Altium Designer

Advanced Training with Altium 365

Creating and Using Design Reuse Block in a Workspace

Software, documentation and related materials:

Table of Contents

Creating and Using Design Reuse Block in a Workspace 3

1 Purpose 3

2 Shortcuts 3

3 Preparation 4

4 Workspace Structure 5

5 Accessing the Design Reuse Panel 6

6 Creating a Design Reuse Block 7

6.1 New Design Reuse Block 7

6.2 Copying the Existing Information to Define the Reuse Block 8

6.3 Sync Schematic and PCB from the Design Reuse Block 10

6.4 Saving the Reuse Block to the Server 11

7 Using an Existing Design Reuse Block 12

7.1 Creating a New Project 12

7.2 Using a Reuse Block 14

Creating and Using Design Reuse Block in a Workspace

1 Purpose

The Design Reuse Blocks functionality provides a simple and easy way to save and reuse sections of design circuitry. They can be added into any design, with no need to start from scratch each time. The Design Reuse Blocks system lets you save any selection of:

a) Circuitry on a single schematic sheet

b) Circuitry in a PCB design, including the components and the routing

Additionally, when connected to an Altium 365 Workspace, you can create a Reuse Block that can contain schematic circuitry and its physical representation for the PCB. When such a Reuse Block is placed on a schematic sheet, its physical representation will be placed automatically in the PCB document during the ECO process.

In this module, we will show you how to create a Design Reuse Block in a Workspace and place it in a new design.

2 Shortcuts

Shortcuts used when working with Creating and Using Design Reuse Block in a Workspace

F1	Help
W » V	Vertical Split / Vertical Tile
W » H	Horizontal Split / Horizontal Tile
T » P	Open Preferences
CTRL+S	Save Document
K » R	Open Explorer panel
PCB active document:	
C » K	Component Links
Shift+S	Single Layer Mode
TAB	Extend selection
SCH active document :	
D » U	Update PCB document

3 Preparation

1. Close all existing projects and documents.
2. Next, create a copy of the Training Project: `Creating and Using Design Reuse Block in a Workspace`.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Enable the folder view button  .
5. Navigate to the predefined Training Project `Creating and Using Design Reuse Block in a Workspace (Top\Projects\Altium Designer Advanced Training Course\...)`.
6. Select **Open Project as Copy...** 

.
7. In the new dialog *Create Project Copy:*
   a) Add your name to the project name: `Creating and Using Design Reuse Block in a Workspace – [Your Name]`.
   b) Add a description: `Altium Advanced Training – [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 opened the Project for you in the *Projects* panel, this may take up to 1 minute.

Hint: For details how to copy the predefined training project, see module *03 Getting started - Opening a Project*.

4 Workspace Structure

9. Open the *Explorer* panel from the **Panels** button, or with the combination **K»R**.
10. Navigate to the branch `Managed Content - Design Reuse Blocks`. Use Figure 1 as reference.
11. You will see the branch `Design Reuse Blocks` and two subfolders `PCB Snippets` and `Schematic Snippets`.

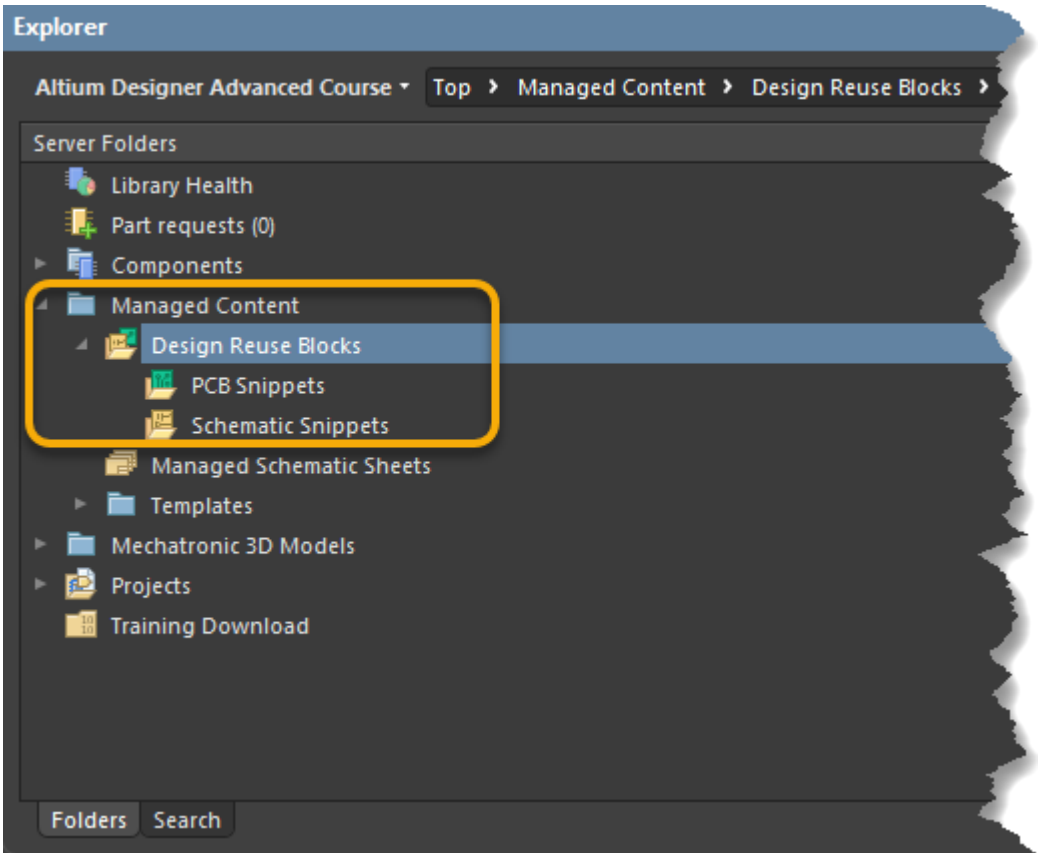


Figure 1. Explorer Panel with the Design Reuse Block branch without a Design Reuse Block configuration

Hint: The Training Workspace you are using, but also the workspace from your company, may already include some Design Reuse Blocks or has a different structure to organize the Items in the workspace.

Note: For this Training, you have the permissions to create items in the branch `Managed Content - Design Reuse Blocks`. In your company workspace, this can be impossible for you, because of various configurations for the user roles and permissions.

5 Accessing the Design Reuse Panel

12. Open the *Design Reuse* panel from the **Panels** button.
13. If the option **Local** is active, change the information to the training workspace. Use Figure 2 as reference. In this panel, you can find the existing Design Reuse Blocks that are ready for use, but you can also create new Design Reuse Blocks.

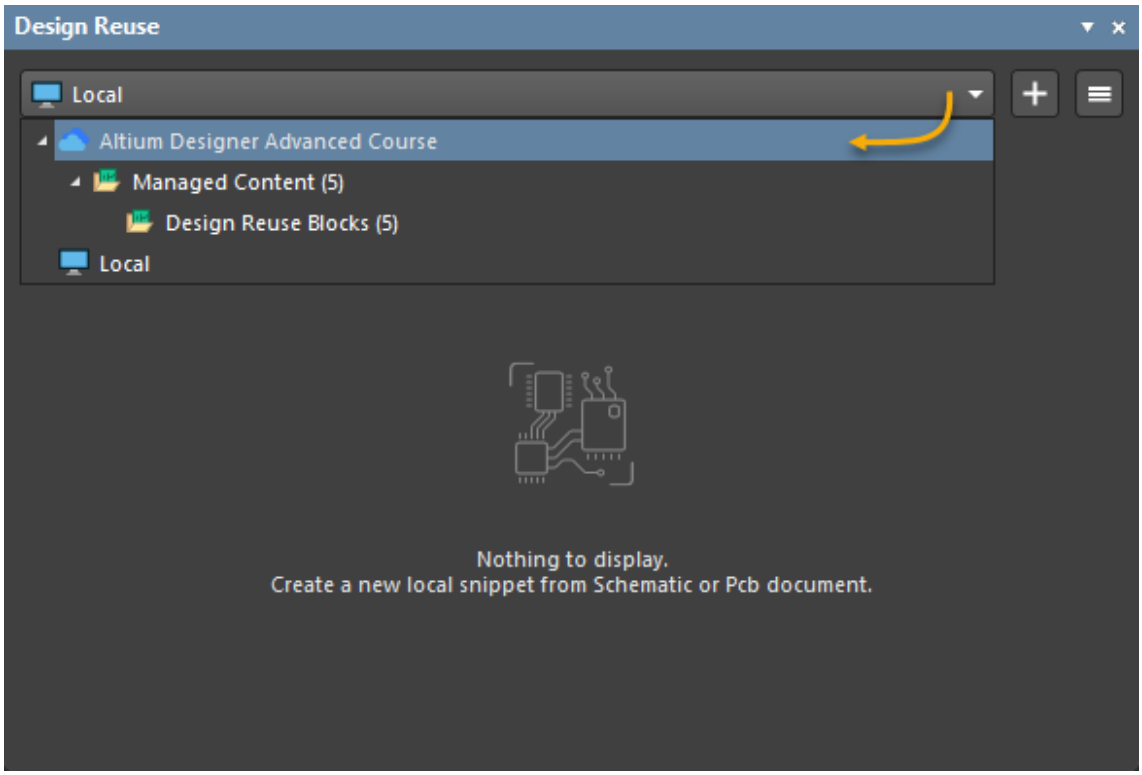


Figure 2. Change from Local Design Reuse to Workspace Design Reuse

6 Creating a Design Reuse Block

6.1 New Design Reuse Block

In this training, you will create a Design Reuse Blocks for a power circuit.

14. Create a new Design Reuse Block, **File » New » Reuse Block...** or select the option **Create Reuse Block** in the *Design Reuse* panel.
15. Open or change focus to the *Projects* panel. A new branch was added to the *Projects* panel. Under the Workspace name, you can see `New Reuse Block` project added with a default Schematic and PCB file, Figure 3.

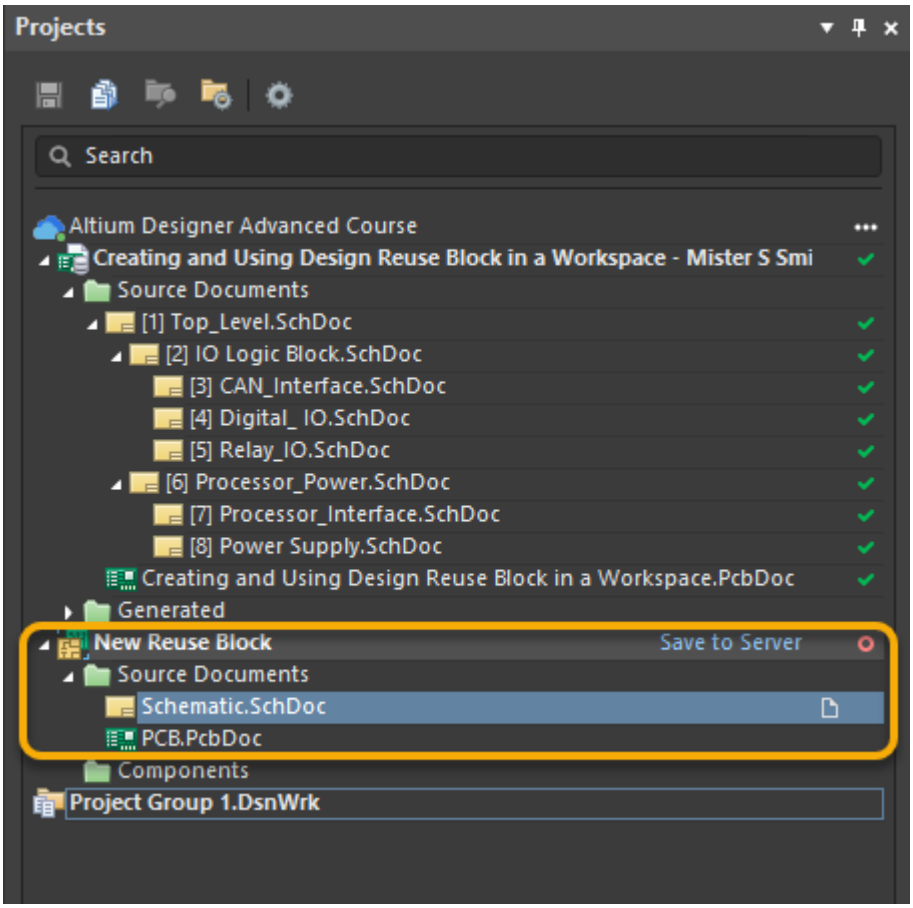


Figure 3. New Reuse Block

16. If you open the Schematic and the PCB from the new Design Reuse Block, you will see that both documents are empty. With the next steps, you will change that by copying information from the existing schematics and PCBs.

6.2 Copying the Existing Information to Define the Reuse Block

17. Before you copy an existing circuit, you need to ensure the preferences are set correctly for this task:
    a) Open the *Preferences* and navigate to the branch `Schematic – Graphical` editing.
    b) Deactivate the option **Reset Part Designators On Paste**.
    c) Close the *Preferences* with **OK**.

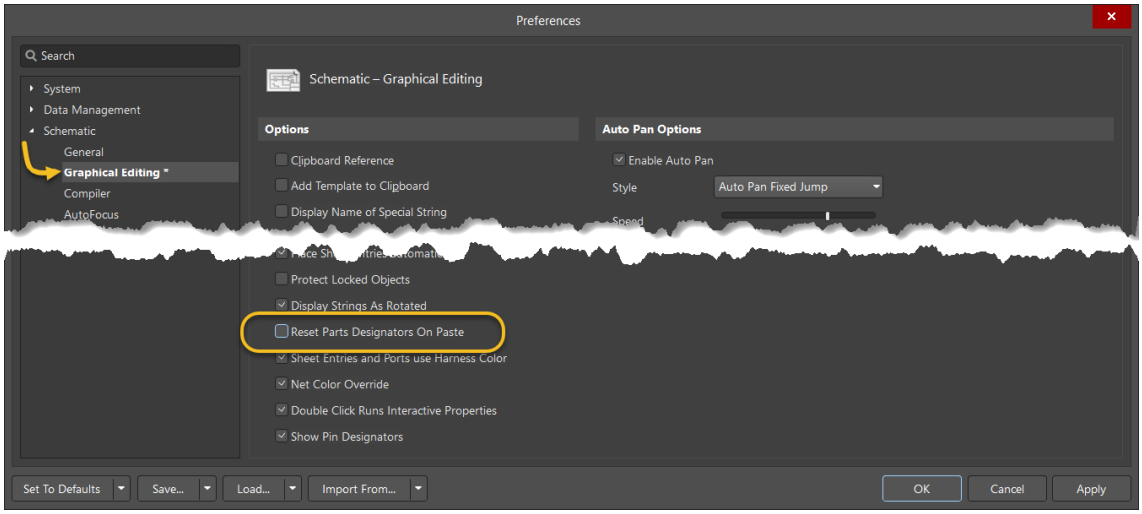


*Figure 4. Preferences, option **Reset Part Designators On Paste***

18. From the Training project `Creating and Using Design Reuse Block in a Workspace`, open the PCB document and the schematic file `Power Supply.Schdoc`.
19. Select all objects in the schematics, as shown in Figure 5.

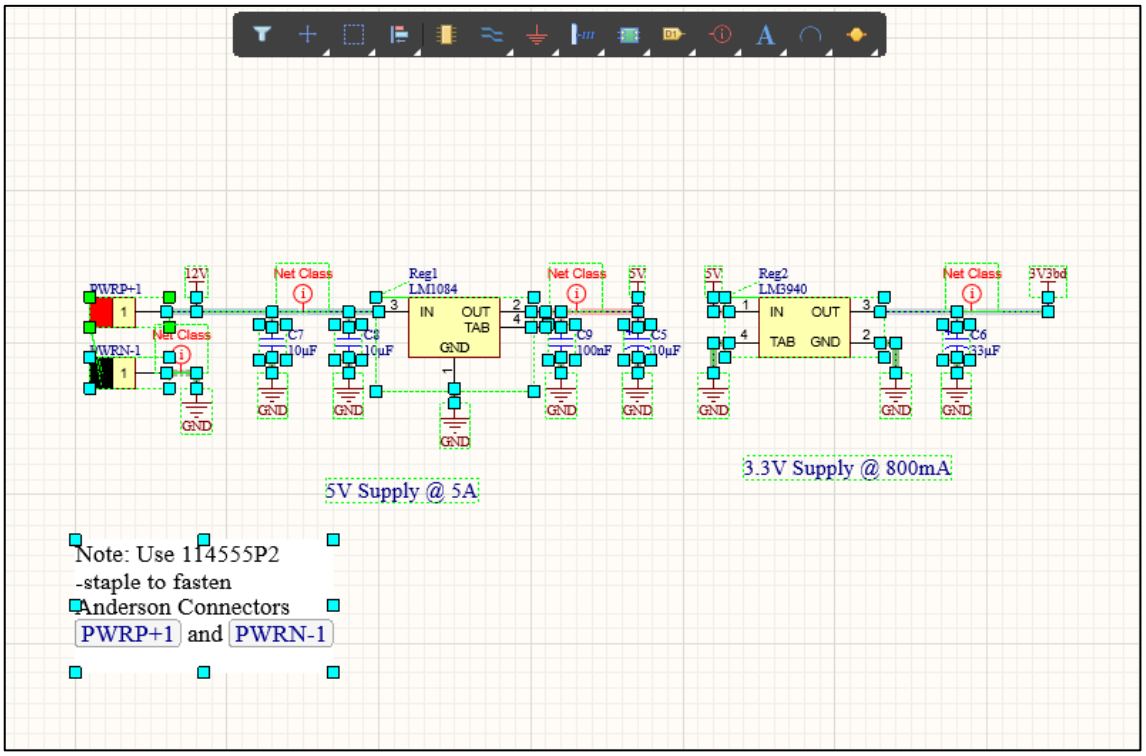


Figure 5. Selecting the Power Supply

20. Next, you will copy the power circuit. Follow the steps below:
    a) Copy all selected objects to the clipboard, **CTRL+C**.
    b) Change the focus to the default schematic `Schematic.SchDoc` from the New Reuse Block.
    c) Paste the power circuit into the New Reuse Block schematic using **CTRL+V**.

> Note: With deactivating the option **Reset Part Designators On Paste**, the **Paste** command doesn't reset the Designators to '?'. This is important for the next steps.

21. Change the focus back to the schematic `Power Supply.SchDoc`.
22. With the power supply circuit still selected, execute the command **Tools » Select PCB Components** (**T»S)**. Altium Designer will change the focus to the PCB document and select the corresponding Footprints from the selected Symbols.
23. Ensure you are in 2D mode.
24. Press **TAB** once to extend the selection to the first segments of the PCB routing.
25. The PCB information you need for your Design Reuse Block is selected. Copy that information into the Design Reuse Block PCB document:
    a) Copy all selected PCB objects to the clipboard, **CTRL+C**.
    b) Select a reference point, for example, **Pad 1** from the component PWRP+1.
    c) Change the focus to the default PCB `PCB.PcbDoc` from the New Reuse Block.
    d) Paste the PCB Footprints and routing from the power circuit into the New Reuse Block PCB using **CTRL+V**. You don't need an exact position, just paste into the Board Region Area.
    e) Save the work you have done so far with **File » Save All**.

6.3 Sync Schematic and PCB from the Design Reuse Block

Next, you need to synchronize the Schematic with the PCB from the New Design Reuse Block.

26. Change the focus back to the Schematic from the New Design Reuse Block.
27. Execute the command **Design » Update PCB Document PCB.PcbDoc.**
    a) In the *Components Links* dialog, select the **Automatically Create Components Links** option.

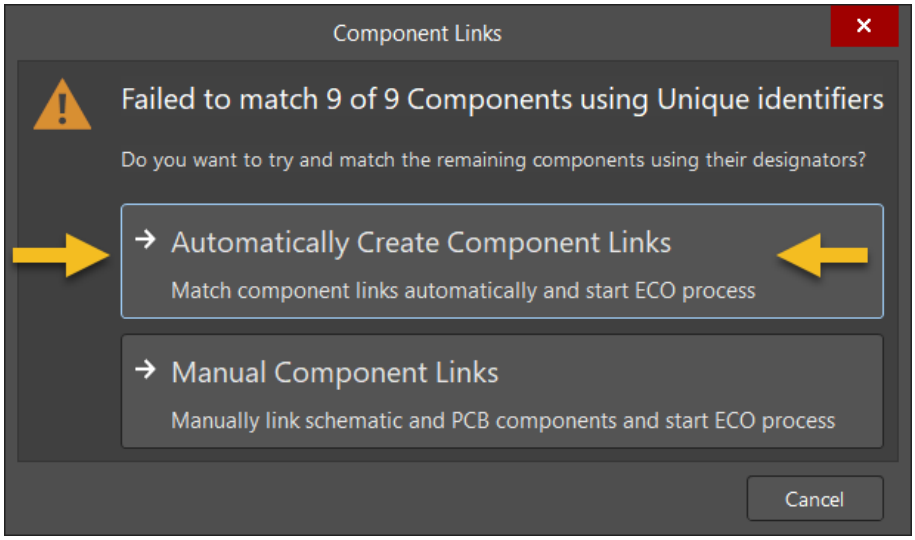


Figure 6. Components Link

b) Confirm the changes with **OK**.
c) In the *Engineering Change Order (ECO)* dialog, select **Execute Changes**.
d) When the update is finished, close the ***ECO*** dialog with **Close**, Figure 7.
e) Now the Schematic and the PCB are synchronized.

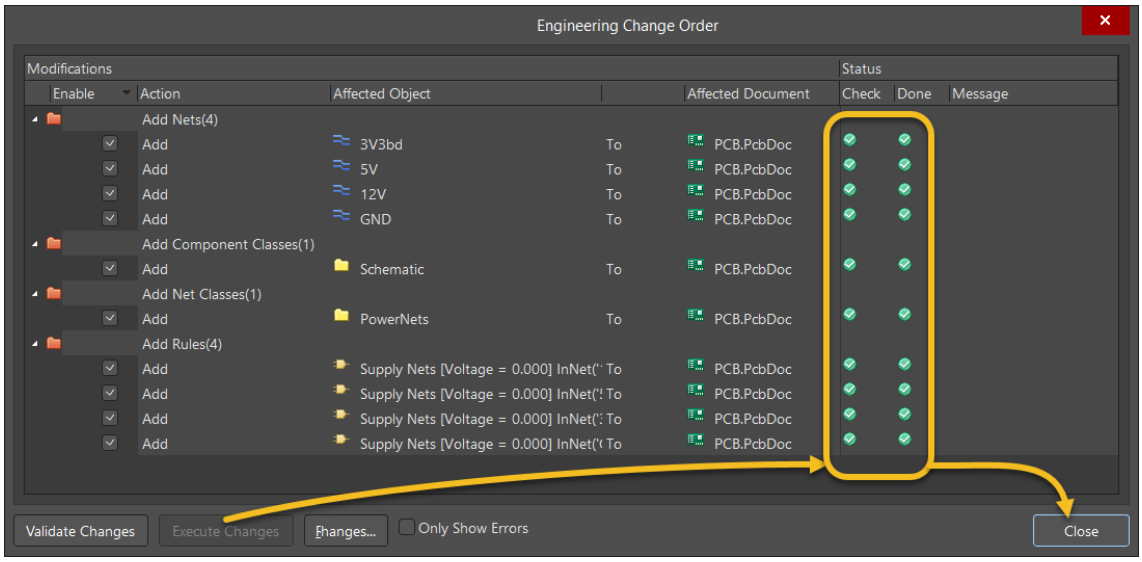
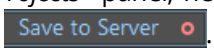


Figure 7. ECO

Note: The online DRC check will show DRC Violations for the PCB. You can ignore these violations for now. Later, when you place the Reuse Block in a new Design, it's important to check, and if needed, update the ERC and DRC (PCB Rules) settings.

6.4 Saving the Reuse Block to the Server

Next, you will upload the Design Reuse Block to the Workspace.

28. Save all documents using **File » Save All**.
29. Save the modifications to the server:
    a) In the *Projects* panel, next to the name **New Reuse Block**, you find the option **Save to Server** 

.
    b) Select **Save to Server**.
    c) In the dialog *New Reuse Block,* using Figure 8 as example.
        i) Add a Name: `Power Supply from WCT project – [Your Name].`
        ii) Add a Description for the New Reuse Block.
        iii) Check the Path. The default path for our Training Workspaces is `Managed Content\Design Reuse Blocks.`
    d) Select **OK** to save.

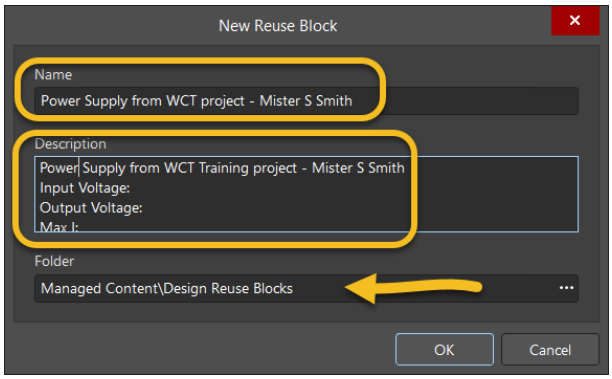


Figure 8. Save the New Reuse Block

30. While saving, you will see the same dialogs with progress bars. When the saving process is completed, the New Reuse Block branch will be automatically removed from your *Project* panel.
31. Close the Project `Creating and Using Design Reuse Block in a Workspace – [Your Name].`
32. Change your focus to the *Design Reuse* panel.
33. Your own Design Reuse Block, along with other Design Reuse Blocks from the other training attendees, will now be available for placement.

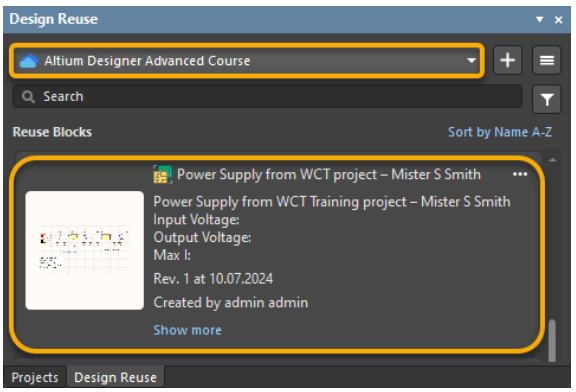


Figure 9. Design Reuse Panel with available Design Reuse Blocks

7 Using an Existing Design Reuse Block

Next, you will place a Design Reuse Block into a new Project.

7.1 Creating a New Project

34. Go to **File » New » Project...** to launch the *Create Project* dialog. The dialog shown in Figure 10 will appear.

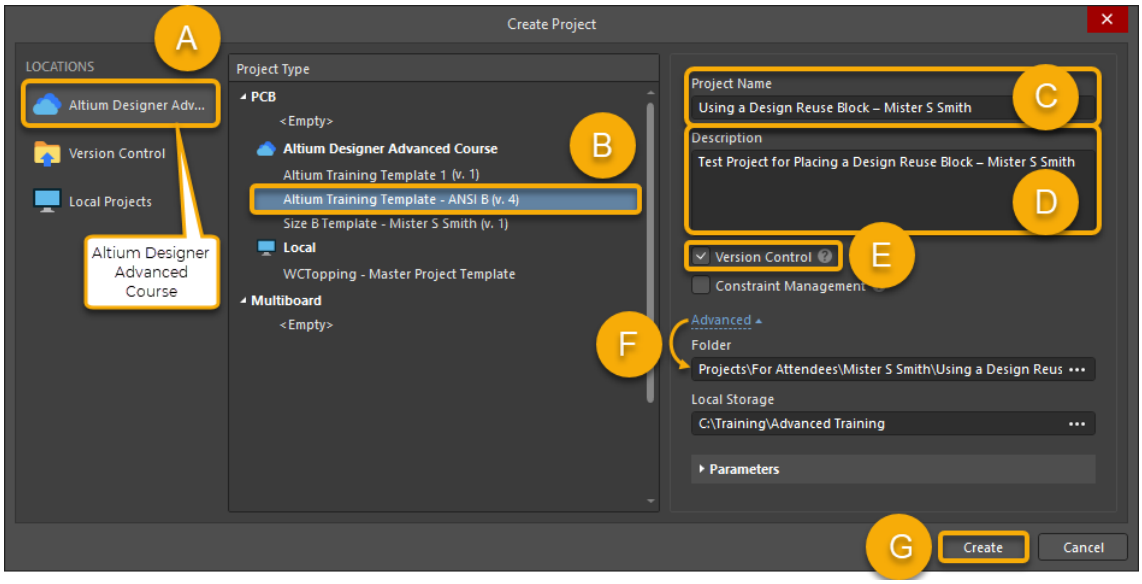


Figure 10. Create a New Project

35. Use Figure 10 above as a reference for the following steps. You may notice the **Constraint Management** option. We won't use this specifically for this project, but we will cover it in other modules of the Advanced training.
    a) Select the workspace `Altium Designer Advanced Course` in the *Locations* area.
    b) Select the **Altium Training Template - ANSI B** template.
    c) Enter the project name, for example, `Using a Design Reuse Block – [Your Name].`
    d) Add a Description, for example, `Test Project for Placing a Design Reuse Block – [Your Name].`
    e) Ensure that the Version Control Checkbox is enabled.
    f) Open the **Advanced Settings**. This is the configuration for where your project will be saved.
        i) *Folder*: Select the ellipsis ••• and chose the *Folder* `Projects » For Attendees » [Your Name Here].`
        ii) *Local Storage*: Check if the predefined *Local Storage* Path is okay for you. If not, feel free to change the path.
    g) Select the **Create** button to create the new project. It may take a few seconds for Altium Designer to create the new project.

36. Return to Altium Designer. In the *Projects* panel, you can see a project called `Using a Design Reuse Block - [Your Name].PrjPCB` with a schematic document, a PCB document, and additional documents, as shown in Figure 11 below.

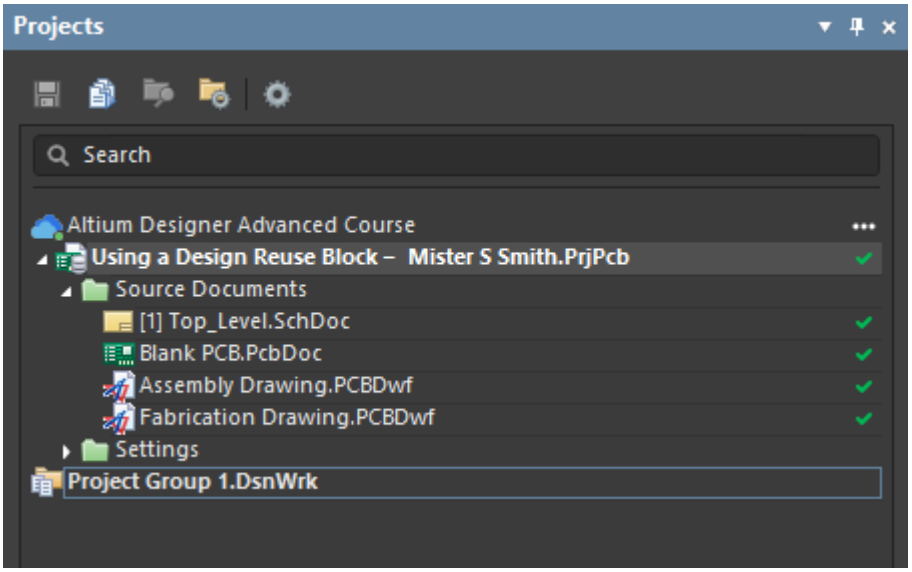


Figure 11. Projects panel with the new project

7.2 Using a Reuse Block

37. If not already open, open the `Schematic` from the *Projects* panel and ensure it's a focused document.
38. Open the panel *Design Reuse* by selecting the **Panels** button Panels and the *Design Reuse* at the lower right side of your workspace.
39. In the *Design Reuse* panel, Figure 12:
    a) From the drop down list, select the Training Workspace `Altium Designer Advanced Course`.
    b) From the list of available Reuse Blocks, select the Reuse Block you created (`Power Supply from WCT project – [Your Name]`).
    c) Place the predefined Circuit in the open schematic sheet. Select **Place...** and **Place...** again.

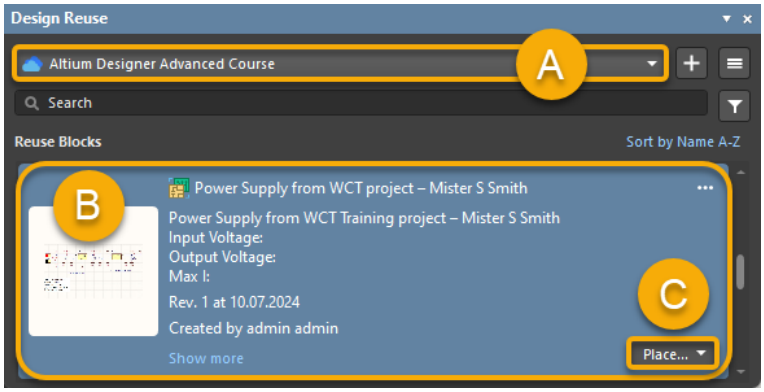


Figure 12. Design Reuse panel

40. Right-click the schematic in the *Projects* panel and click **Save**.
41. Execute the command **Design » Update PCB Document Blank PCB.PcbDoc**.
42. In the *ECO* dialog, click **Execute Changes.** Close the ECO dialog after the update was done.

With the PCB as active document, on the right side, you see the Footprints from the Schematic. For the PCB, the Design Reuse is now a **Union**, which means selecting and moving one Footprint will move all elements from the **Union**.

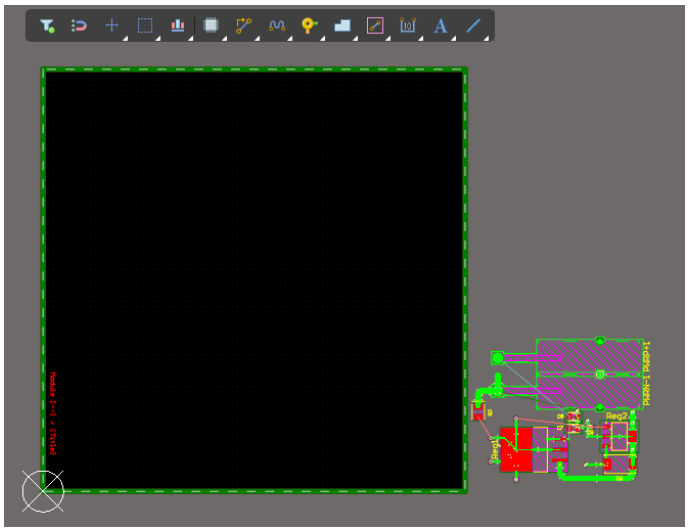
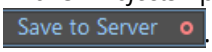


Figure 13. PCB with Design Reuse Block

43. Save all documents using **File » Save All**.
44. Save the modifications to the server:
    a) In the *Projects* panel, next to the Project name you find the command **Save to Server** 

.
    b) Select **Save to Server**.
    c) In the dialog *Save [Project Name]:*
        i) Add the comment `Creating and Using Design Reuse Block in a Workspace - [Add Your Name] - Finished`.
        ii) Select **OK**.
45. When ready, close the project and any open documents, **Window » Close All**.

Congratulations on completing the Module!

Creating and Using Design Reuse Block in a Workspace

from

Altium Designer Advanced Training with Altium 365

Thank you for choosing **Altium Designer**