



# Altium Designer

Advanced Training with Altium 365

Multi-Board Design

**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>Multi-Board Design</b>	<b>3</b>
<b>1 Purpose</b>	<b>3</b>
<b>2 Important notice</b>	<b>3</b>
2.1 Overview	3
<b>3 Shortcuts</b>	<b>4</b>
<b>4 Preparation</b>	<b>6</b>
<b>5 Capture the Logical System Design</b>	<b>8</b>
5.1 Setting up a Multi-Board Schematic Document	8
5.2 Working with Connections	12
5.2.1 Enabling PCB Projects for Multi-Board Design.	12
5.2.2 Populating Modules with design data by importing the project data	13
5.3 System Design Updates	16
5.3.1 Making a change to Child PCB Projects, Part 1	16
5.3.2 Making a change to Child PCB Projects, Part 2	18
5.3.3 Updating the Multi-Board project from child PCB projects changes	18
5.4 Validating the Logical Design	19
<b>6 Creating the Physical Multi-Board Assembly</b>	<b>22</b>
6.1 Overview	22
6.2 Pushing MB Schematic data to your MB Assembly File	23
6.3 Positioning your Multi-Boards within MB Assembly	24
6.4 Applying a Mate to Multi-Boards	27
6.5 Multiboard Assembly Panel	30
<b>7 Multi-Board Creation of Drawings</b>	<b>31</b>
7.1 Creating a Manufacturing Drawing	31





# Multi-Board Design

## 1 Purpose

In this exercise, you will learn how to connect multiple boards together within Altium Designer. This Multi-Board Design involves the physical connection of two separate PCB projects to form a Bicycle LED Head Lamp Multi-Board Assembly.

Note: Abbreviation MB will be used in place of complete word Multi-Board.

Caution: This Module needs a Professional or Enterprise License. For details, please see our [Online Documentation](#).

## 2 Important notice

### 2.1 Overview

Advanced electronic product designs are generally composed of multiple PCB designs that are interconnected to create a complete, functional system. From a design with a main board and a front panel LCD to a complex mother board system with plug-in modules, all are implemented as a system of multiple board designs.

This requires a high-level design system that allows multiple 'child' PCB designs to be electrically and physically connected while maintaining the integrity of their Pin and Net connectivity. Integrated system-level design is supported by Altium Designer in the form of a dedicated Multi-Board Design environment that features the logical (schematic) and physical (PCB) aspects of system design.



Figure 1. Concept of a Multi-Board





## 3 Shortcuts

Shortcuts used when working with Multi-Board Design

### **Common shortcuts:**

CTRL+S	Save Document
--------	---------------

For other common shortcuts not specific to Multi-Board Design, please see link to [Altium Designer Shortcut Keys](#).

### **Multi-Board (MB) Schematic Editor & Board Assembly Editor Shortcuts:**

Ctrl+PgDn	Changes the view in editor window so document is full visible
-----------	---

PgUp	Zoom-in, relative to current cursor location
------	--

PgDn	Zoom-out, relative to current cursor location
------	---

Mouse Wheel	Scroll vertically within the design space
-------------	---

Shift+Mouse Wheel	Scroll horizontally within the design space
-------------------	---

Right-Click	With action of hold and drag, will pan the view within design window
-------------	--

Q	Toggle between Imperial and Metric units for the active Multi-Board
---	---

Shift+E	Toggle the snapping functionality on or off
---------	---

F11	Toggle the display of the Properties panel
-----	--

### **Multi-Board (MB) Board Assembly Axis / Orientation Control Shortcuts:**

Z or Blue axis	Re-orient board movement control looking down the Z axis
----------------	--

Shift+Z	Re-orient board movement control from other side of Z axis
---------	--

X or Red axis	Re-orient board movement control looking down the X axis
---------------	--

Shift+X	Re-orient board movement control from other side of X axis
---------	--

Y or Green axis	Re-orient board movement control looking down the Y axis
-----------------	--

Shift+Y	Re-orient board movement control from other side of Y axis
---------	--

Crtl+WheelRoll	Zoom in/out
----------------	-------------

Right-Clk and Drag	Displays the panning hand cursor, moves board in current plane
--------------------	--

Ctrl+PgDn	Zoom the view to fit all boards including origin markers
-----------	--

Shift and Right-Click and Drag	Rotate the board from current plane
--------------------------------	-------------------------------------



**Multi-Board (MB) Working with Mates Shortcuts:**

Shift+Ctrl+A	Switch to Mating mode. The cursor will highlight potential mate sites
Shift+Space	Rotate Source part of mate clockwise. (whole board will rotate)
Space	Rotate counterclockwise
Up Arrow	Increase distance of mate source board to other board along
Down Arrow	Decrease distance of mate source
Ctrl+Rht Arrow	Increase the offset of the Source part in X direction of mating
Ctrl+Lft Arrow	Decrease the offset, X direction
Ctrl+Up Arrow	Increase the offset, Y direction
Ctrl+Dwn Arrow	Decrease the offset, Y direction
Ctrl+T	Toggle the orientation of the source part in relation to mated
Delete	Remove the current selected mate


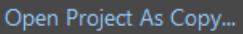





## 4 Preparation

1. Close all existing projects and documents.
2. Next, you will create a copy of the Training Project: Strt-Bicycle\_LED\_HeadLamp.PrjMbd

Note: The Strt-Bicycle\_LED\_HeadLamp.PrjMbd is a Multi-Board type of project. Multi-Board projects use the extension PrjMbd.

3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Enable the folder view button .
5. Navigate to the predefined Training Project Strt-Bicycle\_LED\_HeadLamp].PrjMbd (Projects\Altium Designer Advanced Training Course\01 Multiboard\Strt-Bicycle\_LED\_HeadLamp).
6. Select **Open Project as Copy...** .
7. In the new dialog *Create Project Copy*:
  - a) Add your name to the project: Strt-Bicycle\_LED\_HeadLamp - [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.
9. Repeat steps 1 - 8 above, two more times for two more projects that will be required.
  - a) For the first repeat, substitute name Strt-Bicycle\_LED\_HeadLamp with Bicycle\_Head\_Lamp\_MainBrd.
  - b) For the second repeat, substitute Strt-Bicycle\_LED\_HeadLamp with Bicycle\_LED\_Board.





10. Ensure your *Project* panel looks like Figure 2 below. You should have three projects:

- a) Strt-Bicycle\_LED\_HeadLamp - [Your name].PrjMbd
- b) Bicycle\_Head\_Lamp\_MainBrd - [Your name].PrjPcb
- c) Bicycle-LED\_Board - [Your name].PrjPcb.

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

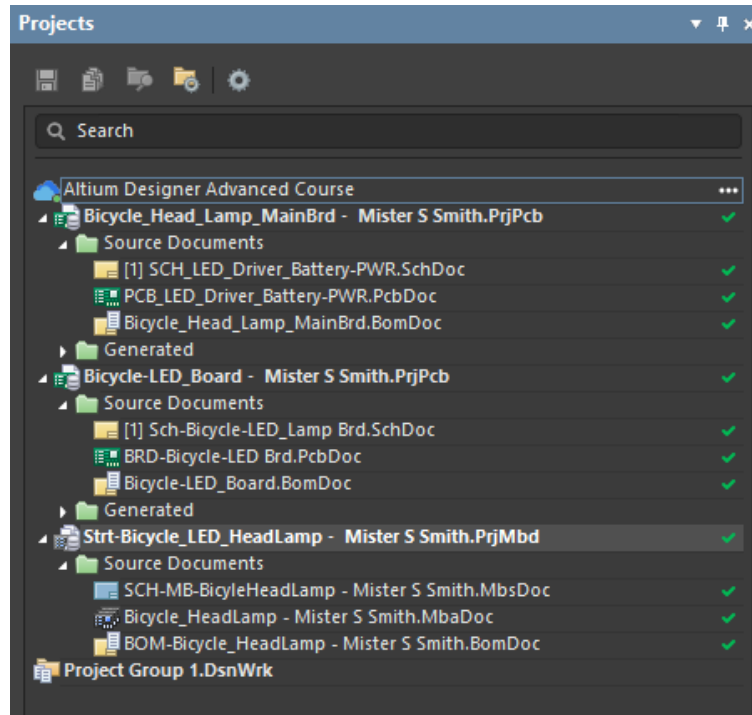
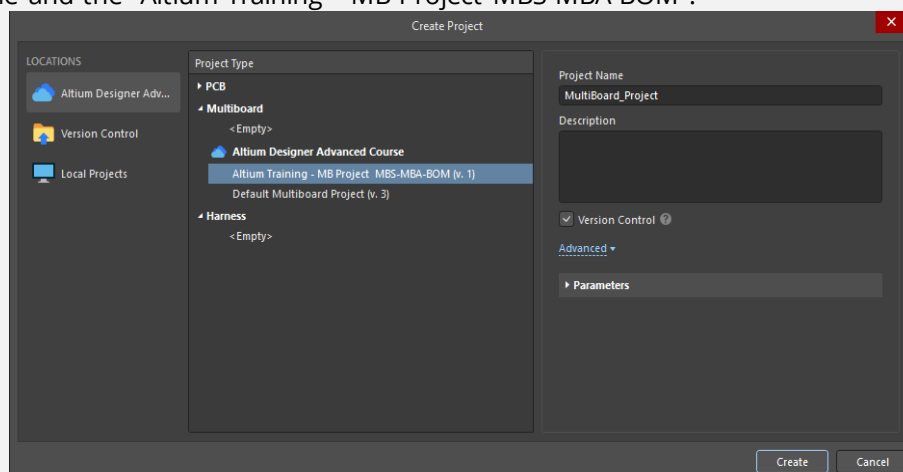


Figure 2. Project Panel showing all three projects

Note: In this training, you copy an existing MB project and rename the files to create a new training MB Project. Altium also supports the MB project template. Instead of creating a copy from an existing MB project, you can create a new MB project using a project template as a starting point.

By using a **File » New » Project** menu, you see a dialog that allows you to select a MB project template and add a project name. The training workspace offers two MB templates: the default one and the “Altium Training - MB Project MBS-MBA-BOM”.





## 5 Capture the Logical System Design

### 5.1 Setting up a Multi-Board Schematic Document

Creating a Multi-Board system Schematic design from multiple child PCB projects includes the following steps:

- 1) Creating a Multi-Board project, adding a Multi-Board schematic document to the project.
- 2) Placing graphic blocks (Modules) in the schematic to represent each child project.
- 3) Linking each Module to its appropriate child project.
- 4) Importing the child project connectivity data into the system level design.
- 5) Adding connections between Modules to create the logical system design.

Figure 3 illustrates a Multi-Board defined project representing a Multi-Board schematic MiniPC.MbsDoc and Multi-Board assembly MiniPC.MbaDoc files.

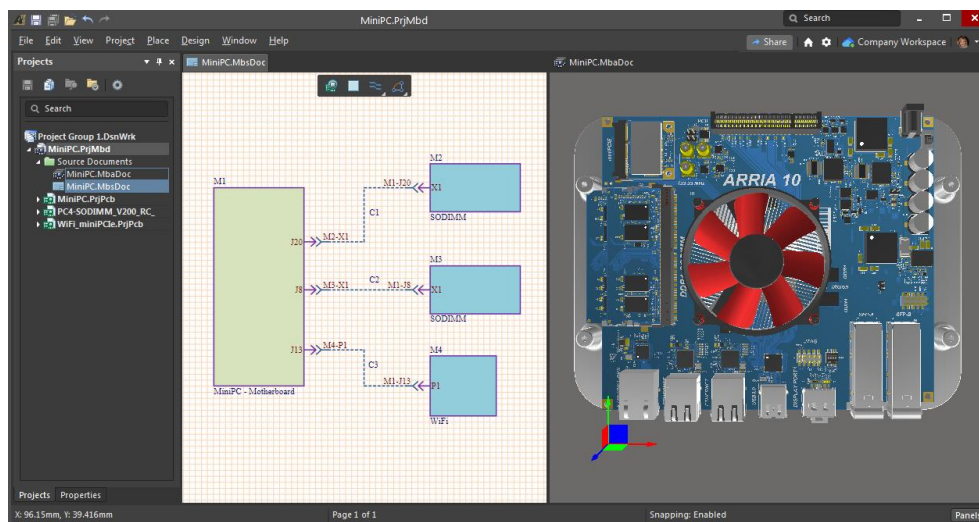


Figure 3. Multi-Board Project, Actual Exercise image will be different

11. Let's begin our Multi-Board project. First, close only the two board projects in the *Project* panel with the PrjPcb extension.

12. Your project tree should look like Figure 4.

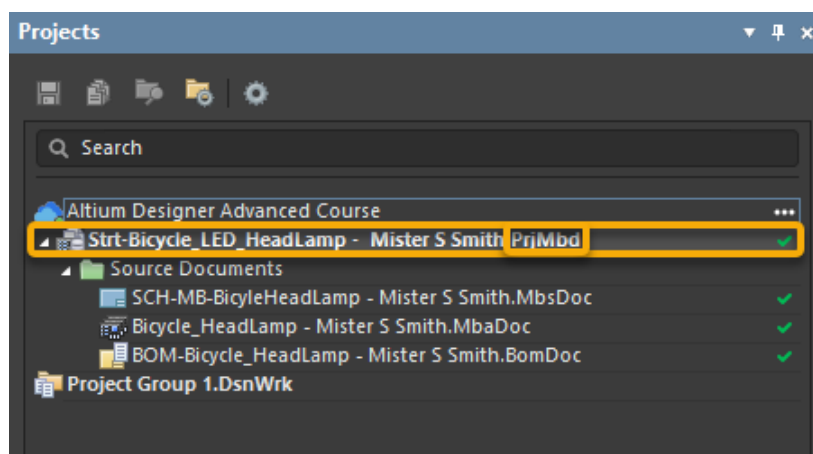


Figure 4. Project Panel with remaining PrjMbd project

13. From *Project* panel, with mouse cursor over schematic named `Schematic.MbsDoc`, **right-click** and select **Rename**. Rename the schematic to `SCH-MB-BicycleHeadLamp - [Your Name]`.
14. Rename the MB Assembly file to `Bicycle_HeadLamp - [Your Name]`.
15. Rename the ActiveBom file to `BOM-Bicycle_HeadLamp - [Your Name]`.
16. Save the files by clicking on the blue **Save to Server** on the project name in the *Project panel*. In the *Save* dialog, include a meaningful *Comment*, then click on the **OK** button. See Figure 5 below.

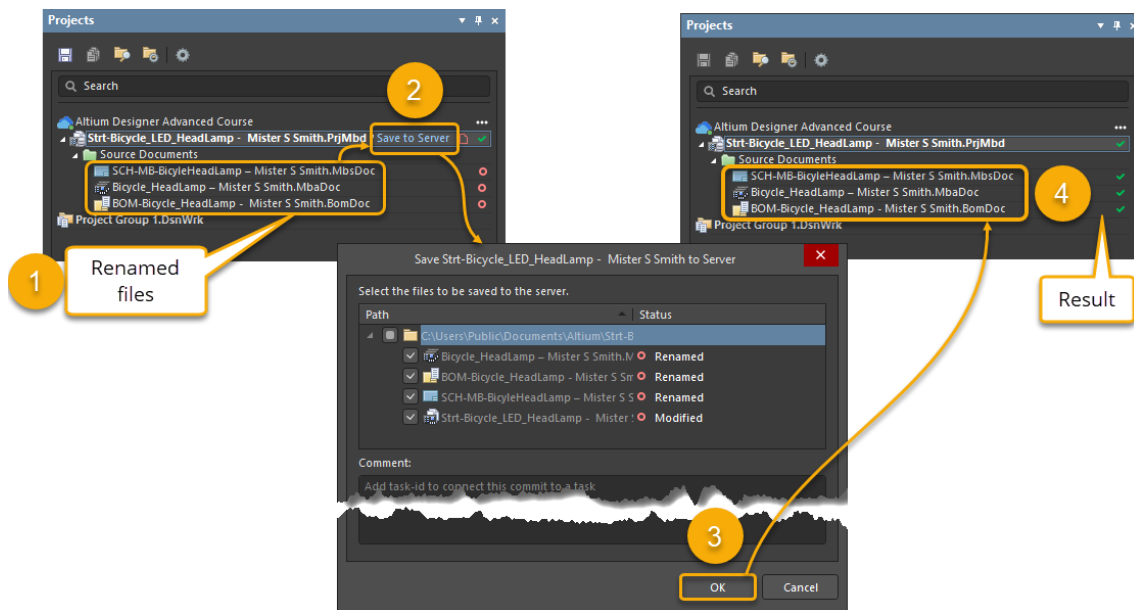




Figure 5. Saving to Server, adding comment, then pressing OK

Note: All red circled icons  will turn to a green check  after saving.

Multi-Board Projects begin with placement of Modules in the MB Schematic.

17. Open the MB Schematic `SCH-MB-BicycleHeadLamp-[Your Name].MbsDoc`. Graphical blocks placed on the schematic as Modules will represent each child project to define the Multi-Board logical system design foundation. From the tool bar seen in Figure 6 below. Click on the **Module Icon**. Cursor will change to small cross-hairs.

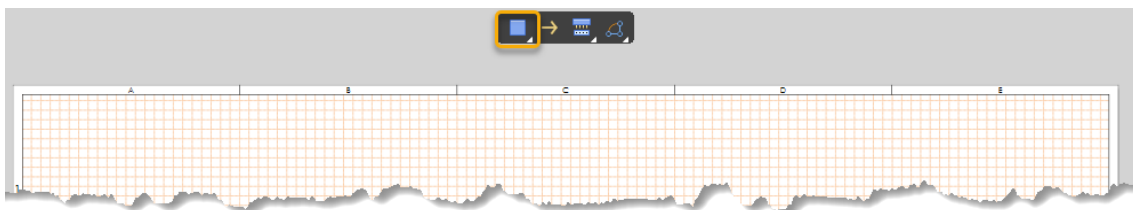


Figure 6. MB Schematic Active Tool Bar

18. With single left mouse click, draw a rectangle from upper left starting point and end with second mouse click hold at the lower right corner of the finished rectangle.
19. Complete a second smaller rectangle in parallel, but to the right of the first rectangle. End the module placement mode with a right mouse click in open schematic space.



20. The two rectangles should resemble Figure 7 below.

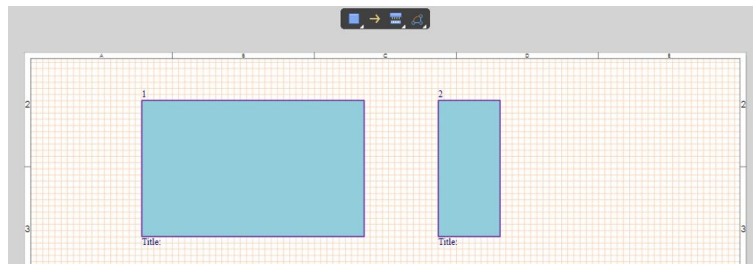


Figure 7. Modules added to MB Schematic

21. Let's configure the properties for the first module placed on the schematic and identified as 1. Open the Properties panel with the mouse double left click on the left module.

22. Configure the properties for this module, using Figure 8 as reference.

- a) **General Tab – Properties:** Type in *Designator* field M1 and *Title* as LED-Drv\_Brd\_PWR-SPLY.
- b) **General Tab – Source:** Left-Mouse click on the **Ellipsis Button** ... and select the *Bicycle\_HeadLamp\_MainBrd-[Your Name] Project* you uploaded to the workspace. Notice the Source Project field populates with the project name, Revision as (Latest) and the Assembly/Board automatically populates as the *PCB\_LED\_Driver\_Battery-PWR.PcbDoc* from the selected Child Project.
- c) The *Message* pane may pop up with some information. Ignore the *Message* pane for now and return to the *Properties* panel.
- d) **General Tab – Graphical:** We will tune the Width to 71.1mm and Height to 50.8mm. Click on the Fill color and change if you like.

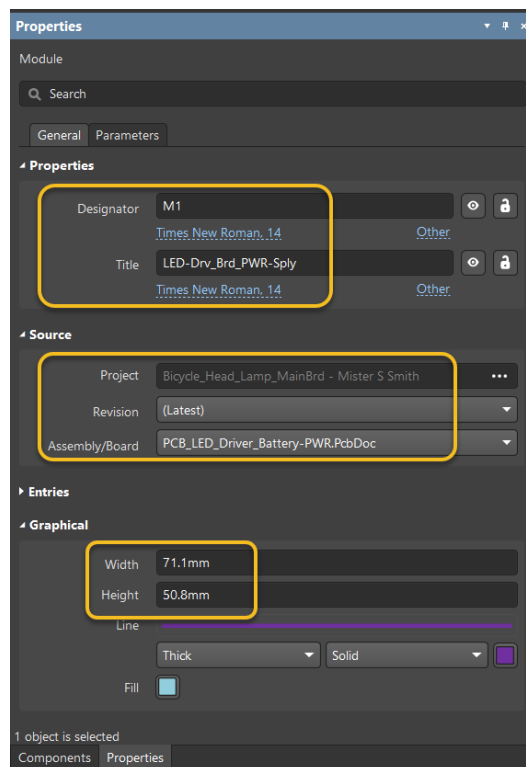
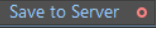


Figure 8. Module M1 Properties





23. Repeat step 22 for the second, smaller module, including the following modifications:
  - a) As done in step 22 a), a designator is M2 and a title is LED\_Cluster Brd.
  - b) As done in step 22 b), the project selected is Bicycle-LED\_Brd-[Your Name].
  - c) As done in step 22 d), Width is 800mil (~20mm) and Height is 2000mil (~50.8mm).
24. As you have done before, select the blue **Save to Server**  on project name.
25. Your configured modules on the MB Schematic should look like Figure 9.
26. The *Project* panel was updated with the new information. The two PCB Projects are now included in the project structure, as sub projects from the MB Project. Figure 10.

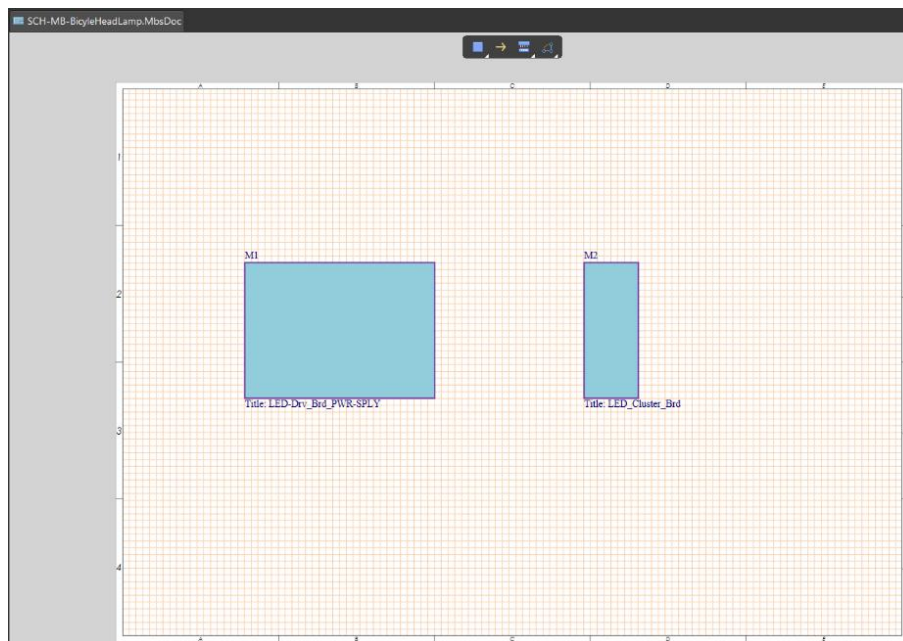


Figure 9. Configured Modules on MB Schematic

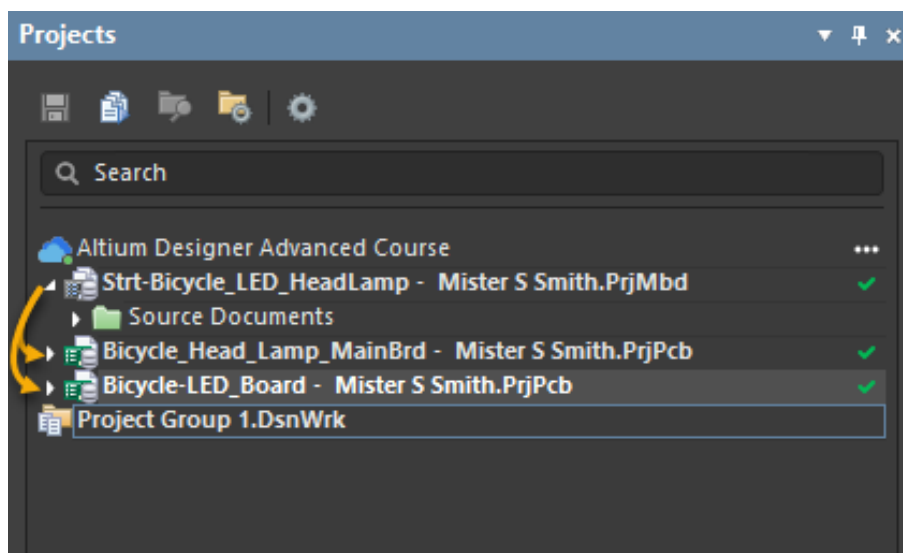


Figure 10. Project Panel with MB project and the two PCB projects



## 5.2 Working with Connections

The connections established between modules in the schematic ultimately represent the connectivity between child project connectors, connector pins, and nets in the overall system design. The Multi-Board schematic editing environment provides comprehensive features that may be used to define, modify, check, and update the connectivity as the overall product design is developed.

Before working with connections on your Multi-Board Schematic, you need to enable the specific connections in your child PCB projects with a specific component parameter for the PCB project connectors – a parameter named **System**. `Connector` used as a parameter value, will be given to the appropriate external connections to each child PCB project in this Multi-Board Design.

### 5.2.1 Enabling PCB Projects for Multi-Board Design.

To simplify this exercise, all Multi-Board external connections have already been configured with the parameter setting noted above.

27. For this exercise, we will explore the external connectors for the required parameter setting noted above and shown in Figure 11.

- Let's open the schematic `SCH_LED_Driver_Battery-PWR.SchDoc` from child project `Bicycle_Head_Lamp_MainBrd-[Your Name]` seen from the *Project* panel.
- The schematic is visible. There's only one connector for this schematic, represented as J1 in two parts, as J1A and J1B.
- Double-click one J1 to open the *Properties* panel.
- Scroll from the right parameters table to search for parameter **System** with value `Connector`. Activate **Show More** [Show More](#) to see the full list, if needed.

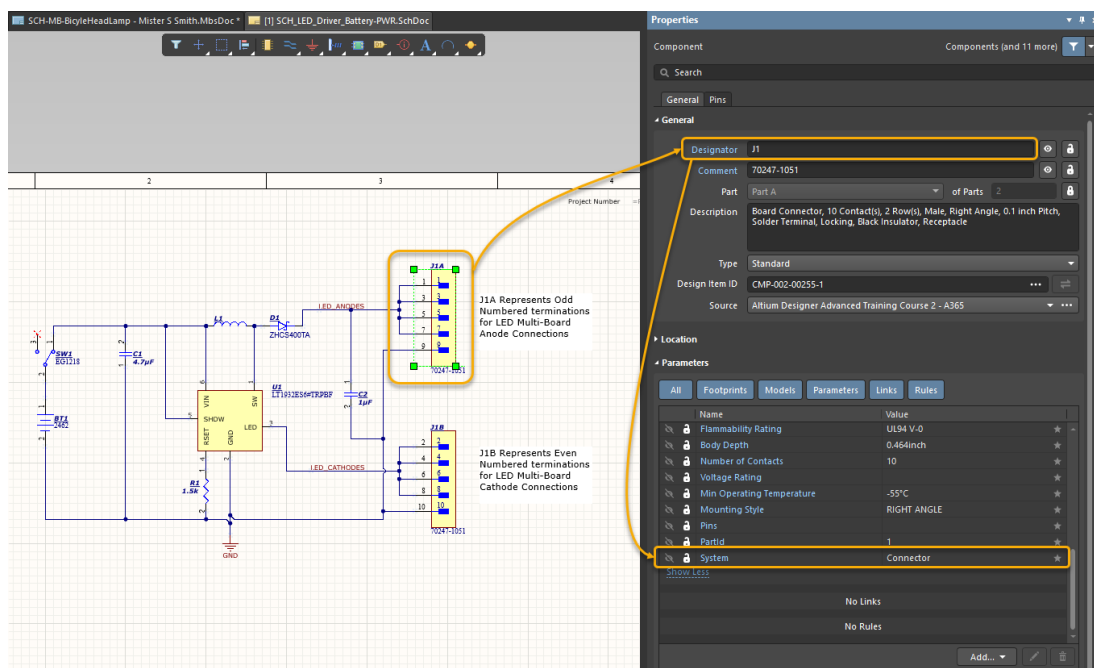




Figure 11. Example of System parameter equal to Connector

Hint: If the System parameter is missing, click on the button **Add**, then **Parameter** from the pop-up menu. A default Parameter1 name is shown at the bottom of the parameter list. This would be renamed as System. The \* character in the Value field would be changed to name Connector.



Hint: Parameters marked as Favorites are shown in the Parameters list, while those not marked are hidden. Hover to the right of a parameter filter name and click the  icon to set the filter as Favorite. To remove the parameter from the Favorites, click the  icon again.

## 5.2.2 Populating Modules with design data by importing the project data

The Modules are populated with design data from their linked child PCB project designs by importing the project data. In the exercise steps below, you will perform the import process required to provide data for the modules in the MB-Schematic.

28. Make the MB schematic SCH-MB-BicycleHeadLamp - Mister S Smith the active document.

- a) Select from **Design » Import from Child Projects** to import data from the child PCB projects associated to the modules M1 and M2 in the Multi-Board schematic, Figure 12.

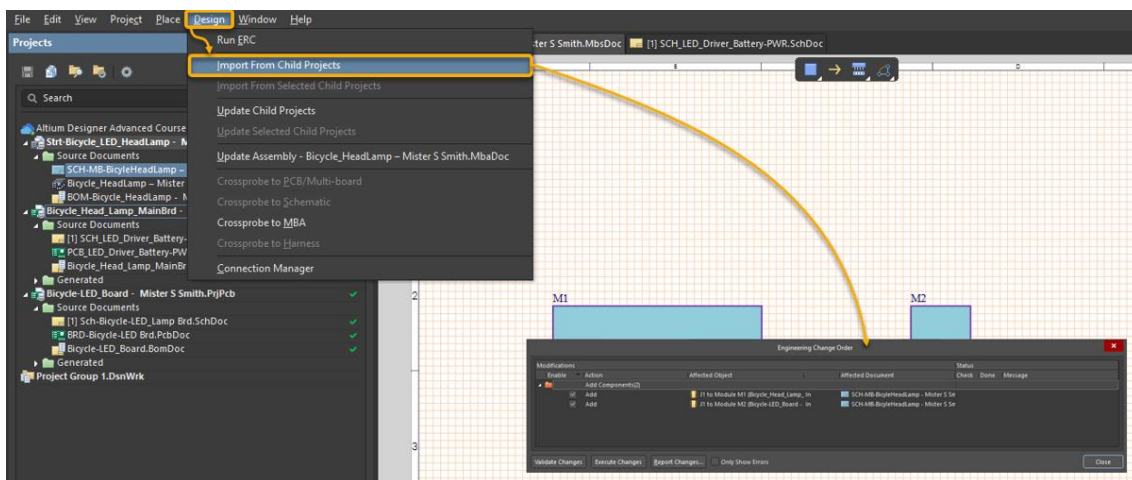


Figure 12. ECO Completed

- b) The *Engineering Change Order* panel is displayed.
- c) Click on the **Validate Changes** button. Confirm green checks under status column *Check*.
- d) Click on the **Execute Changes** button. Confirm green checks under status column *Done*.
- e) Feel free to click on the **Report Changes...** button. Confirm the presence of the Report Preview from the ECO initiated. You can print the report to a printer or PDF for later review.
- f) Click on **Close** to close the *ECO* dialog.
- g) The Multi-Board schematic is updated with external connectors by reference designator from the respective child PCB projects. See Figure 13 on the next page to see an updated MD Schematic SCH-MB-BicycleHeadLamp - [Your Name].

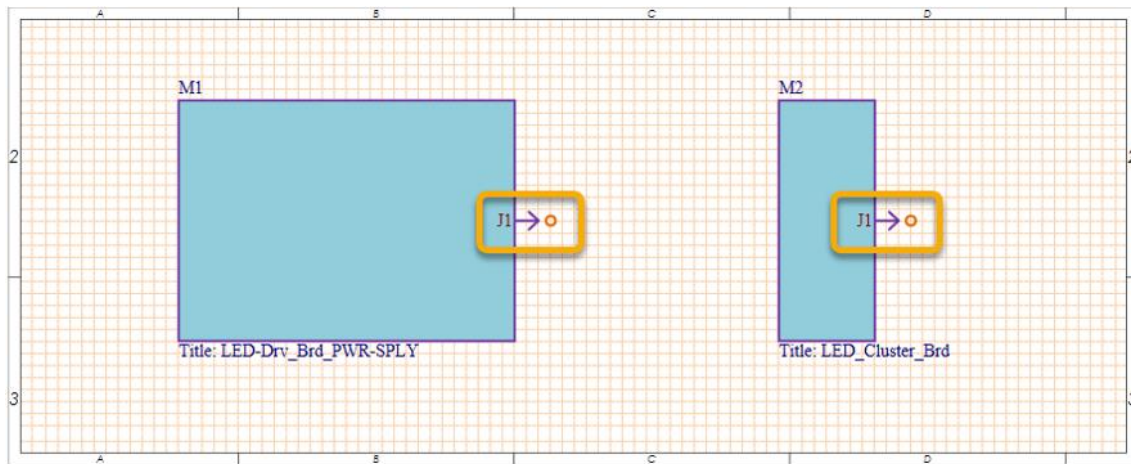
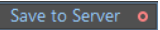



Figure 13. Multi-Board schematic with imported connections

29. Click the blue **Save to Server**  text on the project name from the *Project* panel. Saving files should become a frequent habit and won't be mentioned further in this exercise.
30. Use left mouse click with hold on M2-J1 and drag it around the module boarder to face M1-J1 on the same center line. Release mouse button after moving.
31. Let's complete the connections by connecting the modules together. From the active tool bar within the Multi-Board schematic editor, click on the **Add Direct Connection** button  with a left mouse click.
32. The cursor changes to the add connection crosshairs. Starting with M1-J1, left mouse click on the orange wire node. Then, drag wire connection line to terminate at the M2-J1 with a second left mouse click.
33. End direct connect mode with a right mouse click within the schematic editor workspace.
34. If needed, drag M2 or M1 to increase the distance between the two modules.
35. Your completed Multi-Board Schematic should look like Figure 14 below.

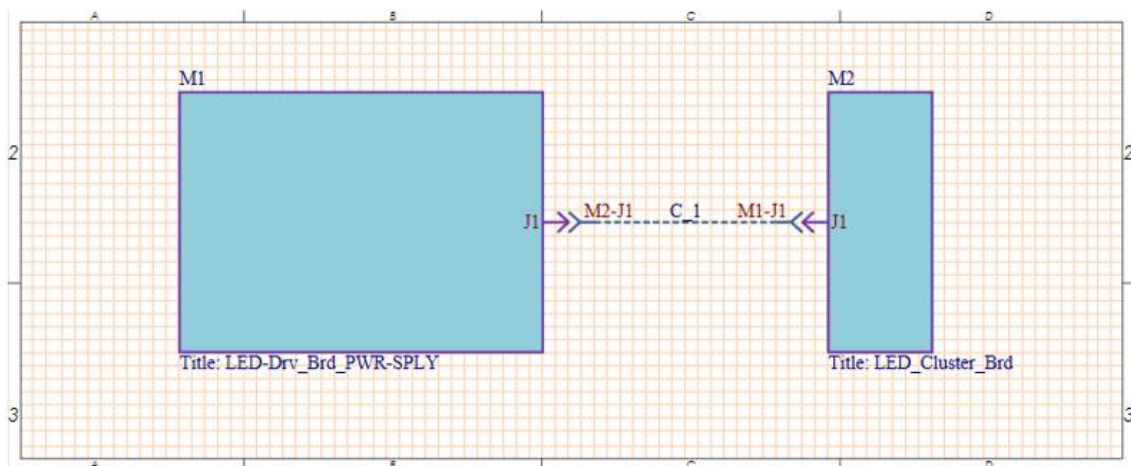


Figure 14. Completed MB Schematic with connected modules





36. Let's explore our connections. With the MB Schematic still open, left mouse click on the wire between the M1-J1 and M2-J1. Then, open the *Properties* panel.

a) **Properties – General Tab – Properties:**

Shows the designator for the connection wire as C\_1. Later, you can change the Designator, if needed. *Number Of Connections* shows as 10.

b) **Properties – General Tab – Connections:**

Shows the connection distribution table between connectors from each Module that represents each child PCB project for all 10 connections.

c) **Properties – General Tab – Graphical:**

Allows you to change the color of the connected wire and choose your wire style. You can change it, if needed.

d) **Properties – Parameters Tab:**

Allows you to add Parameters and includes values. At present, there are no parameters.

37. Save your changes.





## 5.3 System Design Updates

During the course of Multi-Board system design, it's likely that the source child projects will also be developed with further changes while you are concurrently creating a Multi-Board system design. Such changes will require synchronization with the system design to keep an integrity to the respected child PCB projects. In the exercise below, you will intentionally change one of the child PCB projects and synchronize the change.

### 5.3.1 Making a change to Child PCB Projects, Part 1

In this part, you will modify the child PCB Projects for `Bicycle_Head_Lamp_MainBrd-[Your Name]` and `Bicycle-LED_Brd-[Your Name]`. You will delete one of the redundant ground connections between the two boards as an example of a change. Then, you will do the ERC check to see the results of your change.

38. Change your focus to `Bicycle_Head_Lamp_MainBrd-[Your Name]` project.

39. From the Projects Panel, open the `SCH_LED_Driver_Battery-PWR` schematic.

40. Locate `J1B` on the schematic.

- a) Left mouse click on the ground connection to `J1-10`.
- b) Click again to see that connection to `J1B-10` shows two red terminators on the wire connection, Figure 15.

41. Press the **Delete** key to remove the connection from ground to `J1-10`.

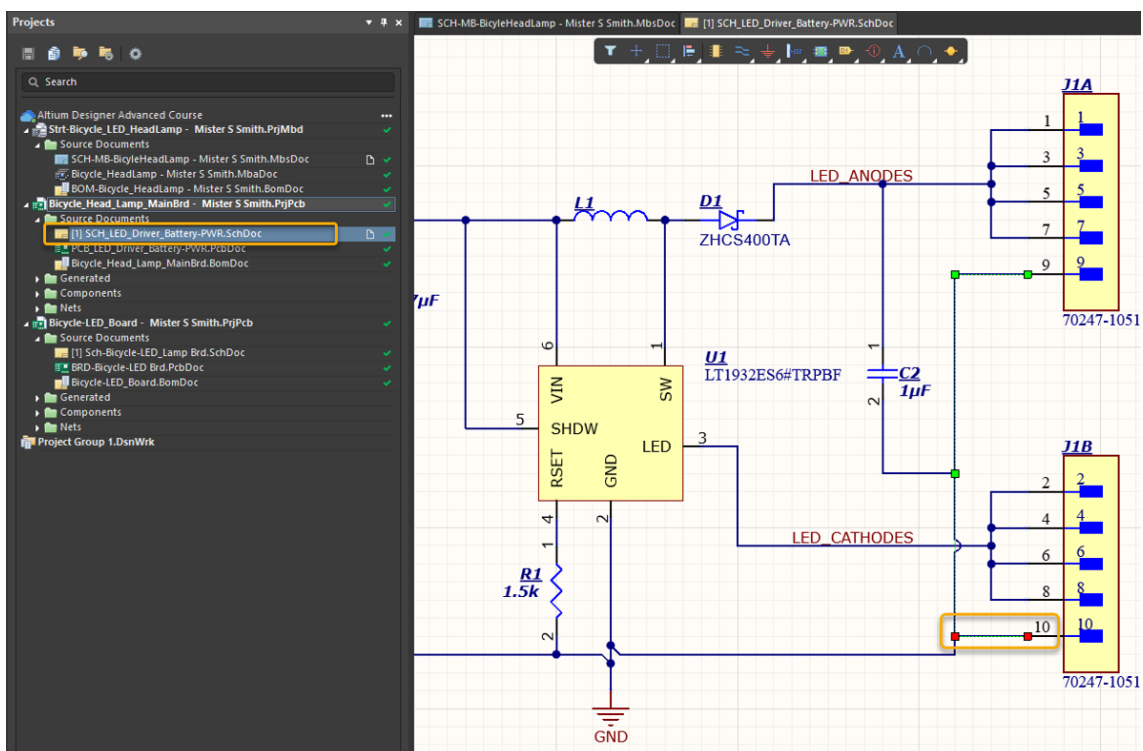
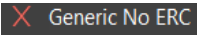


Figure 15. Removal of redundant ground connection



42. Next, apply a **Generic No ERC** marker  on the pin of J1B-10. The **Generic No ERC** marker can be found on the **Active Bar** with a right mouse click on the **Parameter Set** icon to select from Pop-Down menu. See Figure 16 below.

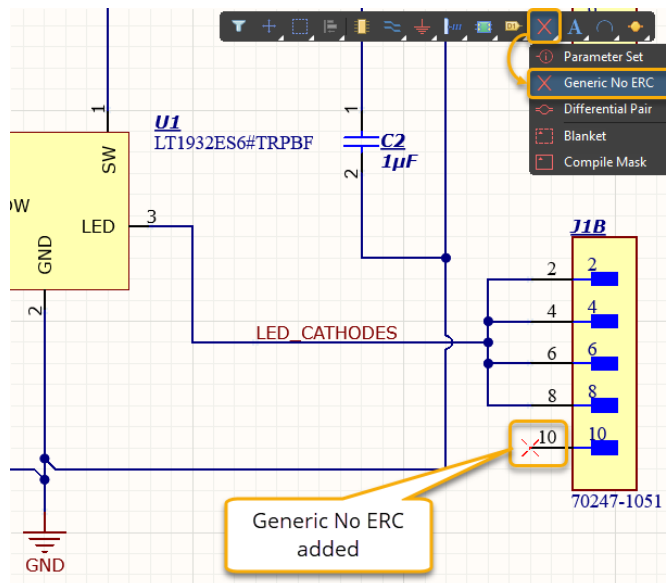


Figure 16. Active Tool Bar, selection of Generic No ERC

43. Let's push schematic update to PCB file. Click on menu **Design » Update PCB Document PCB\_LED\_Driver\_Battery-PWR.PcbDoc**.
44. The Engineering Change Order (ECO) window is displayed:
- Keep the item noted in the **Gold box checked**. All other items noted in the red rectangle should be unchecked, Figure 17.
  - Click on **Validate Changes** to see a green check for each item. Click on **Execute Changes** to initiate the changes to the PCB design.

Note: Since you have removed a connection in the PCB, you may have to open the PCB file. Select menu **Tools » Polygon Pours » Repour All**.

- Save the files.

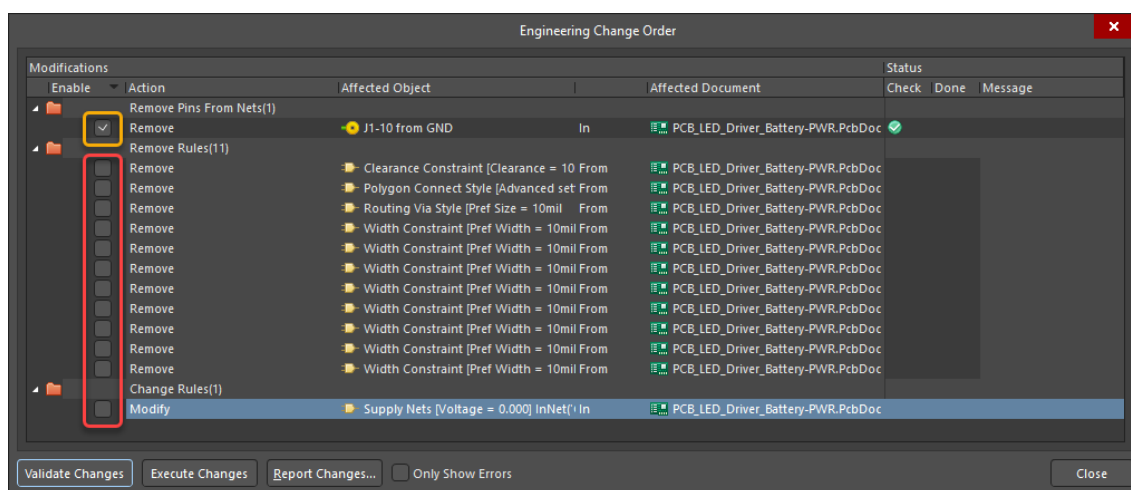


Figure 17. Select what to include in ECO



### 5.3.2 Making a change to Child PCB Projects, Part 2

45. Now use what you have learned from the previous steps to update the `Bicycle-LED_Board-[Your Name]` project. You will remove the GND connection from J1-10.

### 5.3.3 Updating the Multi-Board project from child PCB projects changes

When changes are completed on child projects, you need to update the Multi-Board project to the MB Schematic document. Follow the steps below to update the MB project.

46. Change the focus back to the MB schematic `SCH-MB-BicycleHeadLamp - [Your Name]`.

47. Select from menu **Design » Import From Child Projects**. See Figure 18 below.

- a) Now, the *ECO* dialog is displayed. Ensure to keep the **Check for the Modified** J1-10. Remove the **check** on J1-9. Click on **Execute ECO**.

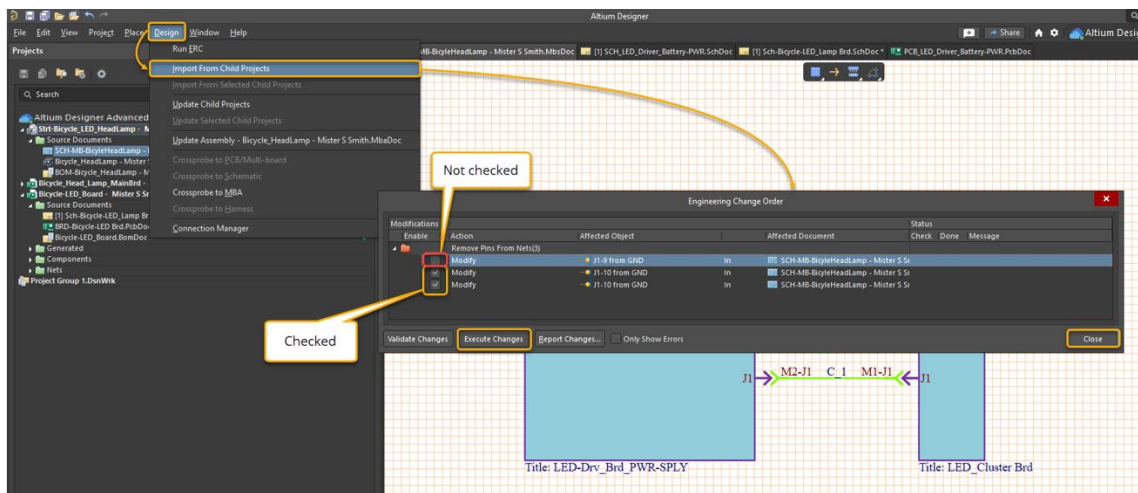


Figure 18. Import changes from child projects to MB Project



## 5.4 Validating the Logical Design

Once the Modules have been connected to each other on the Multi-Board schematic, you can verify the board-to-board connectivity. This will detect net-to-pin assignment errors and pin-to-pin interconnection wiring errors. These errors can be resolved, and corrections pushed down into the affected child PCB projects, or brought back up to the source system schematic.

Next, validate the SCH-MB-BicycleHeadLamp.MbsDoc with the ERC.

48. With the SCH-MB-BicycleHeadLamp-[Your Name].MbsDoc open, left mouse click on **Design » Run ERC**.

49. The *Message* panel will appear displaying the errors found, as seen in Figure 19 below. Let's discover these error messages and develop an understanding of how the messages were interpreted by Altium Designer.

Note: Your MB design isn't at risk when error messages appear. Some errors are allowed with the understanding on how they were derived. The errors displayed are intentional and are part of this lesson.

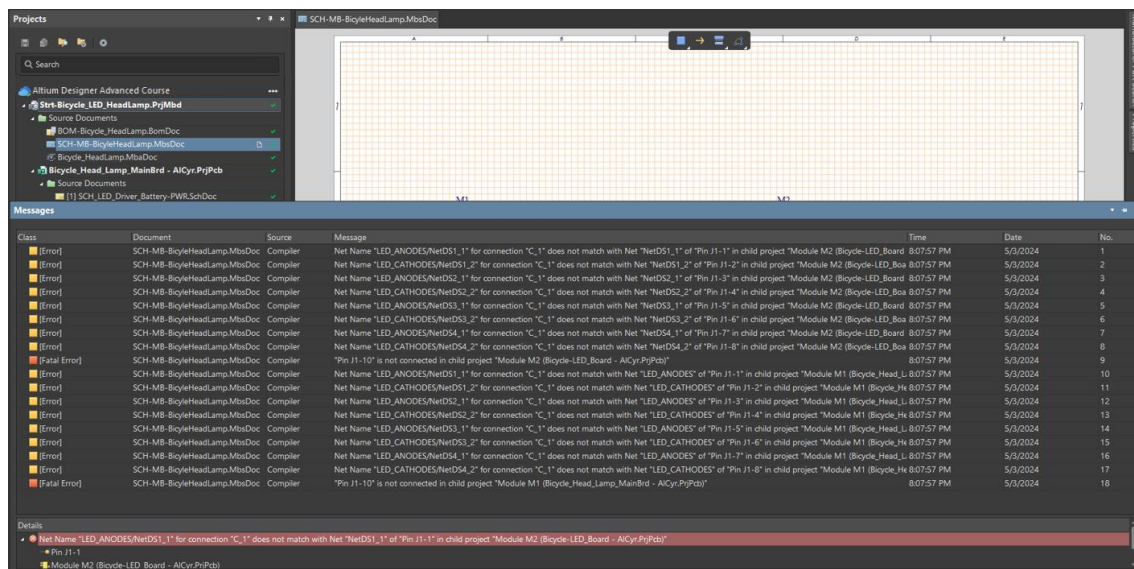


Figure 19. ERC results in Messages panel

- a) **Message class Error** - Net Name [net name] for connection C\_1 does not match with Net [net name] of Pin [Designator - pin #] in child project Module M1 or M2. This message is displayed because the net names on either side of child PCBs don't match. That's not always possible. Figure 20 below shows the net names of each respected child schematic for this MD Design. In this figure you can see a net called DS1-1 on connector J1A-1 for the LED Lamp Board and a net called LED\_ANODES for J1A-1 on LED Driver Main Board. These connections are correct. For the LED Driver Main Board, you have grouped them together with a polygon pour. However, for the LED Lamp board, the designer decided to make individual connections. Since these connections are individual connections on the LED Lamp board, you can't use the same net name for them. In conclusion, this is an allowed error.





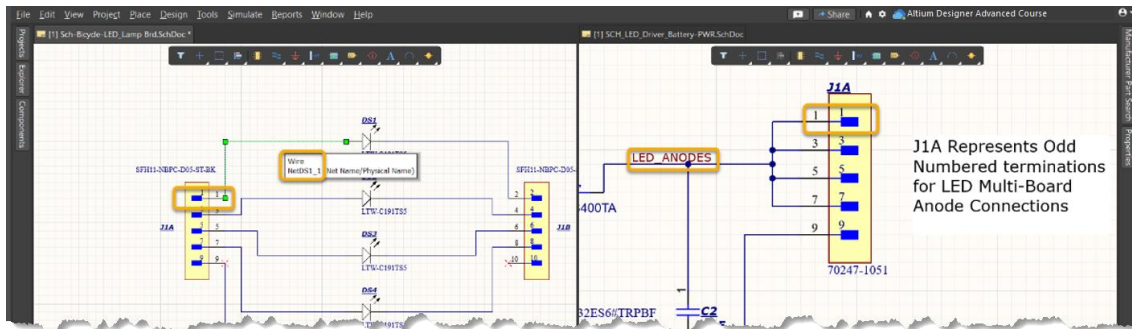


Figure 20. Example of Message class Error with Net Name Expectations

- b) **Message class Fatal Error** – Pin J1 – 10 is not connected in child project M1 or M2. This is another allowed error, even though it's 'fatal'. In an MB Design, this error is common. It may happen that the pins of a selected connector are not all used. In MB Designs, the external connectors are assumed to have connections on all pins and therefore the program will generate an error because it's expecting that all pins should have distributed nets.

Note: The disconnection of J1-10 was intentionally done in section 5.3 System Design Updates. If you allowed J1-10 to be connected to ground on the child PCB projects, you would have no fatal errors.

As seen in Figure 21, there are four grouped images, from A to D:

- Image A shows a schematic section of Main LED Driver Board with J1-10 not connected.
- Image B shows a PCB section of Main LED Driver Board layout with J1-10 not connected to the red ground plane, as J1-9 shows connected.
- Image C shows a schematic section of the LED Lamp Board with J1-10 not connected.
- Image D shows a PCB section of LED Lamp Board Layout with J1-10 not connected.

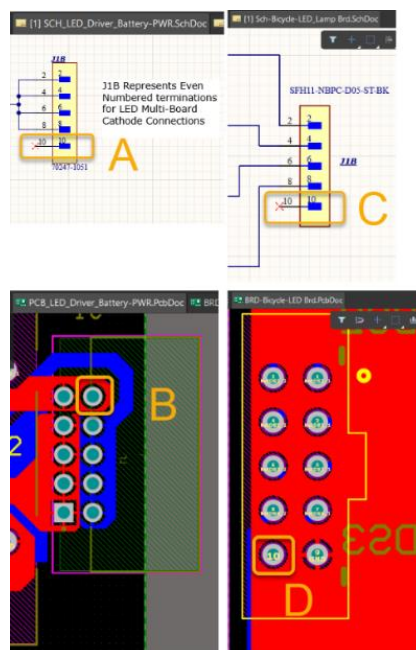


Figure 21. Grouped Images related to J1-10 from child PCB projects - no connection



In conclusion, the Fatal Error regarding the J1-10 lack of connection is an allowable condition in many designs. In our example, as shown in Figure 21, this is an allowable error.

Note: Unlike in a Schematic Document from a PCB project, it's impossible to place No ERC markers in the MB Schematic to suppress these allowable errors.





## 6 Creating the Physical Multi-Board Assembly

### 6.1 Overview

You will create the Multi-Board Assembly in this part. So far, you have created a Multi-Board project with an MB Schematic containing two graphic modules and representing two child PCB projects of a Main LED Driver Board and a LED Lamp Board. Combining these two child PCB projects formulates an assembly of a Bicycle LED Head Lamp Assembly. Having done the ERC, you've learned that the errors reported are not of concern if carefully reviewed.

Now, you will update the Multi-Board Assembly file, as shown in Figure 22. Follow the steps below to accomplish this task.

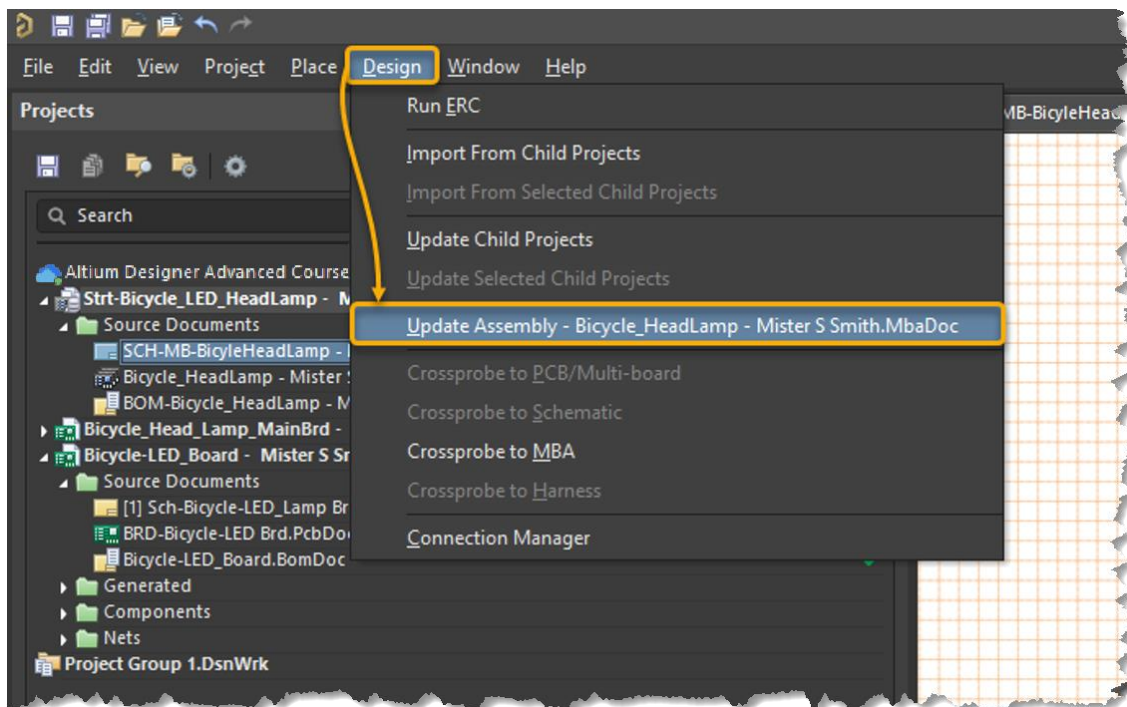


Figure 22. Design » Update Assembly menu selection





## 6.2 Pushing MB Schematic data to your MB Assembly File

50. Ensure your MB Schematic is open. From **Design** menu, select **Update Assembly Bicycle\_HeadLamp.MbaDoc - [Your name]** with a left mouse click, Figure 22 above.
51. The *ECO* dialog is displayed to represent the update change to the MB Assembly file. In this case, the 3D board design, represented as Module M1 and M2, will be added to the MB Assembly file, see Figure 23 below.

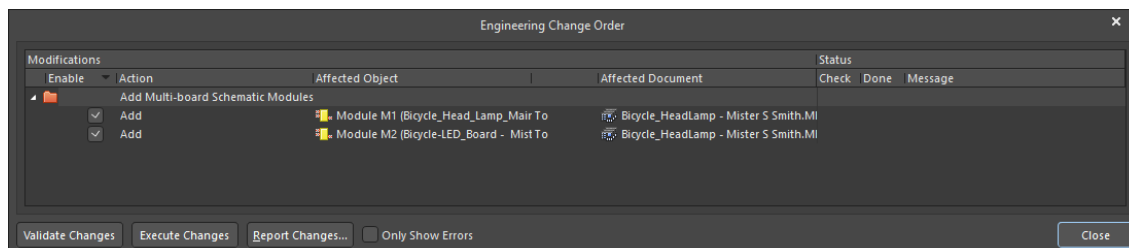


Figure 23. MB Assembly ECO Update Dialog

52. Select **Validate Changes** to see green checks in ECO Section Status Check.
53. Select **Execute Change** to see green checks in ECO Section Status Done.
54. Close the *ECO* dialog with the **Close** button.







## 6.3 Positioning your Multi-Boards within MB Assembly

55. Open the MB Assembly `Bicycle_HeadLamp - [Your Name].MbaDoc` file from the *Project* panel.
56. Figure 24 shows the initial view of your two boards from the MB Assembly file.

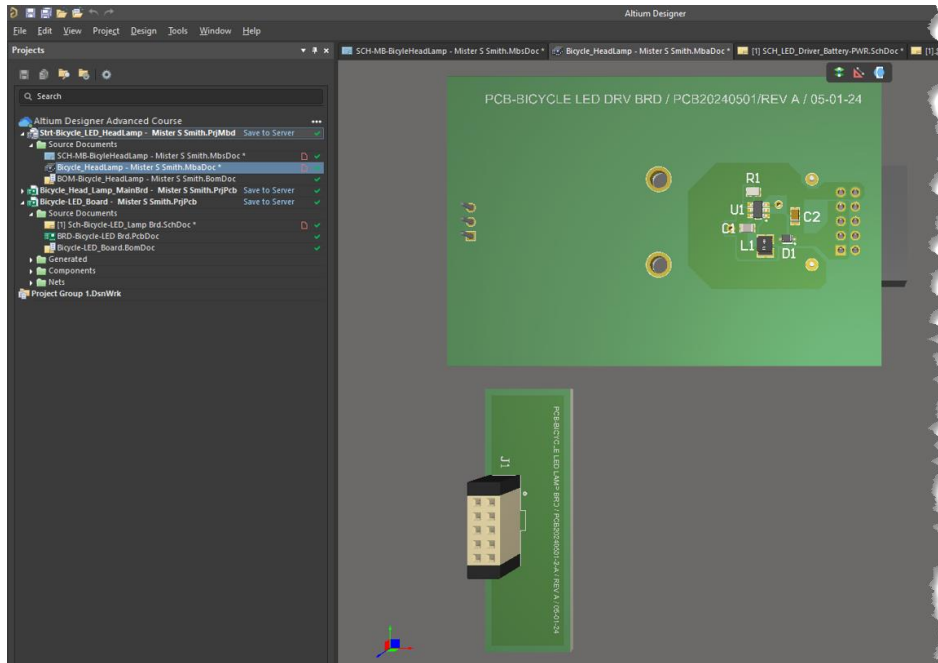


Figure 24. First look at the MB Assembly File

57. Let's first orient the lower, smaller LED Lamp Board. Left mouse click on the board. The board will glow as a highlighted selection and show the workspace Gizmo orientation tool represented in three colors: red, blue, and green, matching the Cartesian plane coordinates of X, Y, and Z.
58. First, orient the J1 connector in the same plane as the J1 connector, as the larger main board. To do this, select the green arc with a left mouse, click and drag to the right until you notice a lock movement of 90°. Results should look like Figure 25.

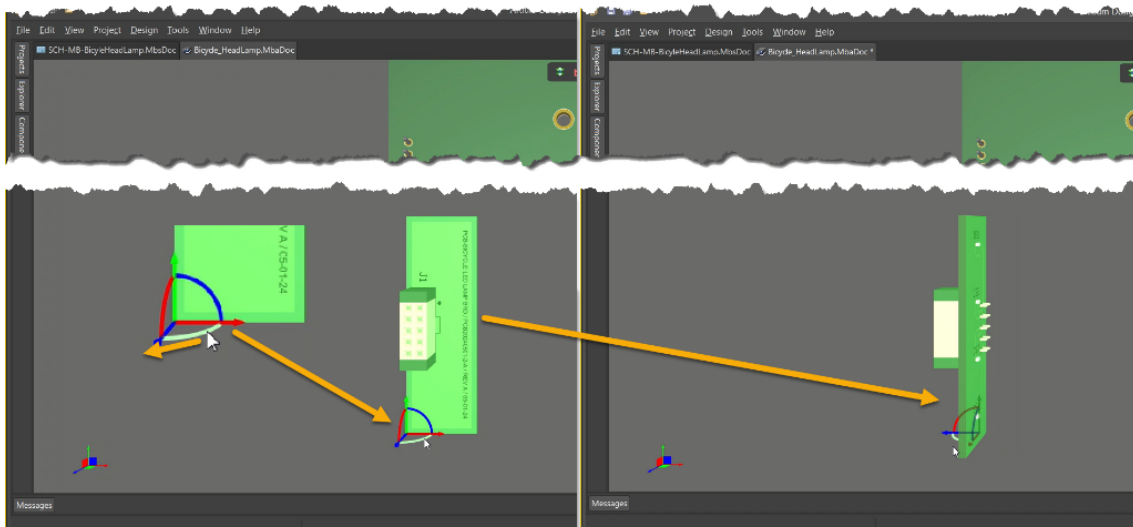


Figure 25. Board flipped upward by 90°



59. Now, move the smaller board towards the larger board connector. Left mouse click and hold while dragging board in front of larger (main) board J1 connector. Once roughly in position, let go of left mouse click. Results should look like Figure 26 below.

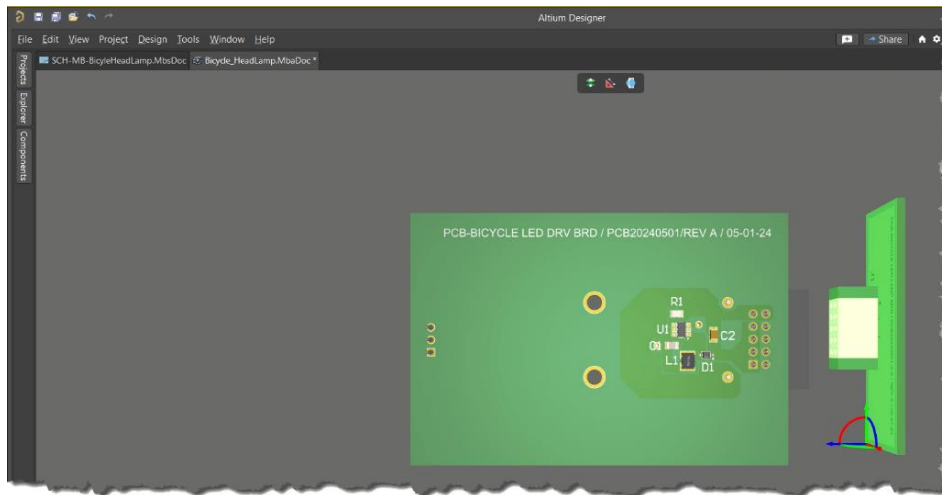


Figure 26. Align smaller board with the larger one

60. Now, flip the small board to correct the pin orientation to that of the larger board. The sequence is shown in Figure 27.
- Left mouse click on the blue arc on the Gizmo tool and notice you can rotate by  $\pm 90^\circ$ .
  - Left mouse click on the same blue arc to rotate  $+$  or  $- 90^\circ$ .
  - Left mouse click on the same blue arc one more time for the last  $90^\circ$  rotation, in the same direction as in step b.
  - Left mouse click with hold to small board and move in front of the J1 connector or larger board.

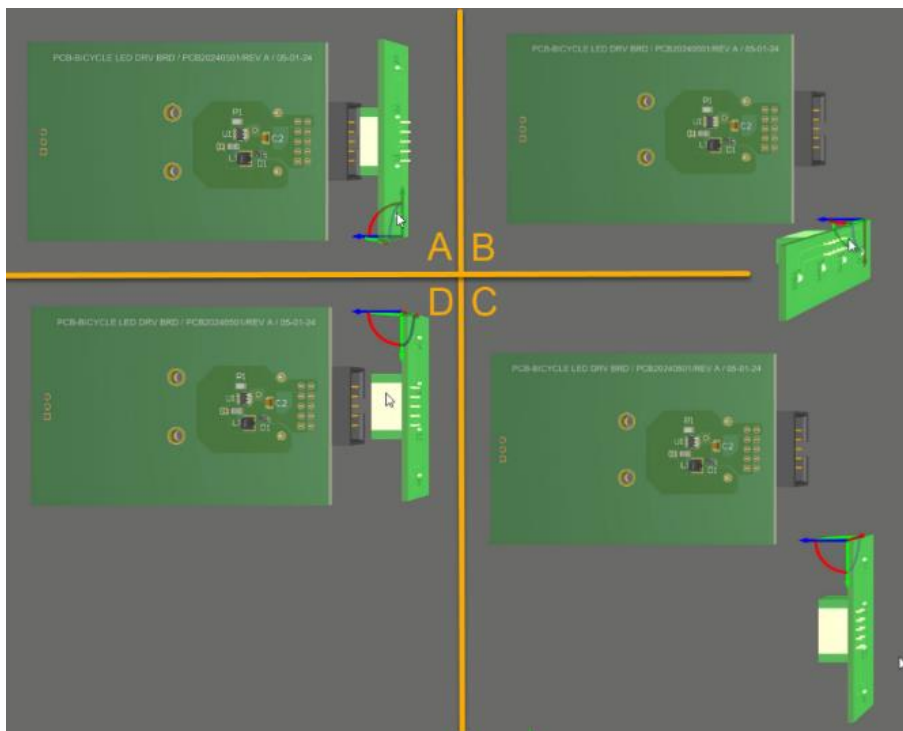


Figure 27. Flip sequence of smaller board and alignment with larger board



61. Next, you will align the smaller board to the larger board J1 connector. Steps from b) to f) follow the same letter captions as seen in Figure 29 below.

- a) Let's toggle the Project Type so that alignment can be more productive. Select from Menu **View » Toggle Projection Type**. See Figure 28.

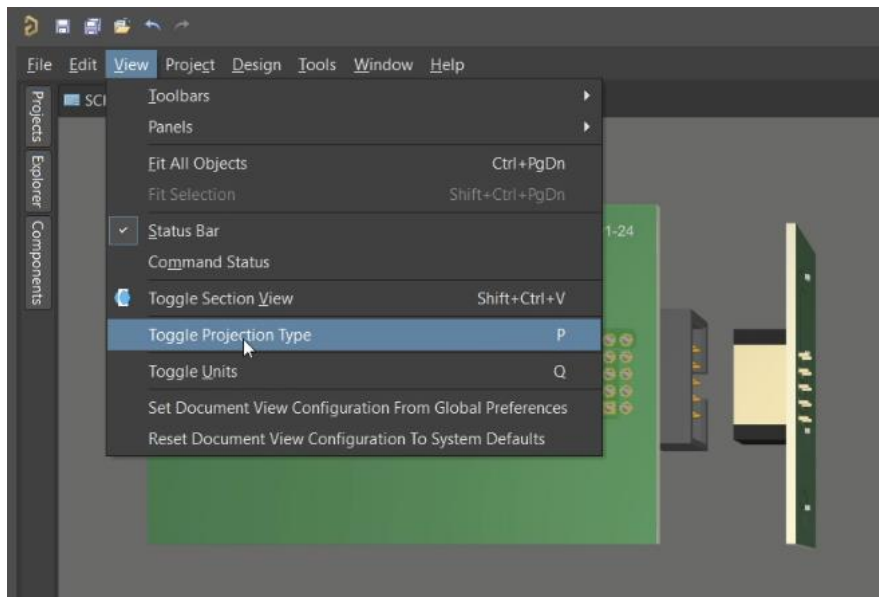


Figure 28. Toggle Projection Type

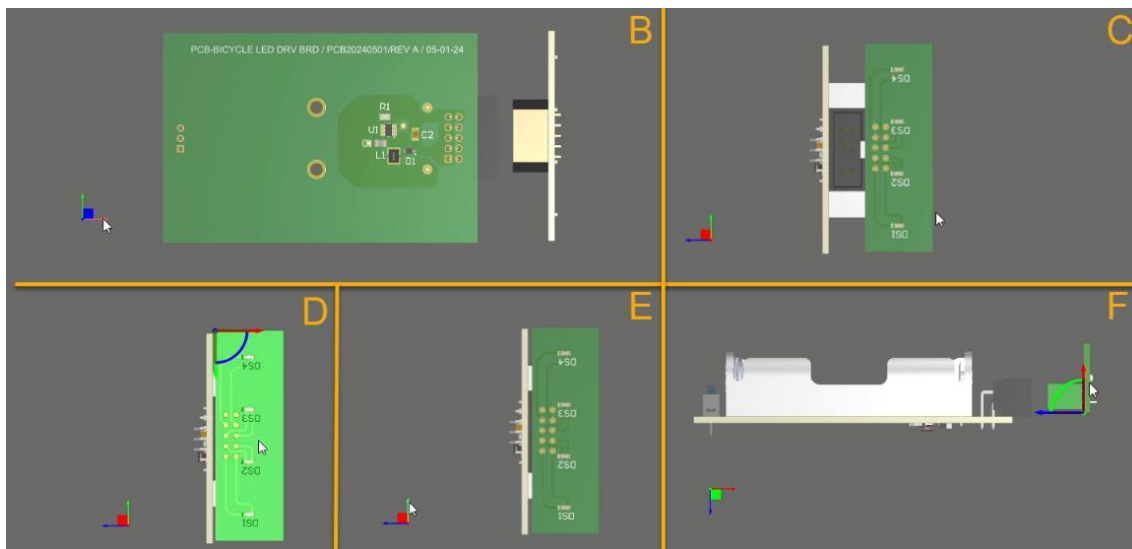


Figure 29. Position the small PCB

- b) Left mouse click on the Red cartesian arrow as seen in image B from Figure 29 to get the front view seen in image C.
- c) Left mouse click on the smaller LED Lamp Board and move it onto the front view of larger board connector in such a way that the J1 connectors of the smaller board are aligned to J1 connector on larger board. Results of alignment should look like image D.
- d) Left mouse click on the Green cartesian arrow to get the side view seen in image F of Figure 29.
- e) Left mouse click on the smaller board and use the red, blue, and green arrows to adjust the position while aligning with J1 on the larger board.





## 6.4 Applying a Mate to Multi-Boards

A mate is a feature often found in mechanical CAD systems, where a surface of one part is mated to the surface of another part through a relationship that could include an offset distance from the mated surface. Altium Designer has such mating features seen from MCAD systems. Follow the steps below to mate the front surface of the smaller board J1 connector to the J1 connector of the larger board in this Multi-Board exercise. Figure 31 will illustrate the steps presented below.

62. Move the smaller board away from the larger board enough to see the back surface of the larger board J1 Connector, Figure 30. Use the skill sets you have attained from the previous steps to move a board within the assembly workspace area.

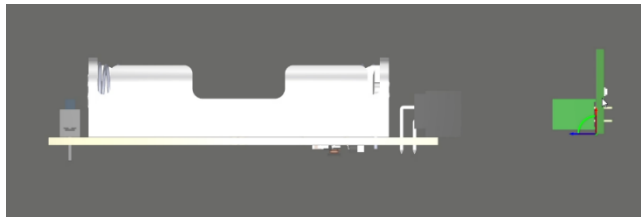


Figure 30. Move Smaller Board Away

63. Arrange the boards to a 3D Isometric view. Looking from an angle, the inside wall of the larger board connector is visible. For this movement, hold the **Shift** key, click and hold the right mouse button, and adjust the view.

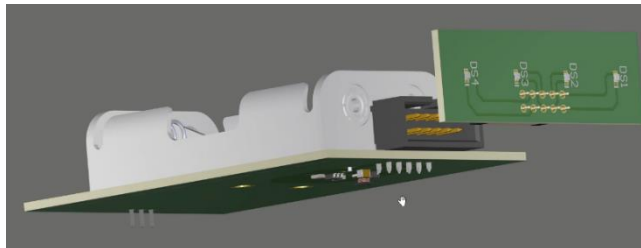


Figure 31. Move Board for best angle view of connector on large board

64. From the MB ActiveBar, select the Mate Mode Icon .

65. Hover the mouse of the back wall of the larger board connector. With a left mouse click, select the back surface of the larger board connector. You should see the Mate symbol in the center of the area, not at the edge of the area. The split circle icon will turn purple as shown in Figure 32.

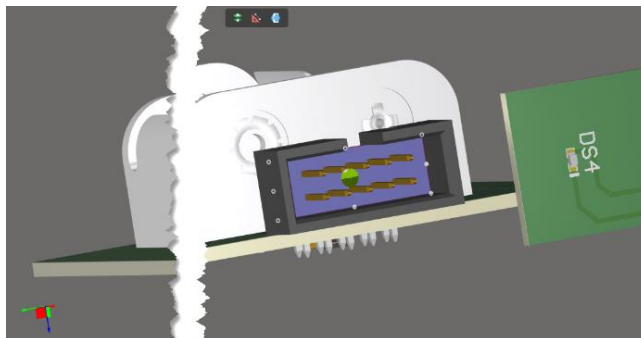


Figure 32. Select back of large board connector as Mate Surface





66. Using technique from step 62, move the view of the arranged board in such a way to have a large angled view of the front surface of the smaller board connector.
67. While still in mate mode, left mouse click on the middle surface area of the smaller board J1 Connector, as shown in Figure 33.



Figure 33. Mate on Small PCB

68. Upon clicking the surface, the smaller board will change position after it mates with the larger board, as seen in Figure 34.

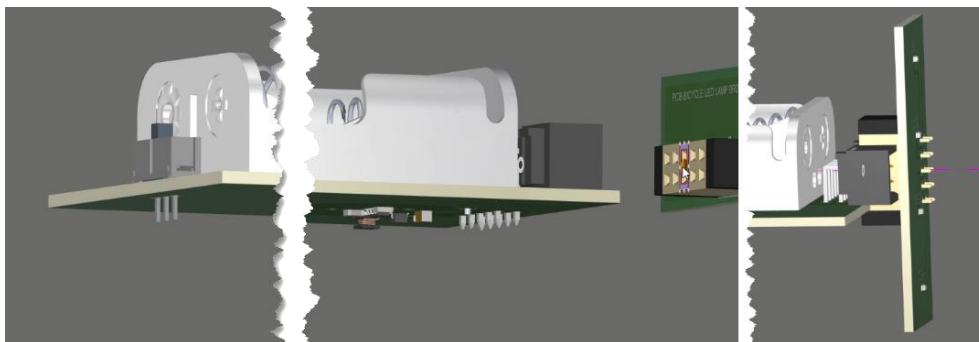


Figure 34. Select the front surface of connector of smaller board

69. Use the *Panels* button to open the *Properties* panel. It provides solutions to rotate the smaller board with the use of the Rotate Counterclockwise with **space bar** or Rotate Clockwise with the **Shift Key + Spacebar**. Use either method to rotate the smaller board against its mated larger board until you achieve what is shown in Figure 35.

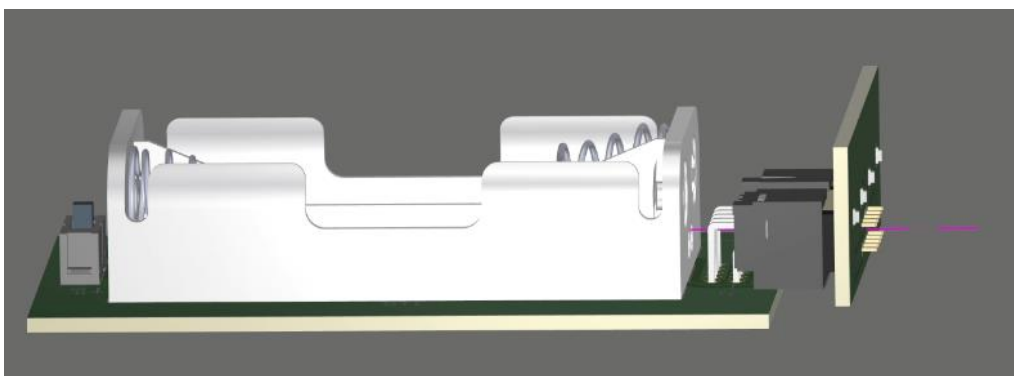


Figure 35. Desired Rotation of smaller board to that of larger board





70. Move the assembled boards in such a way that you can evaluate the insertion of the smaller board connector to that of the larger board, Figure 36.

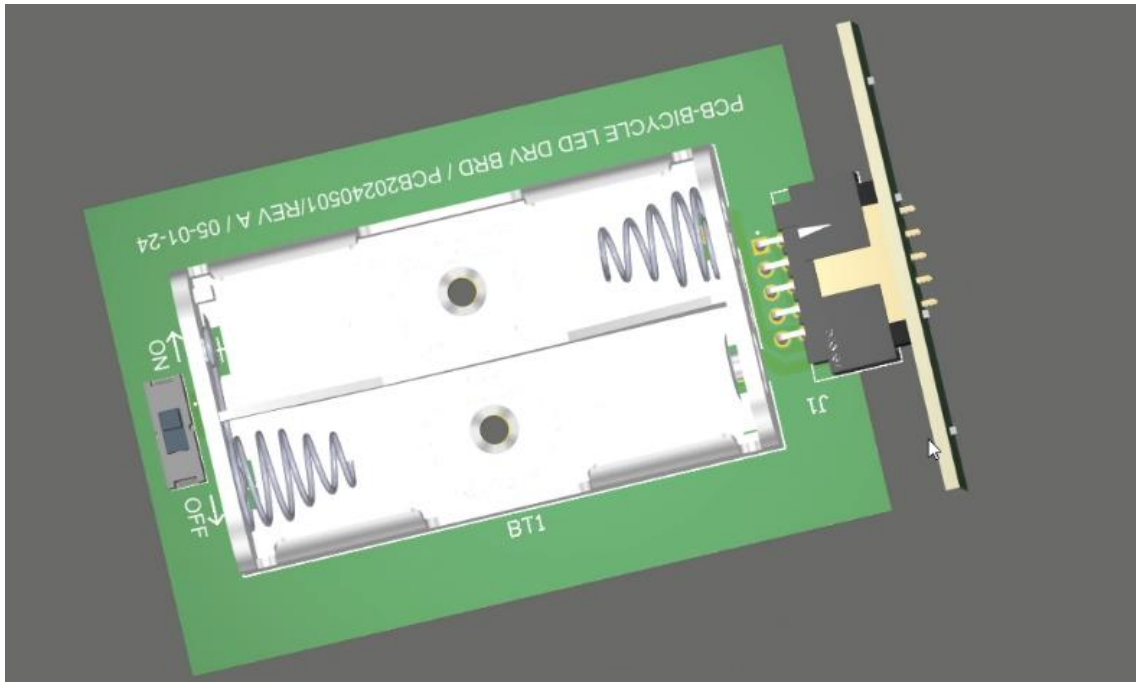


Figure 36. Isometric View showing results of proper mating of boards

Congratulations, your MB is assembled.







## 6.5 Multiboard Assembly Panel

The *Multiboard Assembly* panel allows you to see the details of the Boards and the Mates, change the visibility of objects, and select the defined Mates, so that you can see the parameters of the selected Mate in the *Properties* panel.

71. Open the *Multiboard Assembly* panel from the **Panels** button.
72. Expand the different Branches, as seen in Figure 37, to see details of the Boards and Mate #1.

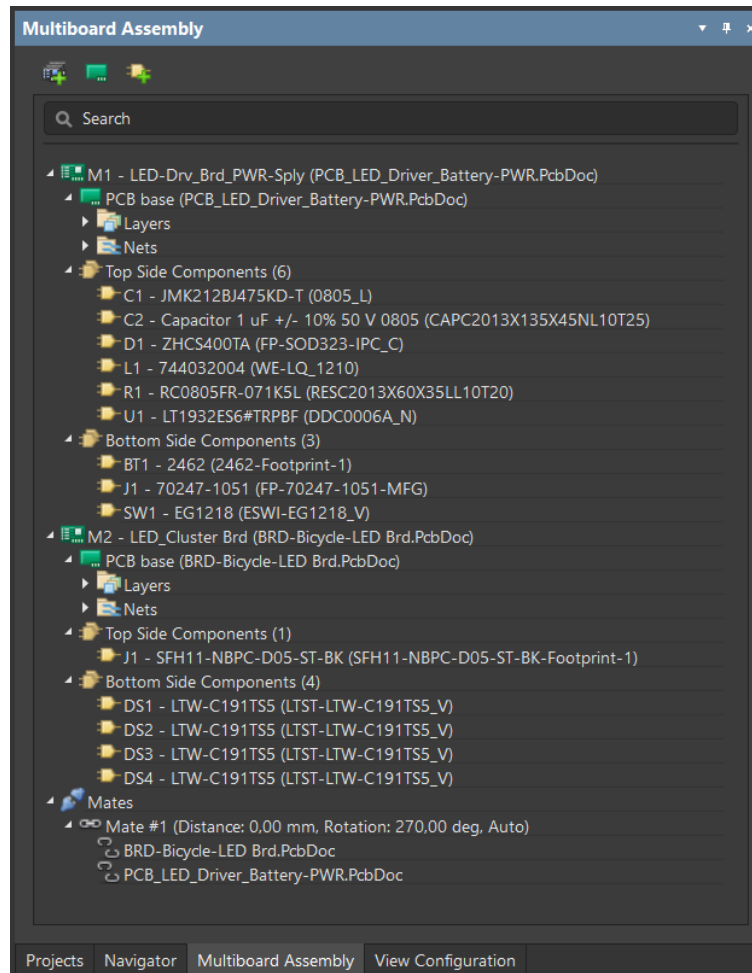


Figure 37. Multi-Board Assembly panel

73. Select the component **BT1** from the *Multiboard Assembly* panel.
  - a) The component is selected in the MB Assembly View.
  - b) Still in the panel, **right-click » Visible** to hide the component. You will see the MB Assembly View without the Battery Pack.
  - c) **Right-click » Visible** to show the BT1 component again.
74. Select Mate #1. This is the Mate you created between J1 and J1.
  - a) Open the *Properties* panel manually or with a double-click on Mate #1.
  - b) Inside the *Properties* panel, you see the properties of Mate #1, similar to the creation step. If needed, you can now change the parameters. Let's change the distance. Add a distance of 0.5mm (~20mil). Check the new position for the small Board.

Hint: If needed, you can delete a Mate from the Multiboard Assembly Panel by using **right-click » Remove Mate**. If you delete a Mate, the objects keep the current position.






## 7 Multi-Board Creation of Drawings

Drawings used in your Multi-Board Project extend beyond the board fabrication and assembly documentation you may have done in the Essentials Training. A Multi-Board Assembly Drawing can include the typical callouts, dimensions, and a BOM. It helps to assemble multiple boards together and provides critical clearance dimensions to measure against.

### 7.1 Creating a Manufacturing Drawing

Follow the steps below to create a new Manufacturing Drawing.

Hint: This exercise doesn't cover all functionalities. For more information, see [Manufacturing Drawing](#) documentation.

75. Let's add a draftsman document to the existing MB Project. From the *Project* panel, hover the mouse cursor over the `Strt-Bicycle_LED_HeadLamp - Mister S Smith.PrjMbd` and with a right mouse click, select from the pop-up menu **Add New to Project » Draftsman Document**.
76. A *New Document* dialog will be displayed, providing the opportunity to select a template. The Training workspace offers the Default. In your own workspace, you may see more options. Select `Default` and click **OK**.
77. The default draftsman document is shown in the workspace. Open the *Properties* panel and dock it with the **pin** . From *Properties* panel, click on the **Page Options**.
78. Select the **Template** tab and select template `ANSI B Landscape`. Notice an immediate change to your document.
79. Let's add the Multi-Board views to our document. Select from the Menu **Place » Multiboard View**. A default view is attached to your cursor. Place this first view within drawing zone C-D, 1-2. You can also use the **Place Multi-Board** icon from the *ActiveBar*.
80. Repeat placing more Multi-Board views. You can change the type of view from the *Properties* panel in **View Side**. From the **View Side**, pull down arrow, select the choice view. You can optimize the position of your view by rotating the orientation with the **space bar**. Use letter **G** for a selection of snap grid.







81. Complete the placement of your views to attain the same image as seen in Figure 38 below.

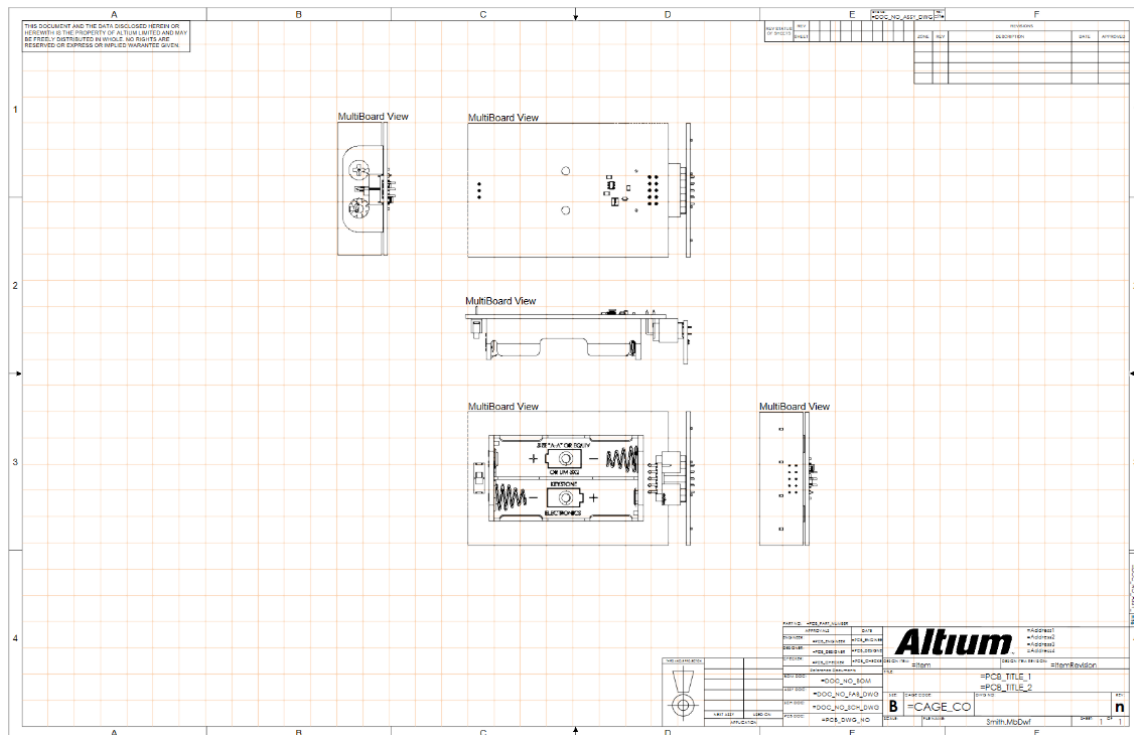


Figure 38. Example Draftsman Document with Multi-Board Assembly Views

A possible final Assembly view, with Dimensions and Assembly Notes looks like Figure 39.

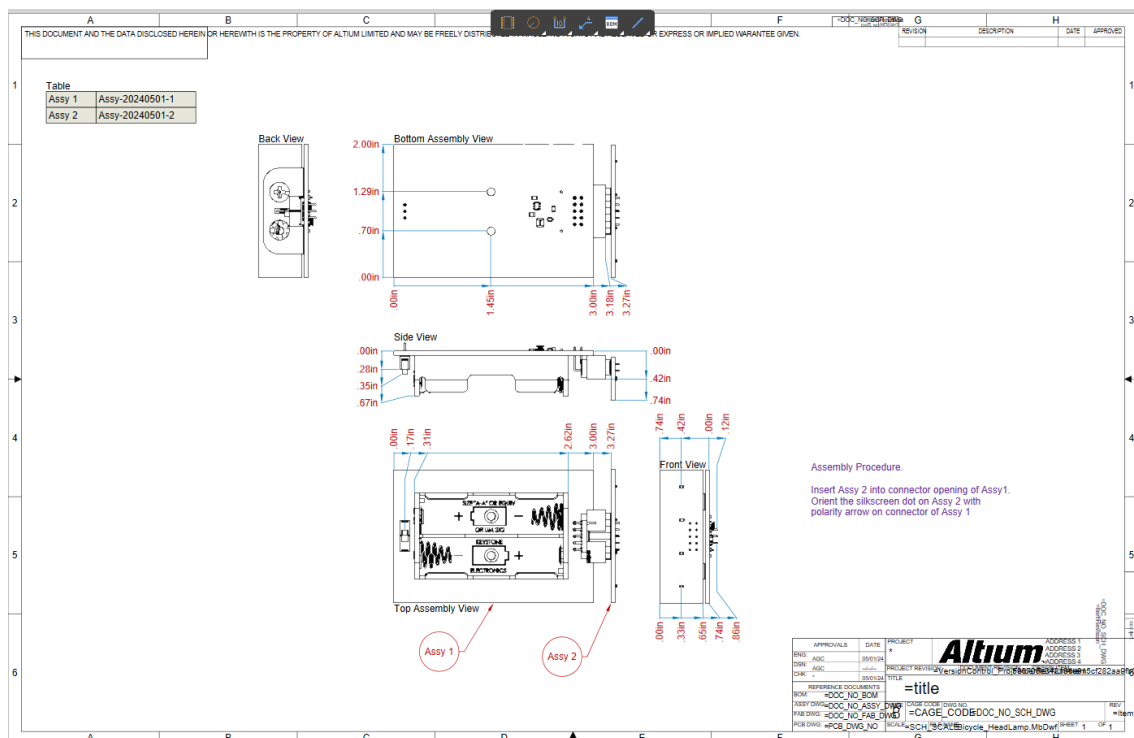


Figure 39. Example of Draftsman Document with Multi-Board Assembly Views, Dimensions, and more





82. Save all documents using **File » Save All**.

83. Save the modifications to the server:

a) In the *Projects* panel, next to the Project name you find the command **Save to Server**

**Save to Server** .

b) Select **Save to Server**.

c) In the dialog *Save [Project Name]*:

i) Add the comment `Multi-Board Design - [Add Your Name] - Finished`.

ii) Select **OK**.

84. When ready, close the project and any open documents, **Window » Close All**.





**Congratulations on completing the Module!**

Multi-Board Design

from

**Altium Designer Advanced Training  
with Altium 365**

Thank you for choosing **Altium Designer**

