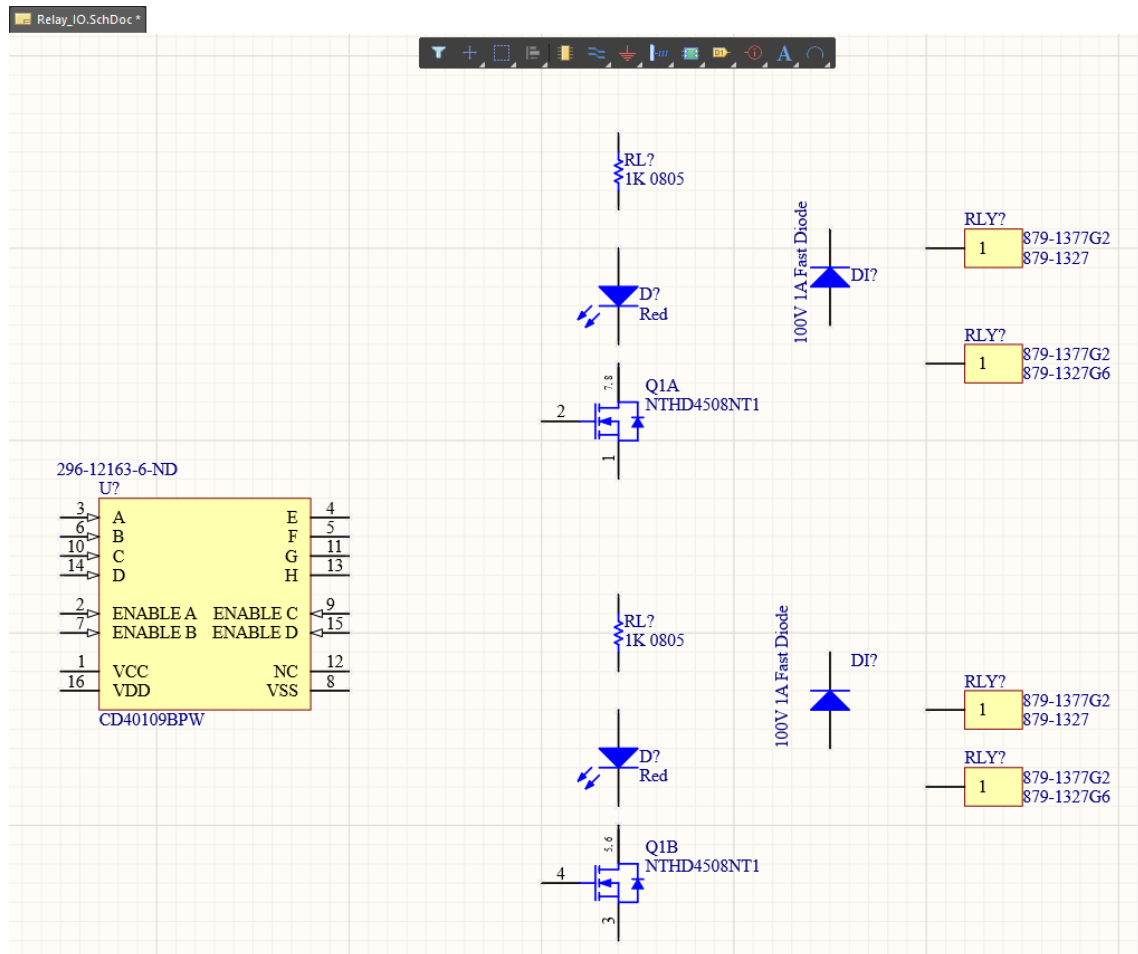
32. Save the `Relay_IO.SchDoc` schematic.

Figure 17. Relay_IO schematic symbol placement

1.7 Adding an Existing Circuit to Your Design

1.7.1 Add New Schematic

33. In Altium Designer, right-click the `WCTopping – [Your Name]` project in the *Projects* panel and select **Add New to Project » Schematic.**

34. Save the new document by using **File » Save All** with the Name `Power Supply.SchDoc.`

35. Change the position of the new schematic with Drag and Drop. The Power Schematic should be at position five, below the existing four schematics.

1.7.2 Add Reuse Block

36. If not already open, open the `Power Supply.SchDoc` schematic from the *Projects* panel and ensure it is focused document.

37. Open the panel *Design Reuse* by selecting the **Panels** button Panels and the *Design Reuse* command at the lower right side of your workspace,

38. At the *Design Reuse* panel, Figure 18:

    a) Select from the drop down list the Training Workspace `Altium Designer Essentials Training Course – A365.`

    b) Select from the list of available reuse blocks `Power Supply WCTopping.`

    c) Place the predefined Circuit at the `Power Supply.SchDoc` sheet.
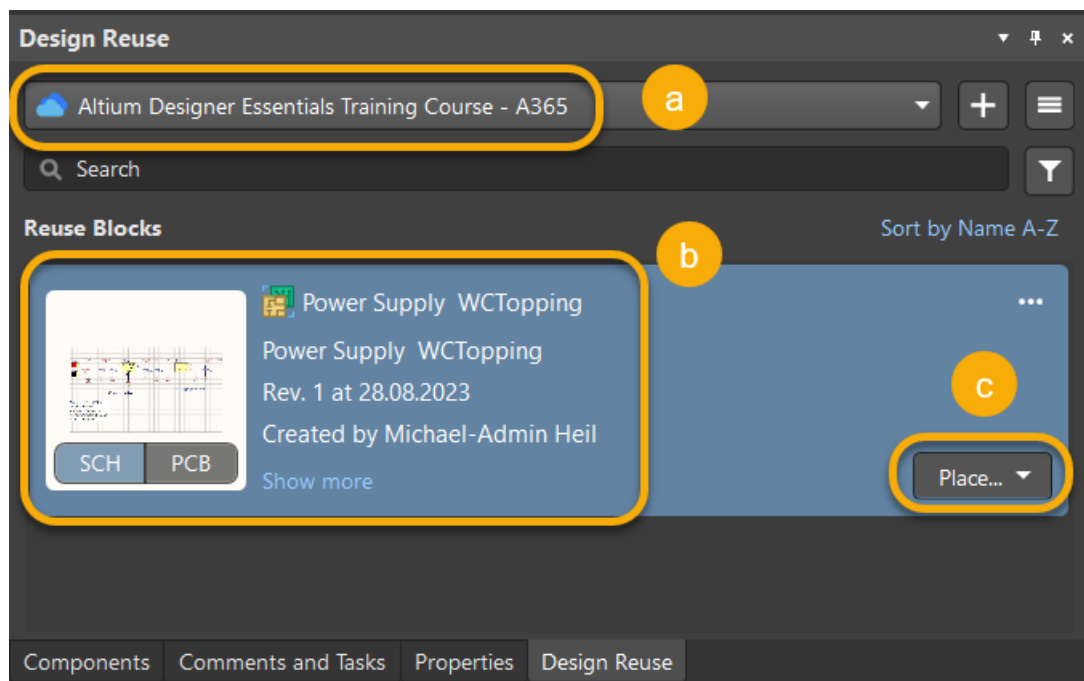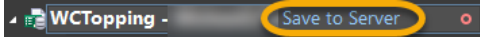

Figure 18. Design Reuse panel

39. Right-click the `Power Supply` schematic in the *Projects* panel and click **Save**.

1.7.3 **Save to Server**

Now that we have updated the project with components and a new page, we can save the modifications to the server.

40. At the *Project* panel, next to the project name you find the command
    Save to Server ◢ 📇 **WCTopping -** ▒▒▒▒▒ (Save to Server) ○ .

41. Select **Save to Server**

42. At the dialog *Save [Project Name],* as seen at Figure 19
    a) Activate the checkboxes for the files that are not under version control, skip the file `WCTopping - Mister S Smith.PrjPcbStructure`.
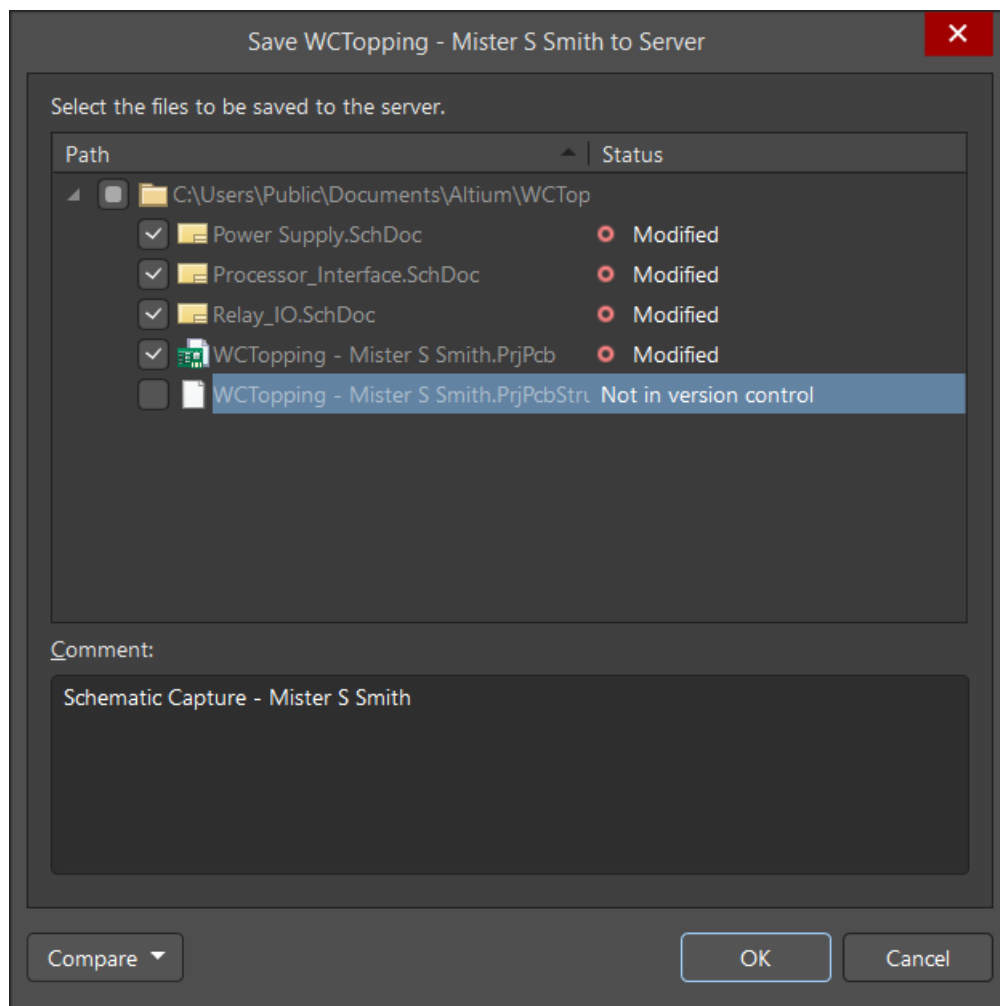   b) Add the comment `Schematic Capture – [Your Name]`
    c) Click on **OK.**

Figure 19. Save to Server

Altıum

43. The *Projects* panel should now look like the one in Figure 20 below. Next to all files names there is a green checkmark, indicating the local project and files are in-sync with the server.
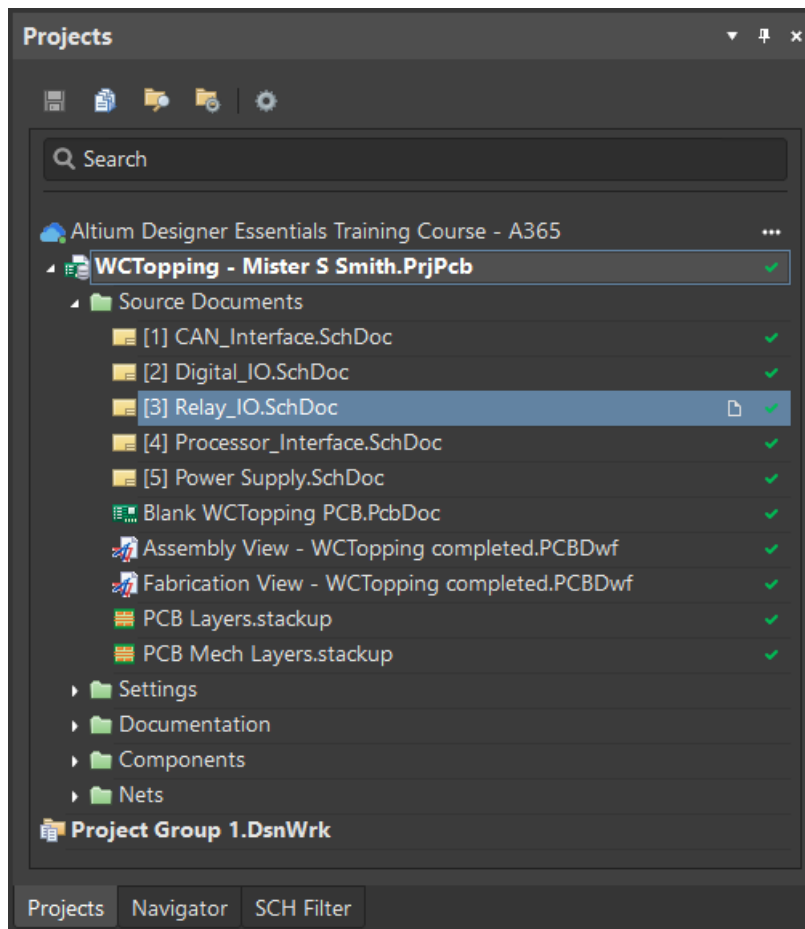

Figure 20. Renamed Project Files

> To find the project folder, where this project is locally stored, **right-click** on the project name and select the **Explore** command.
> The folder you will see in the Window File Explorer is the local storage area for GIT VCS. You will see Altium files (e.g., *. SCHDOC, *. PRJPCB, *. PCBDOC), but also GIT related files (based on your Window File Explorer settings).

44. When finished, close the project and the schematic document (Window » Close All)

Congratulations on completing the Module!

Module 8: Open an Existing Project and Schematic Capture

from the

Altium Designer Essential Course with Altium 365

Thank you for choosing Altium Designer