



ALTium **365**

Altium Designer

Essentials Course - Altium 365

Module 21: PCB Component Placement

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 21: PCB Component Placement	3
1.1 Purpose	3
1.2 Shortcuts	3
1.3 Preparation	4
1.4 General Overview	5
1.4.1 Properties Panel	5
1.4.2 Selection Filter	5
1.4.3 Metric / Imperial system	5
1.5 Placing Mounting Holes	6
1.6 Placement of Components	8
1.6.1 Placing within an Area	8
1.6.2 Placing Individual Components	10
1.6.3 Placing a Design Reuse PCB element	10
1.6.4 Align Command	13
1.6.5 Cross Select Mode.....	14
1.6.6 Position Remaining Components	16

Module 21: PCB Component Placement

1.1 Purpose



This exercise will allow you to practice different techniques for placing components in the PCB. You will also learn specific shortcut keys to help you place components efficiently.

Component placement directly determines the degree of difficulty for routing and the performance of the final product. Component placement is also important when it comes to minimizing connection cross-overs.

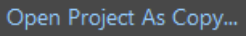

1.2 Shortcuts



Shortcuts when working with Module 21: PCB Component Placement

Space:	Rotate Counter Clockwise
Shift+Space	Rotate Clockwise
L:	Flip Layer
R:	Change Component Pushing Mode
N:	Net Line Connect Mode
T » O » C:	Reposition selected Components
T » O » I:	Arrange within rectangle
M » M:	Move
M » C:	Move Component

1.3 Preparation

1. **Close all existing projects and documents.**
2. Next, create a Copy / Clone of the Training Project Module 21 PCB Component Placement.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Navigate to the predefined Training Project Module 21 PCB Component Placement (Top\Projects\Altium Designer Essentials Training Course\...).
5. Select **Open Project as Copy...** .
6. At the new dialog *Create Project Copy*.
 - a) Add your name to the project: Module 21 PCB Component Placement - [Your Name].
 - b) Add a description: Altium Essential Training - Module 21 - [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.
7. Wait until Altium Designer created the copy of the project and opened the project for you at the *Projects* panel, this may take up to 1 minute.




For details how to Copy / Clone the predefined training project see Module 8 Making the Connection, Step 1.3 Preparation.

1.4 General Overview

1.4.1 Properties Panel

Before we start with placement, let's have a look at the *Properties* Panel.

8. Open the PCB file `Module 21 PCB Component Placement`. Notice all components are placed outside of the board.
9. You will notice, some component and routing are already configured, these are for the Power Schematic, based on the Design Reuse Block that was added to the schematics.
10. Open the *Properties* panel, select the **Panels** button  lower right side and choose **Properties**.
11. Go to the section *Board information*, the *Board Size* section shows the board size and area information:
 - a) Area: 16 inch²
 - b) Component Area: 6 inch²
 - c) Density: 18,8%



The Area for an individual Footprints is calculated automatically but the value can be overwritten in the PCB Library with the *Area* property.

1.4.2 Selection Filter

12. Be aware the pre-selection filter is pre-set to Vias and Components for this Module.
13. Feel free to change the position / size of Designators on layer `TOP Overlay`. To do so change the filter setting to allow the selection of Text.
14. Later you can re-set the filter in the *Active Bar*, as shown in Figure 1.

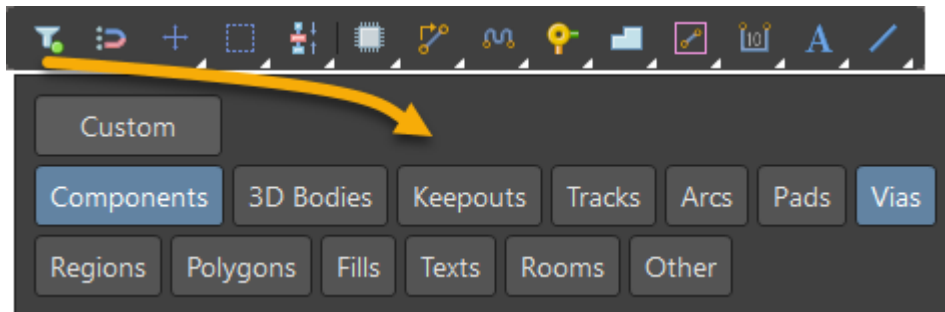


Figure 1. Enable All Objects in the Selection Filter

1.4.3 Metric / Imperial system

15. For the following steps we use metric and imperial values. Keep in mind:
 - a) You can change the PCB units of measurement using the **Q** key.
 - b) To change the units within a dialog, use the keys **CTRL+Q**.
 - c) It is also possible to add alternative units by adding `mm` or `mil` to the value.

1.5 Placing Mounting Holes

16. The training PCB needs four Mounting Holes.
Typically Mounting Holes are Components / Footprints, placed from a Library. For the training we are placing Vias as Mounting Holes, to show you later a specific situation.
17. Open the command **Place » Via**, a Via is attached to your mouse.
18. Press **Tab** key to open the *Properties* panel. Configure the Via with the following parameters, as seen at Figure 2
 - a) Diameter 6.2mm
 - b) Hole Size 2,75mm
 - c) Press **Enter** to go back to the placement mode for the Via.

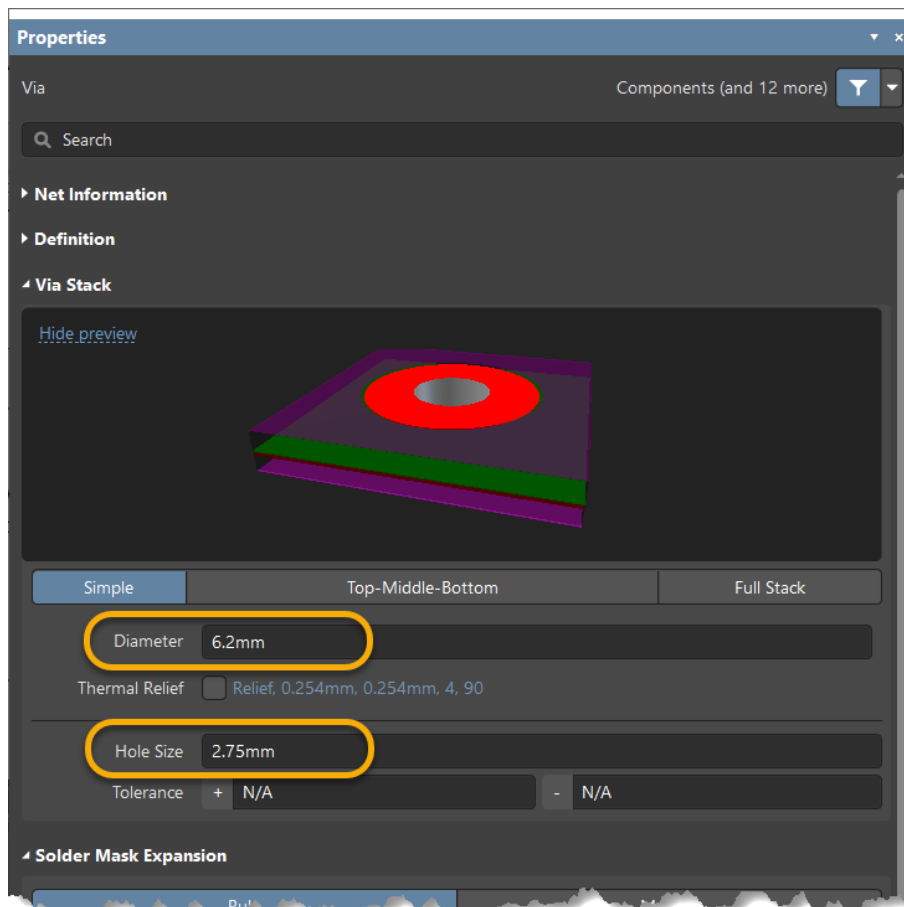


Figure 2. Via as MH Configuration

19. Position four Vias:
 - a) With the first Via attached to your cursor start the **Jump To Location** command with **J » L**.
 - b) Type in the values for the first position and use the **Enter Key** 2x (twice) to #1 jump and #2 position the Via
 - c) Repeat a) and b) for the remaining three Vias.
 - X: 94.593mm Y:53.98mm
 - X: 36.593mm Y:53.98mm
 - X: 36.593mm Y: 4.98mm
 - X: 94.593mm Y: 4.98mm

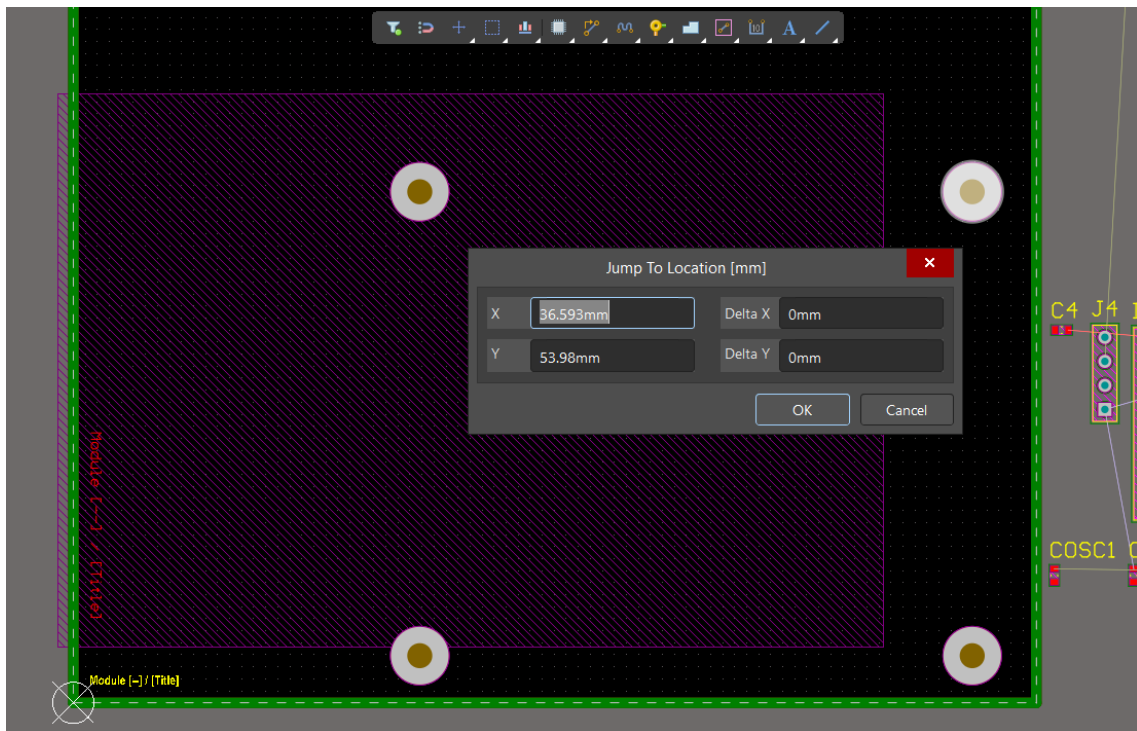


Figure 3. Place Vias with Jump To Location command

20. Select the four Mounting Holes.

- Open the *Properties* panel, select the Panels button **Panels** lower right side and choose *Properties*.
- Tick the **Lock** icon next to the coordinates to lock the position of the four Mounting Holes.

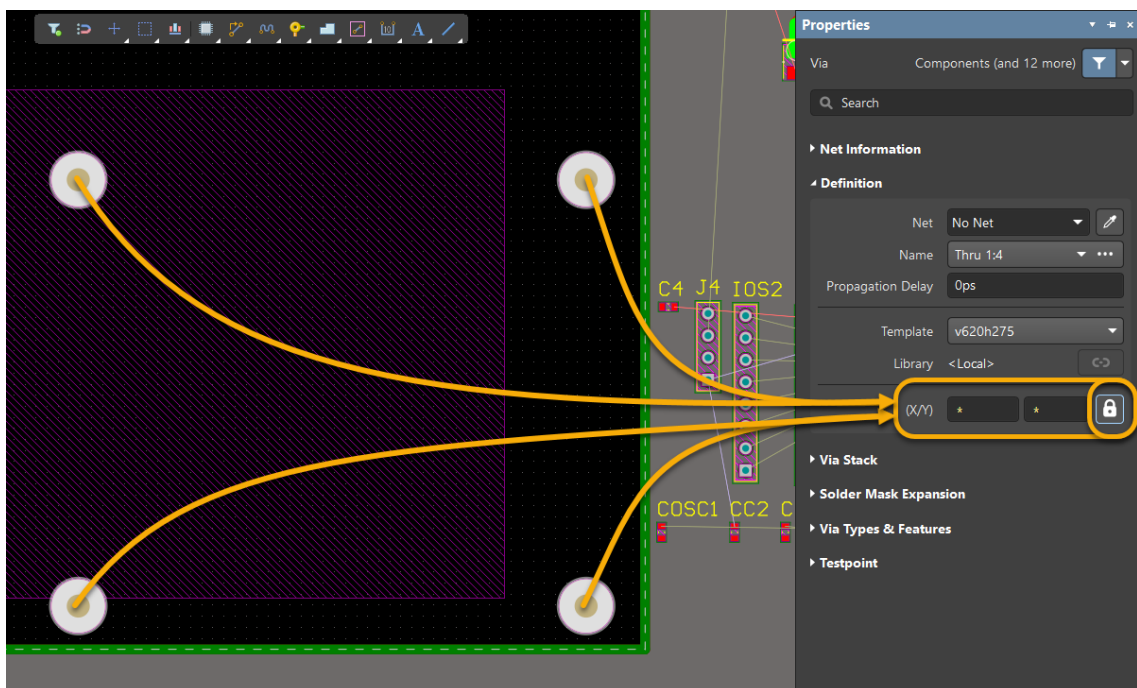


Figure 4. Lock for the four Mounting Holes

1.6 Placement of Components

1.6.1 Placing within an Area

21. Select a Group of components in the design by drawing a rectangle around them, Figure 5.
22. Use the command **Tools » Component Placement » Arrange within Rectangle**.
 - a) Define a rectangle of the entire board outline by:
 - i) First, left-click in the upper left corner of the board.
 - ii) Next, left-click in the bottom right corner of the board.
 - iii) Right-click to exit command.
 - b) Compare your result with Figure 5 below. This illustrates how you can get any selected group of components within a specific area of your board to begin placement.

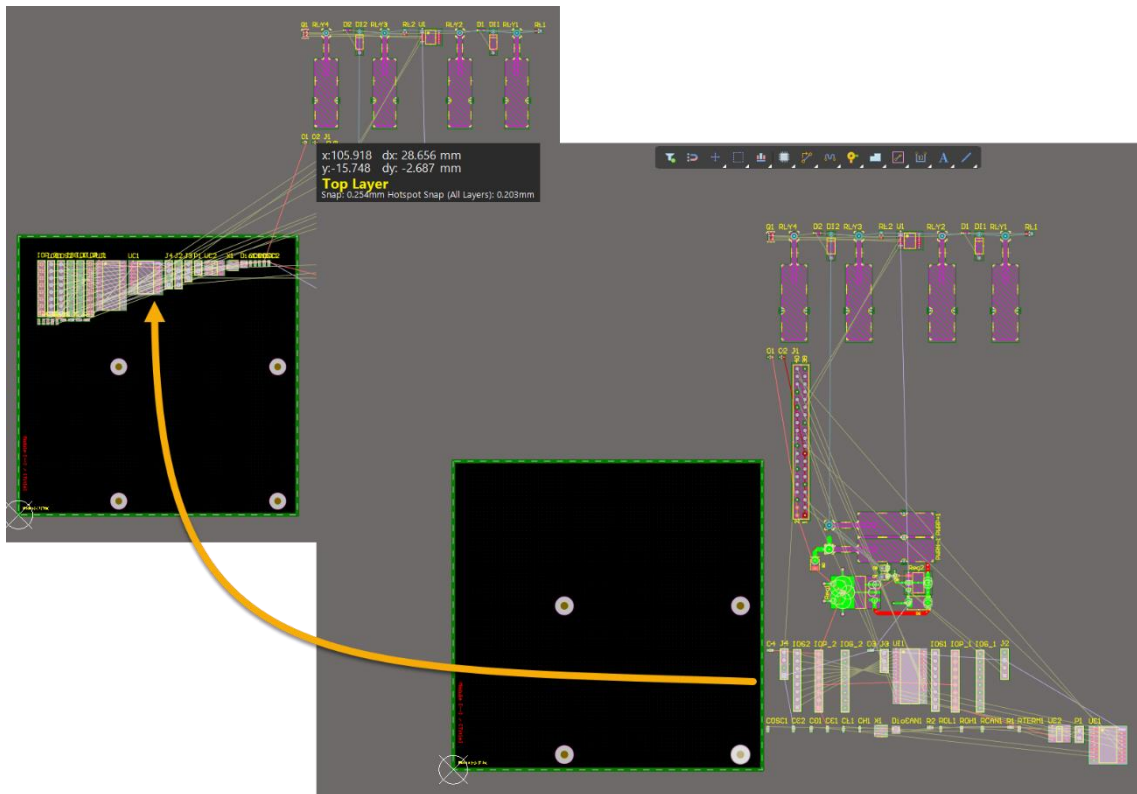


Figure 5. Component Placement within the board area

23. We will now position all the components outside of the board.
 - a) With all the components still selected, click **Tools » Component Placement » Arrange Outside Board**.
 - b) Hit **Shift+C** to clear the current component selection. See Figure 6 to see the end result.

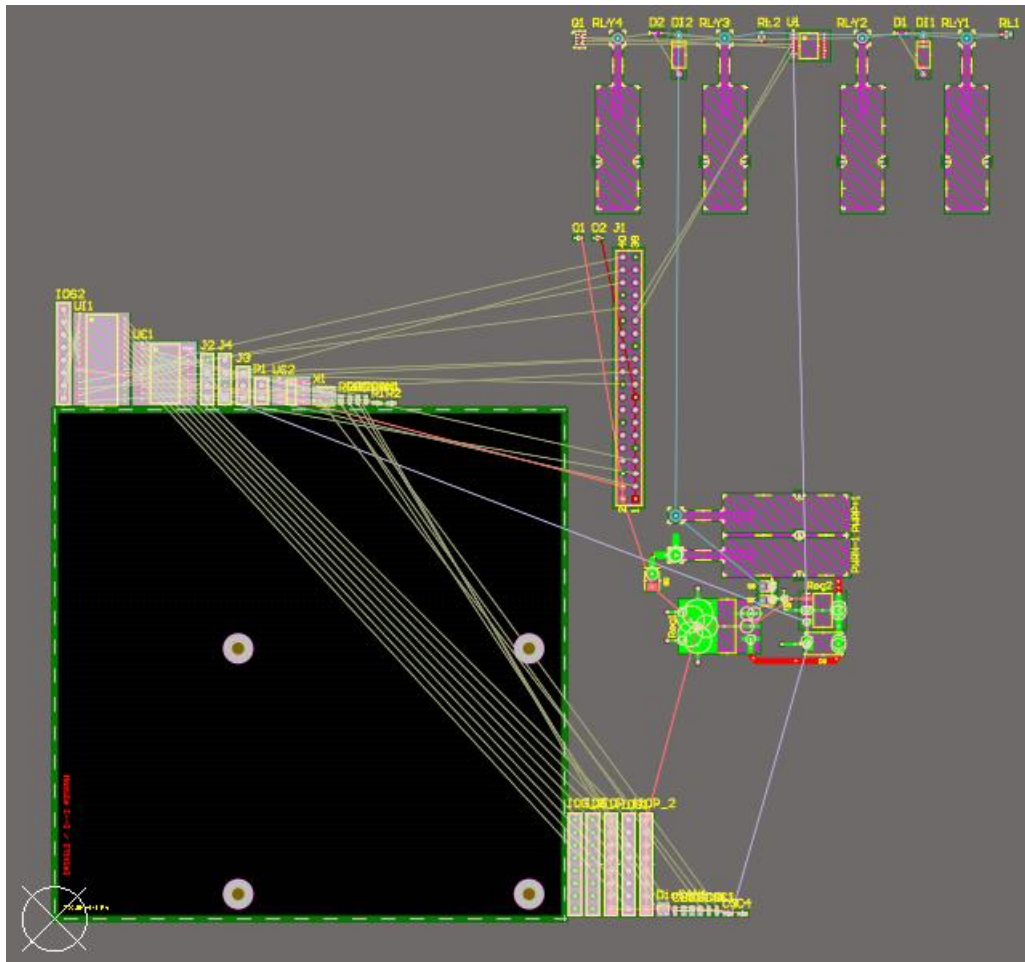


Figure 6. Components Arranged Outside of the board

1.6.2 Placing Individual Components

Just dragging is a not very precise method but it is okay for some components. Interfacing components, such as headers or connectors, often require specific locations.

24. Click on the 40-pin header, component J1,
25. At the *Properties* panel add the coordinate x: 3525mil y: 265mil and a rotation of 180° to position the component, as shown in Figure 7.
26. Lock the position of the component, similar to what you did for the Mounting Holes.



Figure 7. Place J1 within the board area by coordinates

1.6.3 Placing a Design Reuse PCB element

During the schematic creation we placed a Design Reuse element, the Power Supply WCTopping reuse block.

With the Design Update command Altium Designer pulled the PCB part of that Power Supply WCTopping reuse block and placed it, similar to all other components, at the right side of the blank PCB.

The Reuse Block is a combination of components, vias and tracks. In order to keep together the placement and routing of the Reuse block, Altium Designer creates a Union.

Unions allow designers to group different objects together, such as components, vias, tracks, or polygons. This grouping simplifies the manipulation and organization of related elements



Connection lines will appear when dragging a component to show the same net connections between the footprints. While dragging a component, press **N** to toggle the visible component net connections ON/OFF. To hide all connection lines, use **View » Connections » Hide All**.

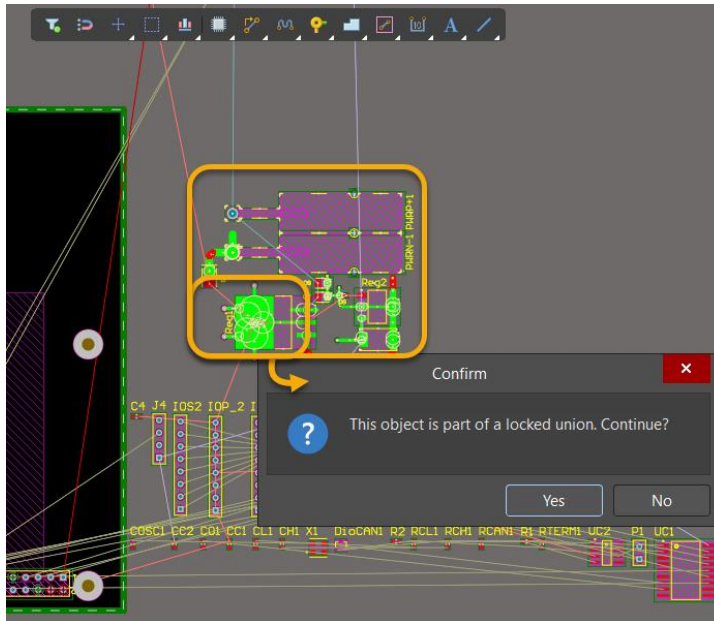


Figure 8. Reg1 member of the Union Power Supply WCTopping reuse block

27. Use the **PCB Panel** or **Jump » Component (J»C)** to find Reg1 .
28. Select Reg1 and start moving Reg1 by pressing and holding the left mouse button.
29. A message to confirm that you want to continue the move command pops up, accept with YES, Figure 8.
30. Position the Power Supply WCTopping reuse block inside the Board, use Figure 9 as reference.
31. Pads and Tracks of the Union Power Supply WCTopping reuse block are highlighted green, indicating a DRC errors. With the **Design » Update PCB ...** command Altium Designer could assign the net information to the Pads, but the Tracks of the Union are without a net information.
32. To fix that, to add a net information to the tracks, execute the command **Design » Netlist » Update Free Primitives From Component Pads...**, Confirm with YES is asked.

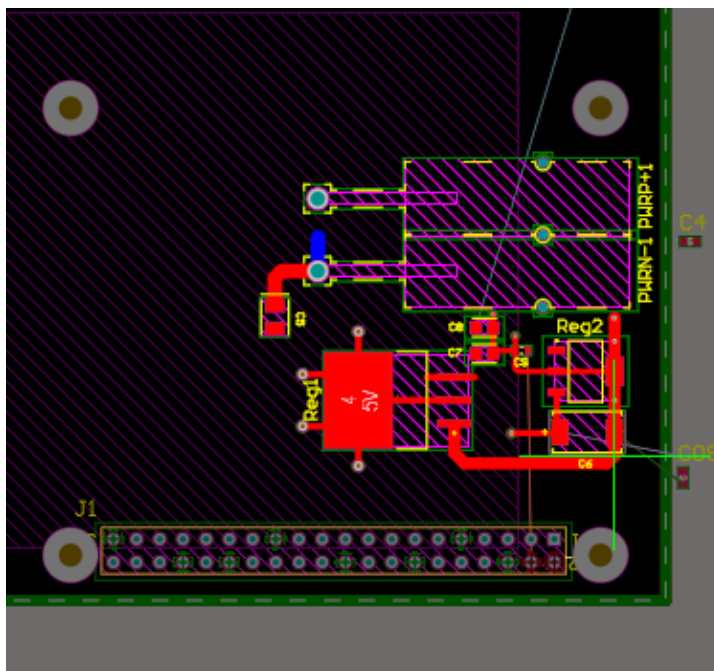


Figure 9. Placed Power Supply WCTopping reuse block

33. Continue placing components using the methods below, while using Figure 10 as a reference:

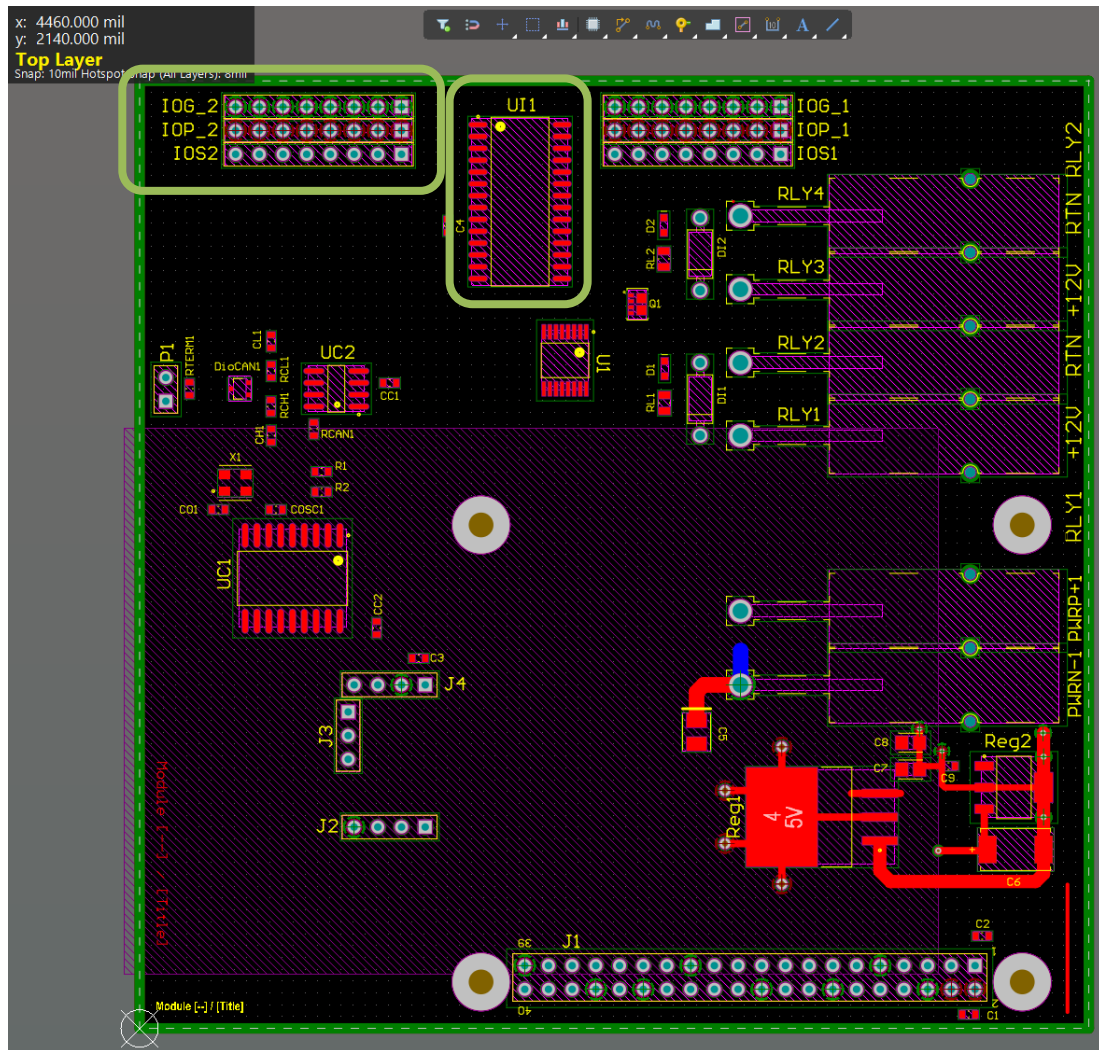


Figure 10. Placement for WCTopping training PCB

- a) Use the **Move** command using the **Edit » Move » Move (M, M)** hotkeys and left-click on component **IOG_2** and move to its new position as shown in Figure 10.
 - b) Rotate **IOG_2** with **Spacebar** if needed.
 - c) Still in the Move command select **IOS2** and **IOP_2** and place them below of **IOG_2**, as seen at Figure 10.
 - d) Right-click to exit this command when you're done placing the component.
34. Continue placing components using the methods below, while using Figure 10 as a reference:
- a) Use the Move command using the **Edit » Move » Move (M, M)** hotkeys and left-click on component **UI1** and move to its new position as shown in Figure 10.
 - b) Rotate **UI1** with **Spacebar**. Right-click to exit this command when you're done placing the component.
 - c) Hold the left mouse button to select and drag a component. The component will remain on your cursor and will be placed at the desired location once you let go of the left-mouse button.



While moving a component, press **Spacebar** to rotate the component counter-clockwise and press **Shift + Spacebar** to rotate the component clockwise.

1.6.4 Align Command

Next, position components RLY1, RLY2, RLY3 and RLY4 so that the result looks similar to Figure 10.

35. Select components RLY1, RLY2, RLY3, and RLY4 and position the Anderson Connectors as seen at Figure 11 A.

The position does not need to be exact. Position the four Anderson connectors in a way that they are more to the left as the 2 Anderson connectors already placed with the Union Power Supply WCTopping reuse block, Figure 11 A.

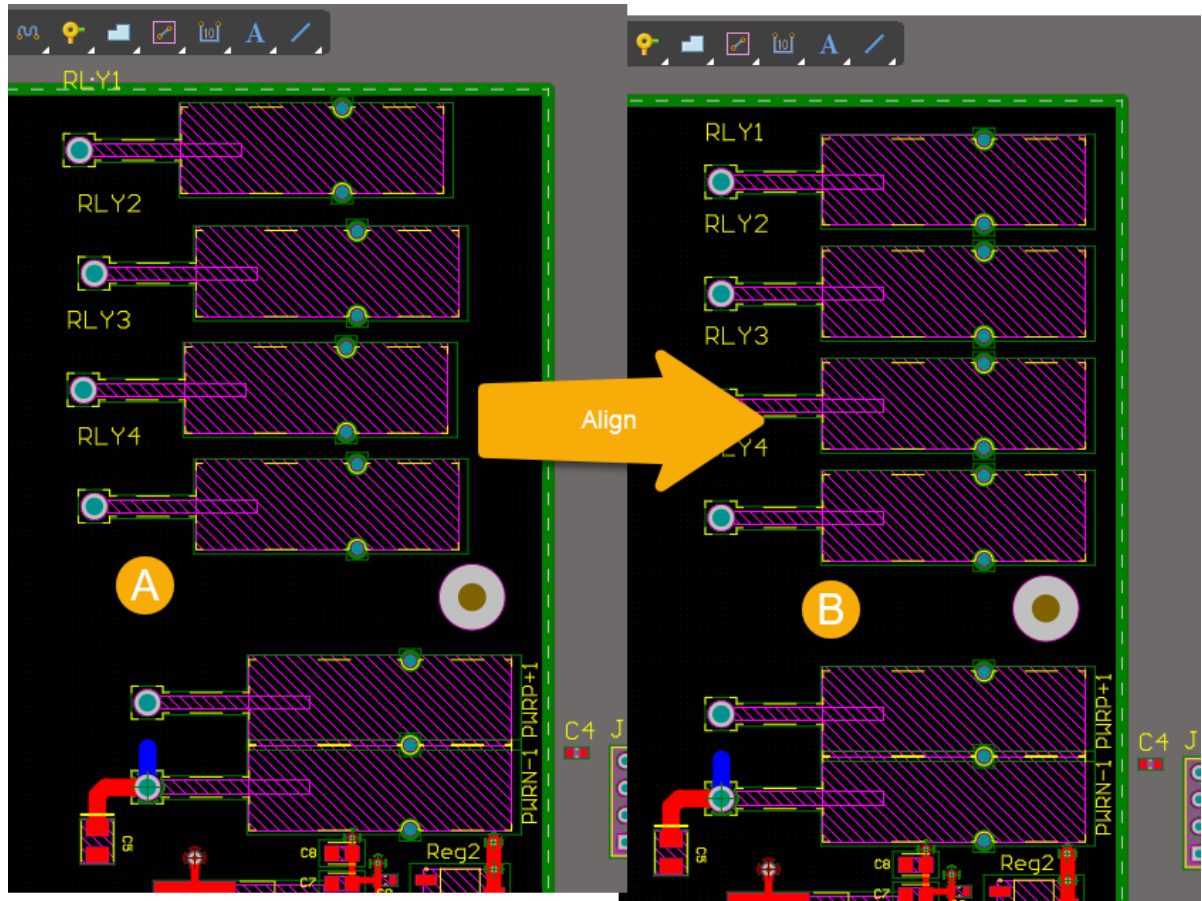


Figure 11. First placement for Anderson RLY1-RLY4 and the final position after using the Align commands

36. We're now going to align the Anderson components:
- Select components RLY1, RLY2, RLY3, RLY4 and PWRP+1 and PWRP-1
 - Use the command **Edit » Align » Align Right**. The 4 components RLY1, RLY2, RLY3, RLY4 are moved to the right, with PWRP+1 and PWRP-1 as reference.
 - Select components RLY1, RLY2, RLY3, RLY4
 - Use the command **Edit » Align » Distribute Vertically**. The components should now be Vertically spaced out evenly.
 - If needed, you can Increase / Decrease the distance between the components with **Increase Vertical Spacing** and **Decrease Vertical Spacing**
 - Your result should be similar to Figure 11 or Figure 10




Alignment and move commands can be performed on component properties, such as Designators.

1.6.5 Cross Select Mode



Tools » Cross Select Mode used in conjunction with the PCB Editor's **Tools » Component Placement » Reposition Selected Components** command is a very effective way to place components by selecting them from the schematic.

37. Open the  *Preferences*.
38. From the *System* branch, on the *Navigation* page, verify that the **Cross Selection** checkbox is enabled under *Cross Select Mode*, Figure 12. Click **OK** to exit the *Preferences*.

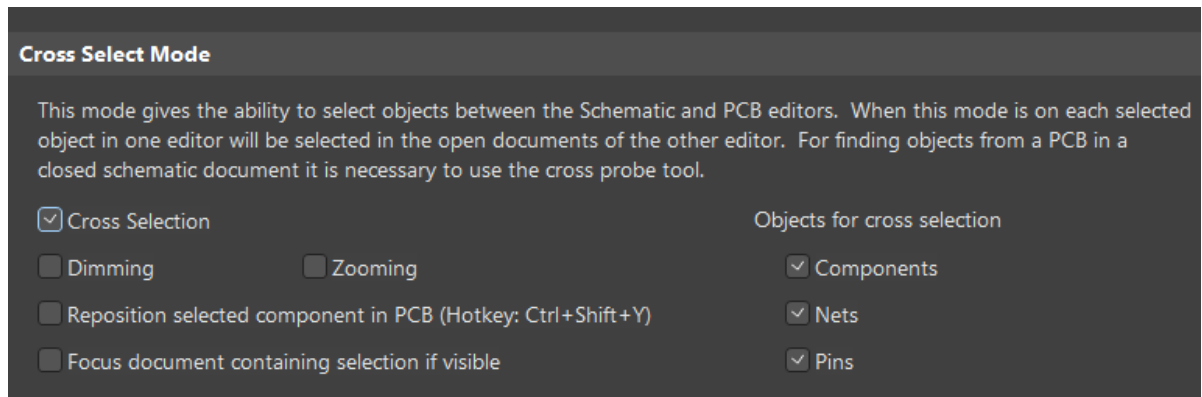


Figure 12. Cross Select option

39. Enable Cross Select Mode from **Tools » Cross Select Mode**. If the icon has a blue outline around it, then Cross Select Mode is already enabled as shown in Figure 13.

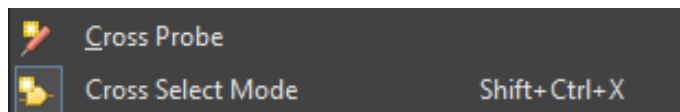


Figure 13. Blue highlighted icon indicating Cross Select mode is enabled

40. In the `Processor_Interface.SchDoc`, select capacitors C1 and C2 in order using **Shift+Select** as shown in Figure 14.



Be sure to click on the body of the capacitor rather than just the designator or comment. The component will not be selected in the PCB if the body of the symbol is not selected.
Change the Grid (**G** key) if it is difficult to select the body from the capacitor.

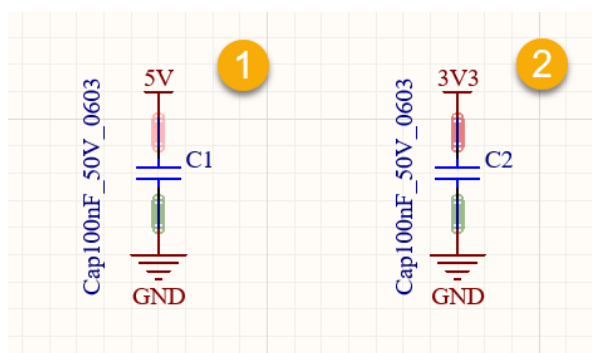


Figure 14. Selecting Capacitor symbol bodies



If you select the components with a rectangle the order for the next step, PCB placement, may be different. You can use **S** during placement to move the current component back to virtual Component stack.

41. Make the **PCB File** the focused document.
42. In the PCB Editor, make sure the corresponding footprints are currently selected at the top of the PCB document.
43. Now, go to **Tools » Component Placement » Reposition Selected Components**. Component C1 should automatically jump to your cursor for placement.
44. Click to place C1 in the bottom right corner as shown in Figure 15, using the **Spacebar** to rotate.
45. Component C2 should now automatically jump to your cursor. Place it in accordance with Figure 15.

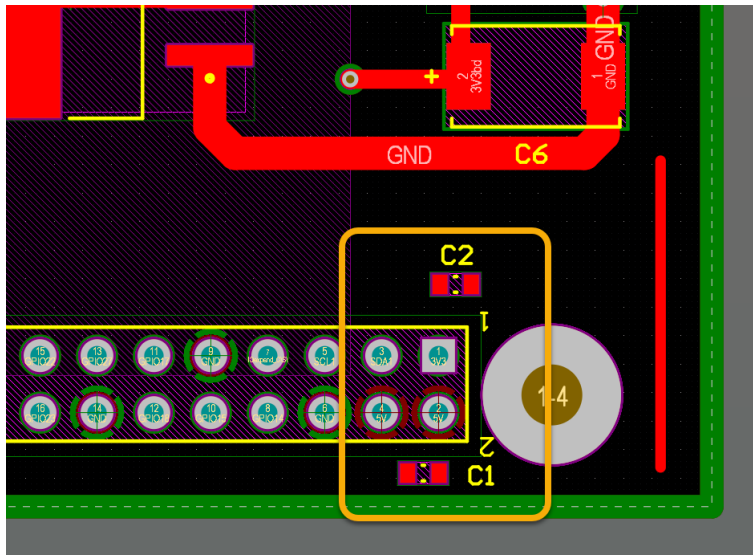


Figure 15. Positioning components C5 and C6 loosely

46. Now we will position the leftmost capacitor exactly in the position we want.
 - a) Double click on capacitor, C1 to open its *Properties* panel dialog.



Depending on the Preference setting **Double Click Runs Interactive Properties** (PCB Editor – General) instead of the Properties panel the Polygon Pour dialog could be opened.

- b) Change the (X/Y) position to (88.8mm, 1.5mm) as shown in Figure 16.
 - c) Press **Enter** to accept the change.

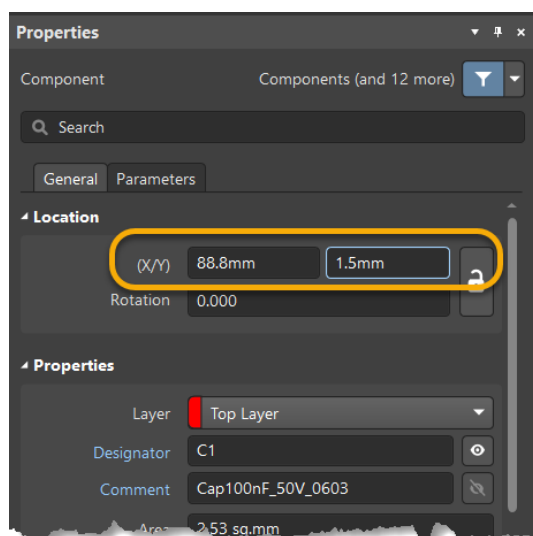


Figure 16. Placing C1 at a specific location on the board

1.6.6 Position Remaining Components



Please bear in mind due to time constraints, we do not need to place all the components, just move the components Classes to specific areas of the PCB, and after the training you could start the real placement. The next module will contain the fully place board, ready for routing.

47. To place some of the remaining components:

- a) Use click and move method.
 - i) Click in the center of a component and hold down the left mouse button.
 - ii) The component will now move with your cursor.
 - iii) Release the mouse button at the desired location.
- b) Refer to Figure 17 and the following instructions for the placement of components from the *PCB* panel:
 - i) Open the *PCB* panel.
 - ii) Set the Drop-down at the top to **Components**.
 - iii) For drag and drop, in the select boxes at the top of the panel, uncheck **Select** and check **Clear Existing**. Enabling the **Zoom** option will zoom into the selected component.
 - iv) Click on the *Component Class* `U_CAN Interface` or `<Outside Board Components>`.
 - v) All class components / outside board components will be listed in the *Components* pane.
 - vi) Directly from the *Components* section in the *PCB* panel, click on a component and drag it directly into the PCB board area.
 - vii) Release the mouse at the desired location.

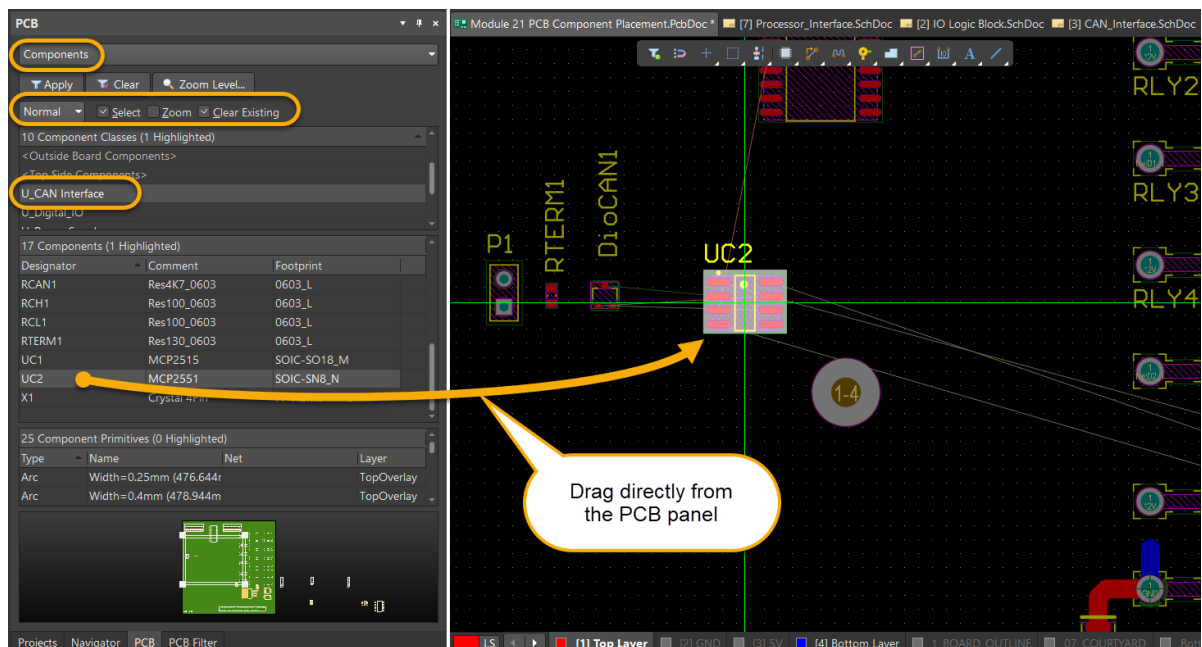


Figure 17. Placing components from the *PCB* Panel

- c) Placing multiple components from the *PCB* panel:
 - i) Try selecting multiple components from the *Components* pane using **Ctrl + Click**.
 - ii) Drag one of the selected components into the PCB.
 - iii) After you place the first component, the rest of the selected components will be on your cursor, ready to place.

48. For the remainder of the components, in the *PCB* panel, enable the **Select** checkbox at the top of the panel.
49. Select the <Outside Board Components> class.
This will select the remaining components outside the board area.
50. Go to **Tools » Component Placement » Reposition Selected Components** to sequentially place the rest of the components.
51. Place 5 or 6 footprints for practice, then right click to escape the *Move* command.
52. Refer to Figure 18 as a sample of a PCB component placement for this training module.
53. Feel free to change the position / size of Designators on layer *TOP Overlay* during placement. To do so change the filter setting to allow the selection of Text.

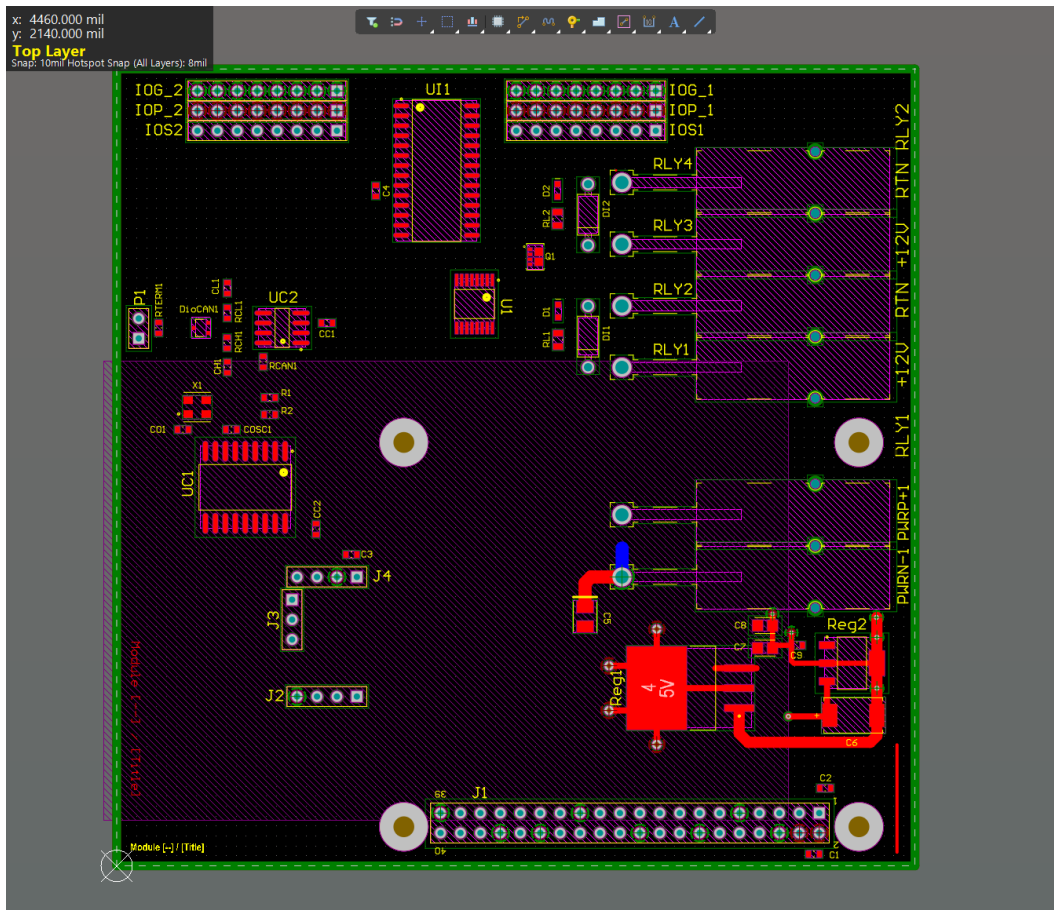
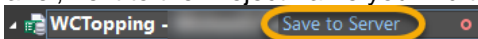


Figure 18. Possible end result of the PCB layout during Training.

54. Save all documents using **File » Save All**.
55. 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) Activate the checkboxes for the files that are not under version control.
 - ii) Add the comment `Module 21: PCB Component Placement - [Add Your Name] - Finished`.
 - iii) Select **OK**.
56. When ready, close the project and any open documents.

Congratulations on completing the Module!

Module 21: PCB Component Placement

from the

**Altium Designer Essential Course
with Altium 365**

Thank you for choosing Altium Designer