



Altium Designer

Advanced Training with Altium 365

PCB CoDesign

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

PCB CoDesign	3
1 Purpose	3
2 Shortcuts	3
3 Create Teams	4
3.1 Team Assignment	4
3.2 Step 1	4
4 Preparation	5
5 Overview	6
6 Updating the PCB	8
6.1 Updating the PCB	8
6.1.1 Repositioning Components	9
6.1.2 Diff Pair Routing	10
6.1.3 Length Tuning	11
6.1.4 Placing Component	12
6.1.5 Changing Polygon Size	13
7 Uploading to Server and Merging	14
7.1 Uploading to Server - Team Member 1	14
7.2 Uploading to Server – Team Member 2	16
7.3 Merging	18
7.4 Saving to Server	19
7.5 Downloading Head Revision	20





PCB CoDesign

1 Purpose

In this exercise, two team members will open the same workspace-managed PCB file as a local copy on their PC. Then, they will merge the individual modifications to create one new head revision.

Altium Designer's latest PCB CoDesign enhancements offer a more efficient and streamlined collaboration process. You can now precisely compare and apply changes to unions for targeted modifications. Also, the 'Show on PCB' option simplifies highlighting changes in specific categories for better visualization. These improvements aim to boost efficiency and provide a more organized and seamless collaborative design experience.

Altium Designer PCB CoDesign simplifies collaboration, ensuring design integrity for engineering teams. This tool streamlines cooperation within your workspace, eliminating manual change tracking. With PCB CoDesign, you can visualize alterations, compare layouts, and merge updates effortlessly in a unified environment.

Caution: This Module requires a Professional or Enterprise License. For details, please see our [Online Documentation](#).
You must be connected to a Workspace in order to use this feature.

Note: Only Git repository-hosted projects are supported. To leverage the PCB CoDesign features when working on an SVN repository-hosted project, it must be migrated to the Git repository first.

Note: The Mini PC project is modified for this training. It's not the same as the default example.

2 Shortcuts

Shortcuts used when working with PCB CoDesign

Shift+B	Check previous status of modified object (PCB CoDesign panel)
L	View Configuration Panel
Ctrl+S	Save Document





3 Create Teams

3.1 Team Assignment

Before we start, it's important to keep in mind a few key points:

- For this module, we need teams with two Team Members.
- If no one else organizes the teams, the Trainer can organize them.
- It's important to remember your team number for the following instructions.
- Each team has a Team Member 1 and a Team Member 2.
- It's also important for the training flow to remember if you are Team Member 1 or Team Member 2.

3.2 Step 1

The following flow with John and Mia (Team Member 1 and Team Member 2) represents the flow for this training.

Let's begin with Step 1.

The data is available in the workspace, but not yet downloaded as local a copy to the individual PCs and not opened in Altium Designer.

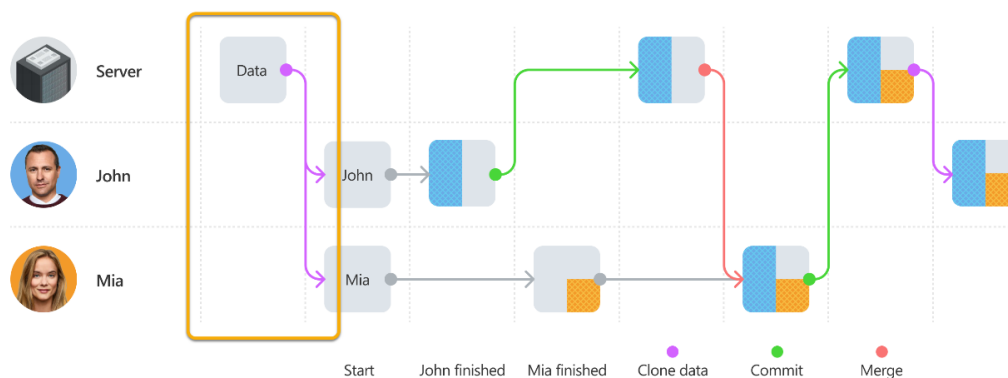


Figure 1. Training Flow – Step 1

Hint: We recommend that designers using the PCB CoDesign features have the same version of Altium Designer.

Note: When merging polygon pours and selecting the **Combined** or **Remote** option, note that a merged polygon needs to be repoured (you can do this, for example, by right-clicking the polygon in the design space and selecting **Polygon Actions » Repour Selected** from the context menu). If the **Repour Polygons After Modification** option is enabled on the [PCB Editor – General page](#) of the *Preferences* dialog, the polygon pour will be automatically repoured after merging.



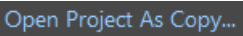



4 Preparation

1. Close all existing projects and documents.

Team Member 1: Follow the Instructions Step 2 to Step 7.

Team Member 2: Wait until Team Member 1 has created the local copy from the training project, then follow the instructions Step 8 and Step 9.

2. Create a copy of the Training Project PCB CoDesign.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Navigate to the predefined training project PCB CoDesign (Top\Projects\Altium Designer Advanced Training Course\...).
5. Select **Open Project as Copy...** .
6. In the new dialog *Create Project Copy*:
 - a) Add your team number and your name to the project: PCB CoDesign - [Team Number] - [Your Name].
 - b) Add a description: PCB CoDesign - [Team Number] - [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 team folder: Project\For Attendees\[Team Folder]
 - ii) Select **OK**.
 - e) Change the **Local Storage** path if needed.
 - f) Select **OK** to create the copy.
7. Wait until Altium Designer has created the copy of the project and opened the project in the *Projects* panel. This can take up to 1 minute.

Team Member 2:

After **Team Member 1** has created the local project, open the training project PCB CoDesign - [Team Number] - [Team Member Name].

8. Select **File » Open Project...** to open the *Open Project* dialog.
9. Navigate to the training project that Team Member1 just created PCB CoDesign [Team Number] - [Team Member Name] Path: Top\Projects\For Attendees\[Team Number]\... .






5 Overview

Team Member 1 and Team Member 2

10. From the *Projects* panel, open the `MiniPC.PcbDoc`.

Note: The Mini PC project is modified for the training. It's not equal to the default example that could be found in the Altium Example folder.

11. The PCB is preconfigured with a Layer Set (Default Training), an active filter, and shelved polygons.
12. In the lower left-hand area and in the upper right-hand corner of the PCB, you can find zone information for **Team Member 1** and **Team Member 2**, created on a mechanical layer, as shown in Figure 2.
13. In the upper right corner of the PCB area, Altium Designer adds an icon for each person that has also opened the document, for example,  as shown in Figure 2.

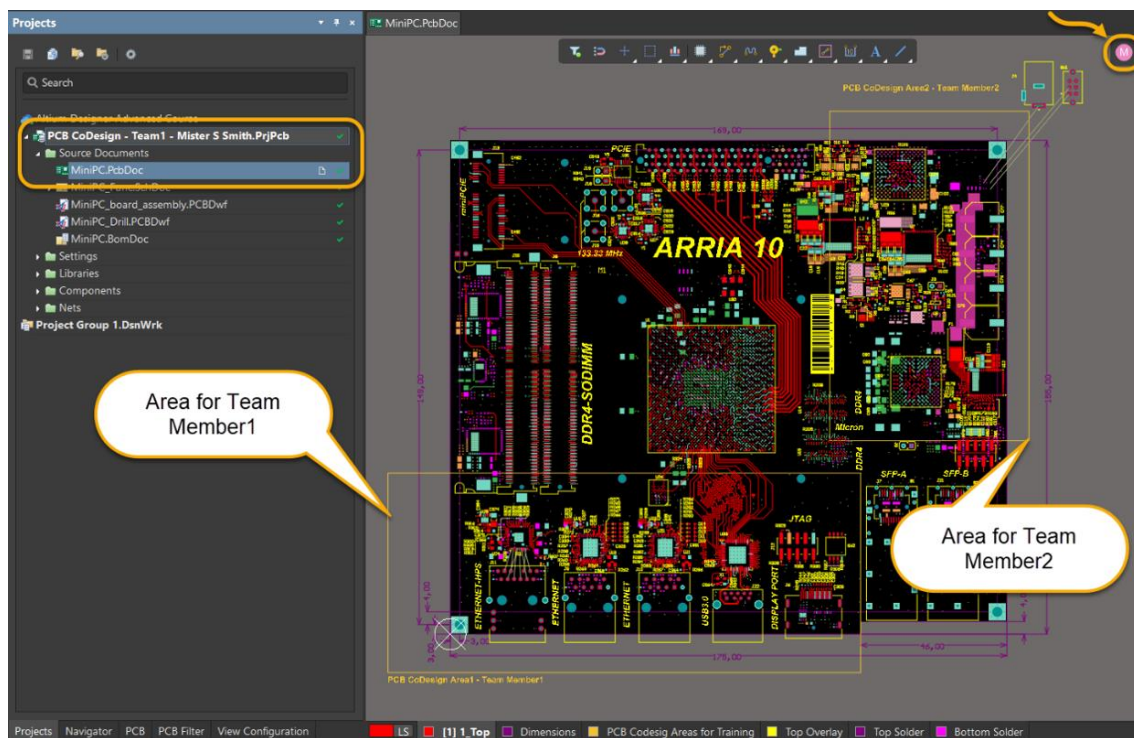


Figure 2. MiniPC PCB with predefined areas for PCB CoDesign





14. We are now at Step 2 in the flow. Mia and John have a local copy of the project, Figure 3.

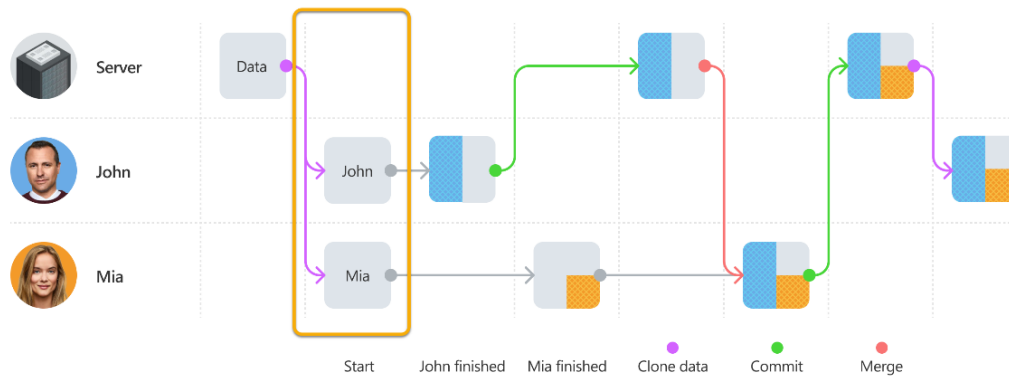


Figure 3. Training Flow – Step2: The team members have a local copy of the project






6 Updating the PCB

6.1 Updating the PCB

Now that both team members have the PCB open, they can start updating.

For CoDesign **Area 1**: the example project has been prepared with three types of modifications.

For CoDesign **Area 2**: the example project has been prepared with two types of modifications.

During the time that Team Member 1 is editing the PCB, Team Member 2 will also start editing the PCB. Altium Designer detects that two engineers are now working on the same PCB and will update the icon in the upper right corner .

Hint: You may see messages from the system, like *[Name] started editing*. The message you see depends on whether you first started to edit the PCB or if Team Member 2 first started to edit the PCB.

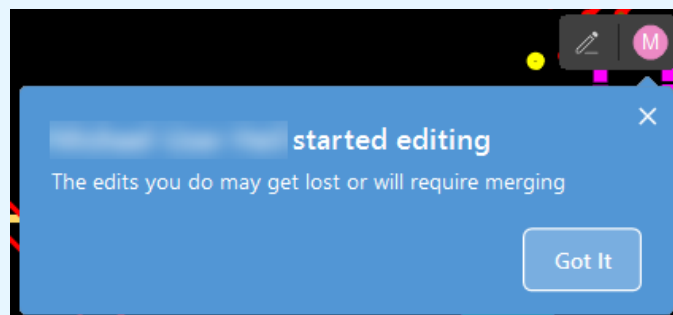


Figure 4. Example for CoDesign message - Team Member 1 starts first with editing the PCB

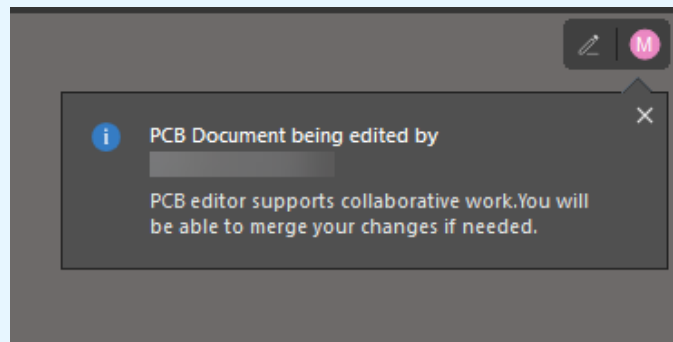


Figure 5. Example for CoDesign message - Team Member 2

Select **Ignore Lock** to continue.

If you are Team Member 1, continue with 6.1.1 Step 15.

If you are Team Member 2, continue with 6.1.4 Step 28.





Team Member 1:

6.1.1 Repositioning Components

15. Search for R304 and change the zoom level so that you see R304 and the area around it, as seen in Figure 6. Feel free to choose the command you prefer to find R304 (**Jump - Component**, *PCB* panel, Search field, or other).

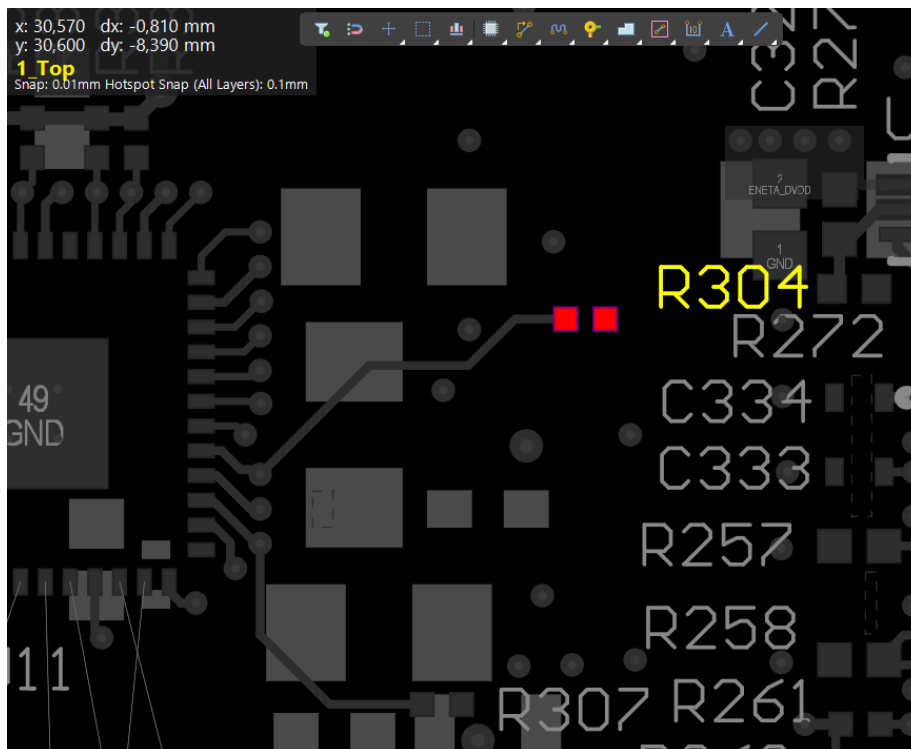


Figure 6. Search for R304

16. Reposition R304 and move it close to U11, as seen in Figure 7. Reroute the trace as needed.

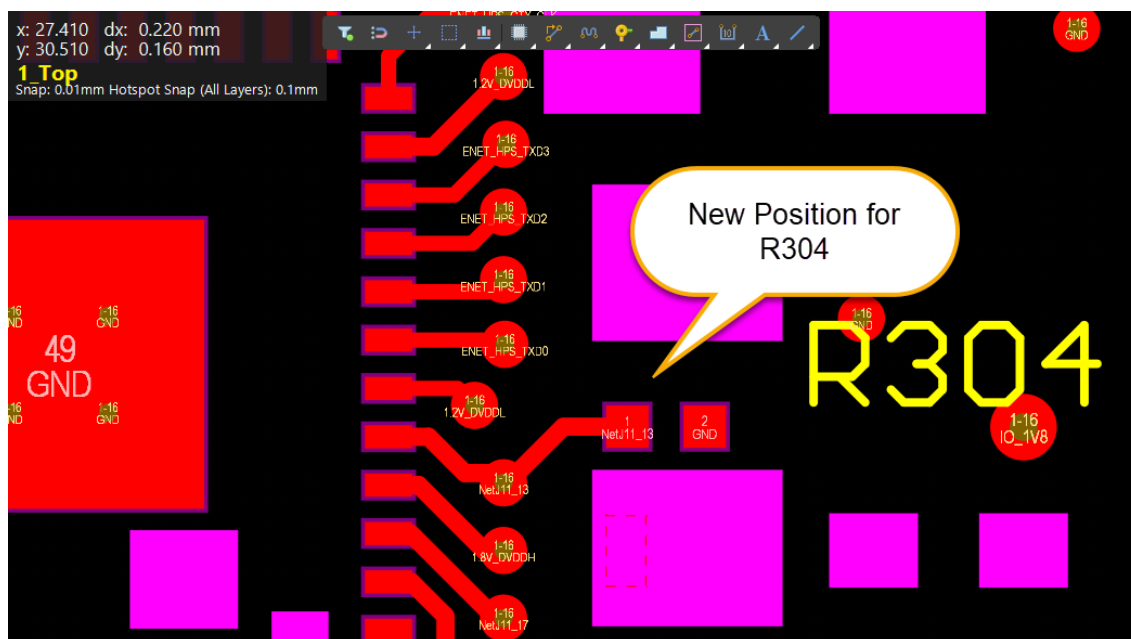


Figure 7. New Position for R304





6.1.2 Diff Pair Routing

17. Jump to U11.
18. You will find four un-routed Differential Pairs, from TRD1 to TRD4, going downwards from U11 to J11.
19. Route two of the four differential Pairs, TRD1 and TRD2, as seen in Figure 8. Start the Differential Pair routing command with your preferred method, for example, **U » I**.

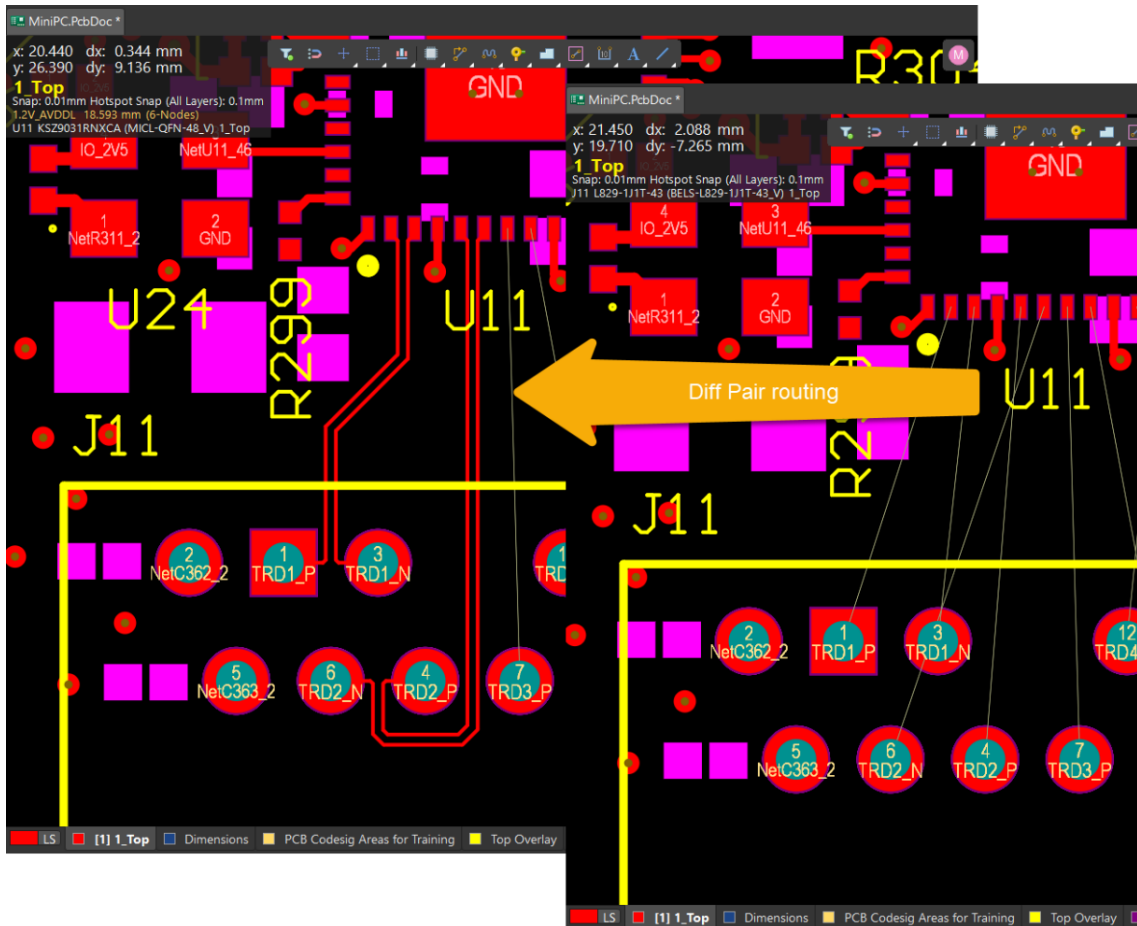


Figure 8. Differential Pair Routing





6.1.3 Length Tuning

20. Jump to Net `USB3_D27`. Feel free to choose any command you prefer (*PCB* panel, Search field, or other).
21. Add a length tuning pattern, **U»R**, to the Net `USB3_D27`. Check that the **Clip to target length** is active and the target length is between 77,951mm and 78,951 mm.
 - a) Max Amplitude: 1mm
 - b) Space: 0.3mm
 - c) Miter: 25%
 - d) Style: **Mitered Lines**
22. Repeat Step 21 for the net `USB3_D26`, as seen in Figure 9.

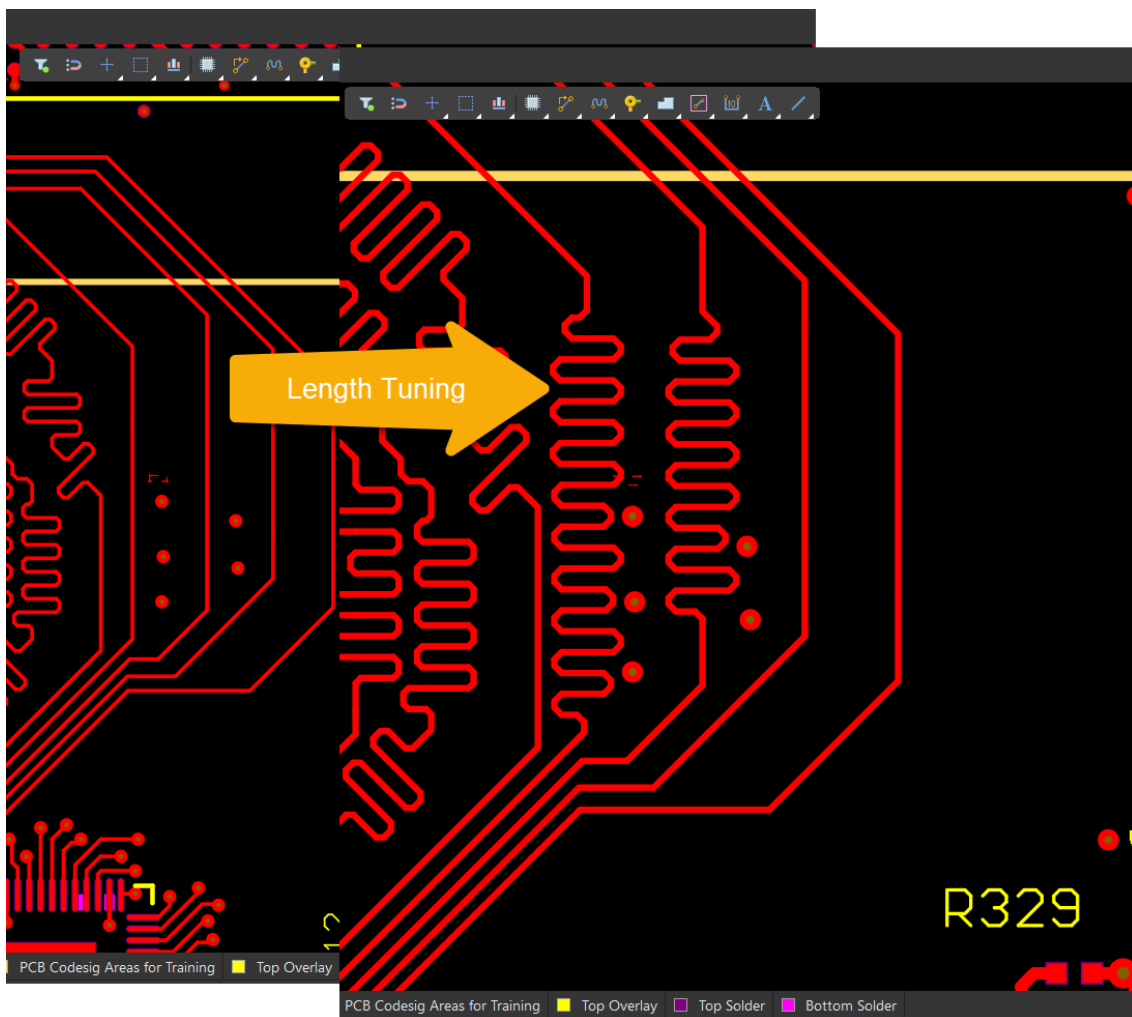


Figure 9. Length Tuning

23. Restore the Polygon Pours with **Tools » Polygon Pours » Restore...**
24. Repour all Polygon Pours with **Tools » Polygon Pours » Repour All.**
25. Save your modifications with **File » Save (CTRL+S)**.
26. Wait until **Team Member 2** has finished their modifications.
27. Next, continue with Step 36.



**Team Member2:****6.1.4 Placing Component**

28. Search for J4. Feel free to choose any command you know to find J4 (**Jump – Component**, **PCB** panel, **Search** field, or other).
29. Next, position J4 at the coordinates x: 162.2mm / y: 150.2mm. Feel free to choose any method you know to position J4, Figure 10.
30. Lock J4 to avoid moving J4 later.

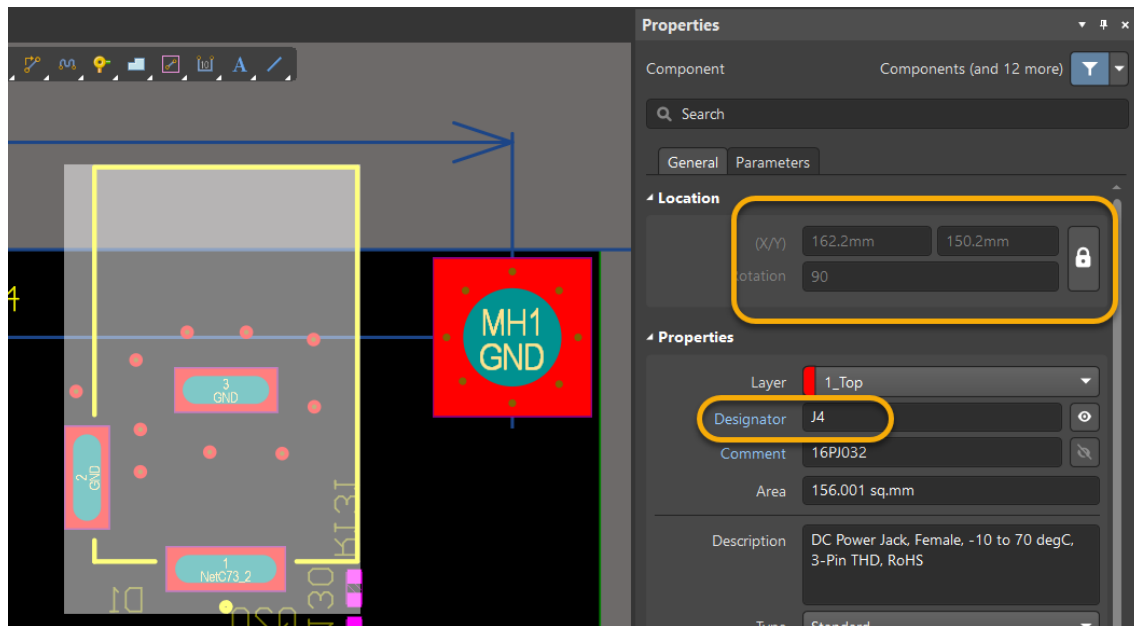


Figure 10. Placement of J4





6.1.5 Changing Polygon Size

31. Jump to the location x:165mm / y: 135mm with the command **J » L**.
32. On the Top Layer '1_TOP' is a polygon pour with the name `MAIN_12V`. Next, you will change the size of this polygon pour.
 - a) Select the polygon pour.
 - b) Select the upper edge of the Polygon and change its position in a way that the polygon pour overlaps the Vias in the copper area around Q20, as seen in Figure 11.
 - c) Run the command **Tools » Polygon Pours » Repour All** to update all Polygon Pours.

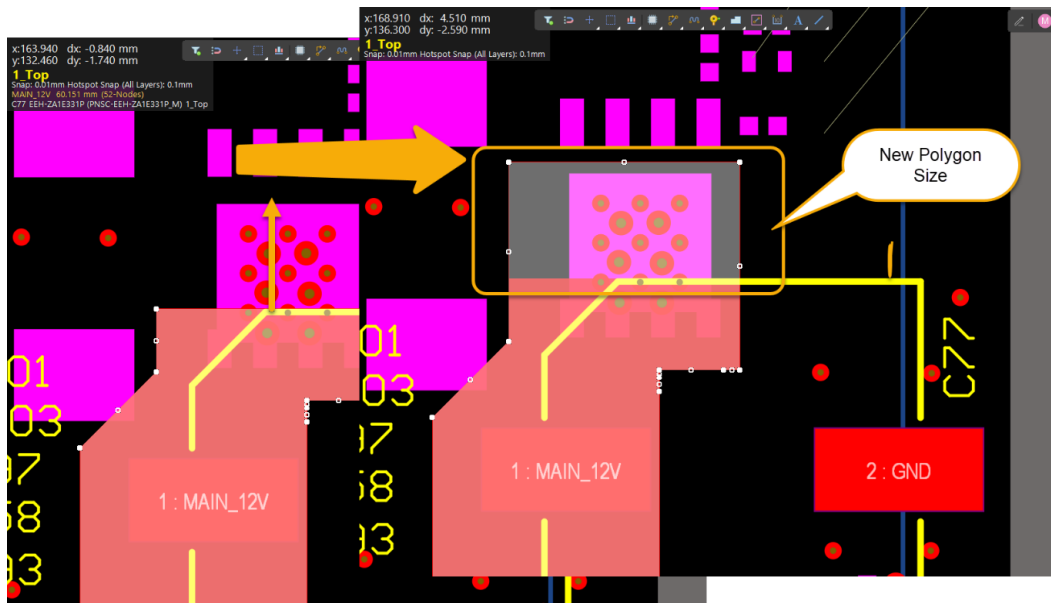


Figure 11. New Polygon size

33. Save your modifications with **File » Save (CTRL+S)**.
34. Wait until **Team Member 1** has finished their modifications.

Team Member 1 and Team Member 2:

35. Wait until your Team Member has finished their part of the exercise.
36. Now, you are at Step 3 of our flow. John and Mia have made local modifications.

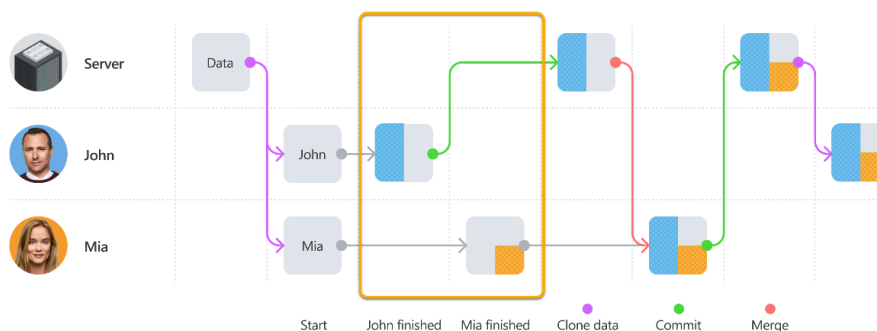


Figure 12. Training Flow - Step 3: John and Mia made local modifications





7 Uploading to Server and Merging

Next, Team Member 1 will commit their changes to the workspace. Team Member 2 could continue with updating the PCB. However, for the training, Team Member 2 will wait until Team Member 1 has committed their changes.

7.1 Uploading to Server - Team Member 1

Team Member 1:

37. From the *Projects* panel:

- a) An icon next to the PCB name is showing a conflict situation.
- b) Select the command **Save to Server** [Save to Server](#).

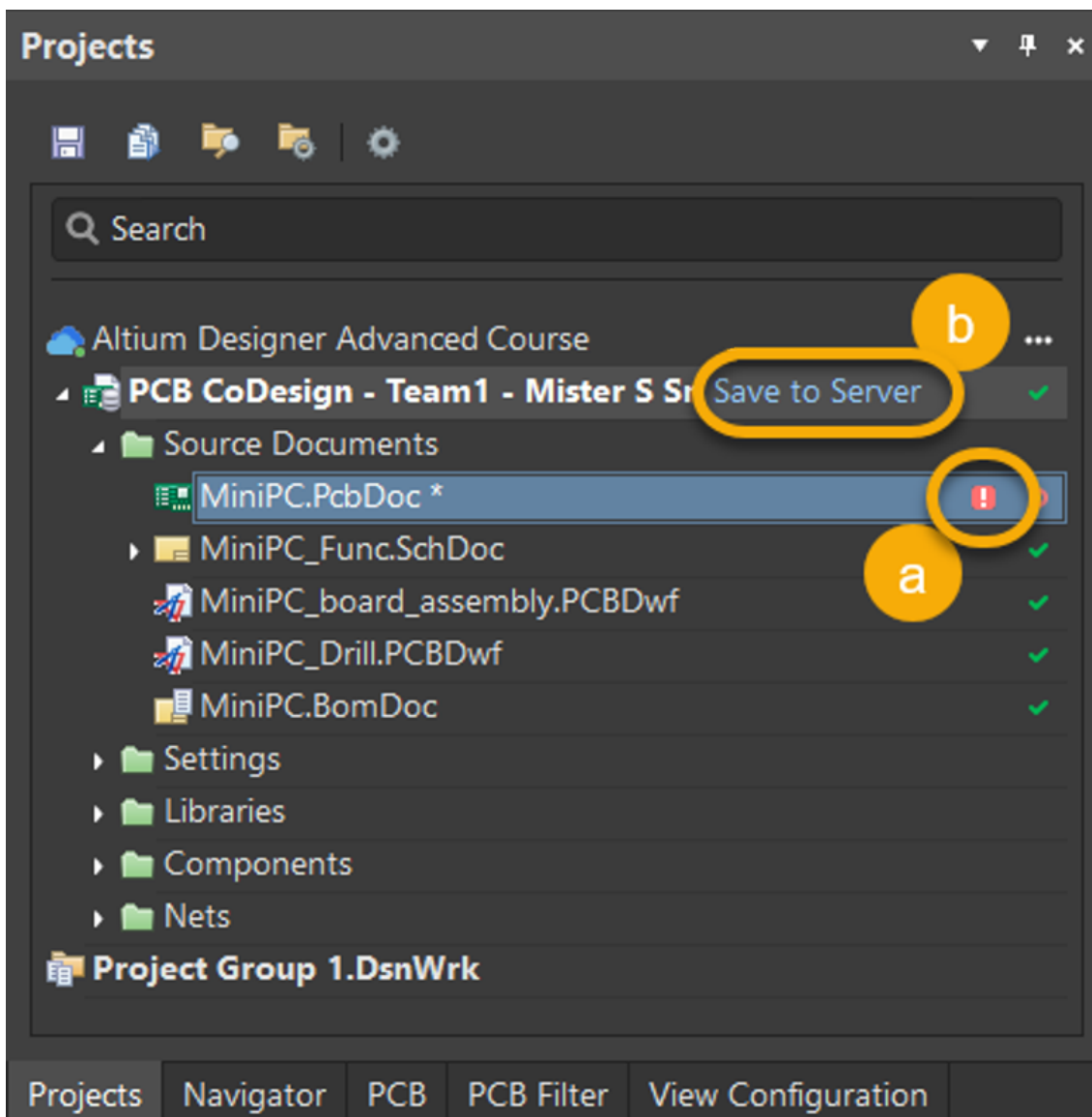


Figure 13. Team Member 1 - save changes to server





38. In the *Save to Server* dialog, see Figure 14:

- Check that the checkbox for the PCB is enabled.
- Add a comment, for example, Area 1 - Team Member1 Component Placement, Differential Pair Routing, Length Tuning.
- Select **OK** to save the modifications.

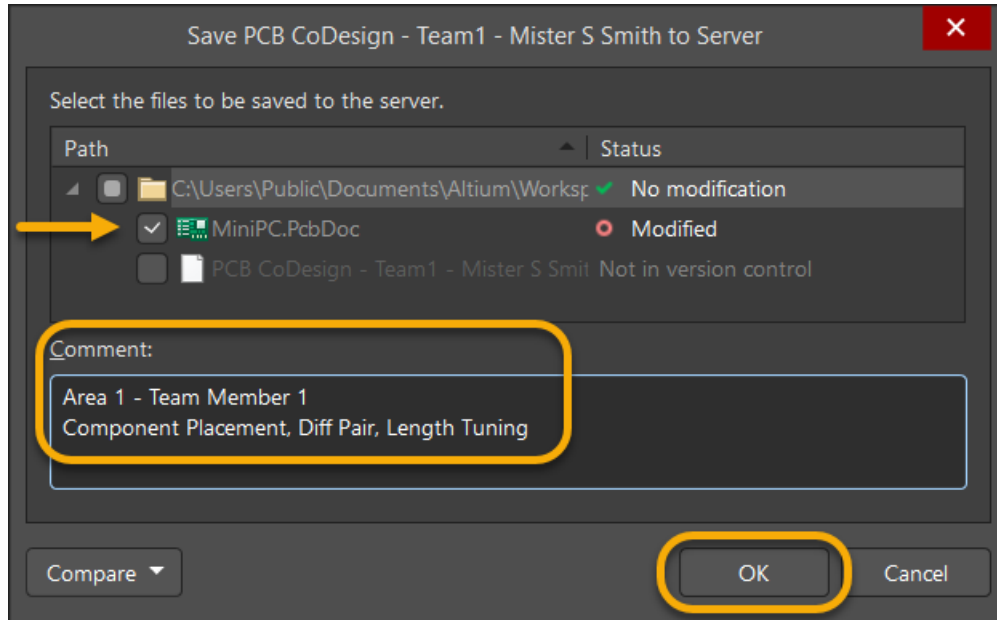


Figure 14. *Save to Server* dialog

39. This is Step 4 of the flow, where John has committed his changes to the server.

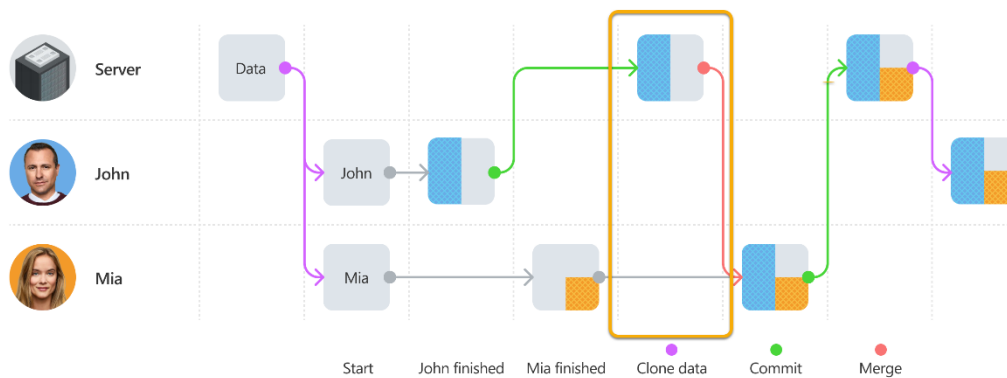


Figure 15. *Training Flow – Step 4: John has committed his modifications*






Team Member 2:

After **Team Member 1** has committed their changes, **Team Member 2** will now commit and merge their changes.

7.2 Uploading to Server – Team Member 2

40. After Team Member 1 has uploaded their changes, Team Member 2 will see a message: *Document is Out of Date*, see Figure 16:

- Left-click on the GIT status icon  in the *Projects* panel to open the *Conflict detected* menu options.
- Note the additional *Document is Out Of Date* dialog at the bottom of the PCB file.

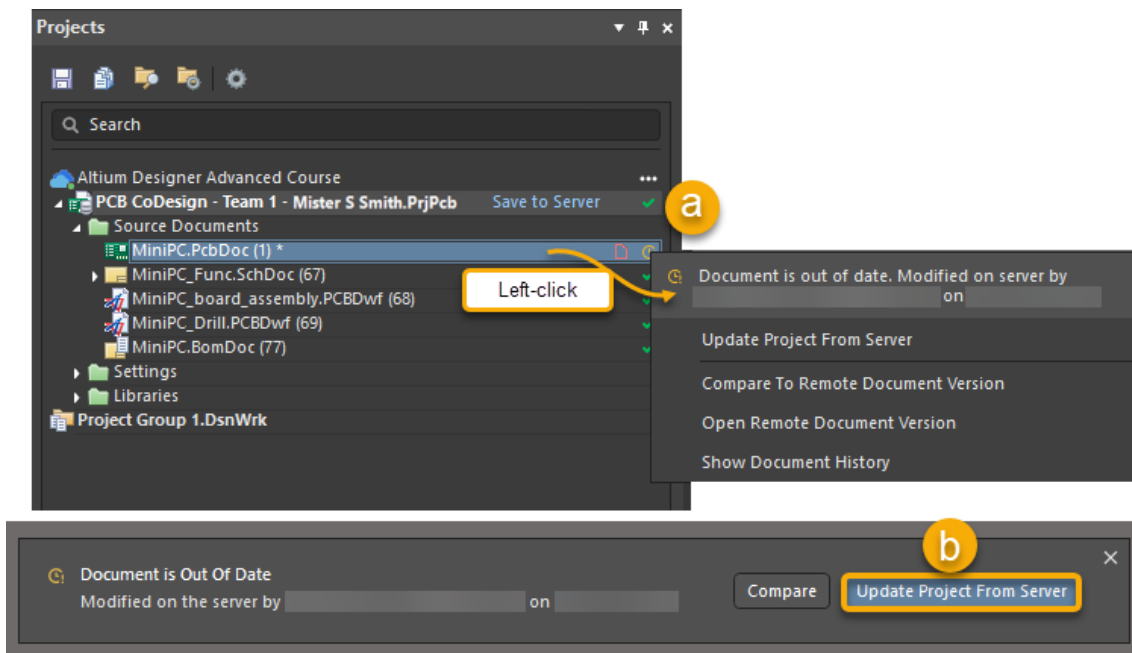


Figure 16. Out Of Date Message - Team Member 2

- If not already open, open the *PCB CoDesign* panel.
- From the *Projects* panel or from the *Document is Out of Date* dialog, start the **Update Project from Server** command.
- When comparing is in progress, you see an information dialog with the progress, as seen in Figure 17. Wait until comparing is finished.

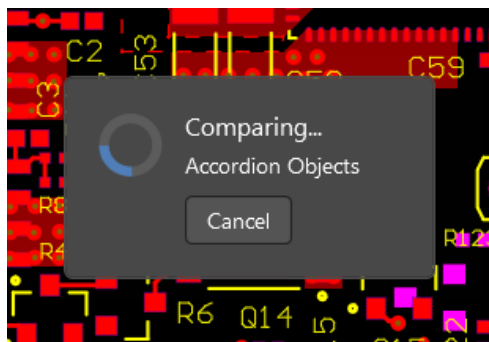


Figure 17. Compare progress dialog





44. When comparing is done, Altium Designer shows the differences between the local information and the Server information in the *PCB CoDesign* panel. The panel displays several sections:
- a) Comparing To
 - b) Changes
 - c) Conflicts

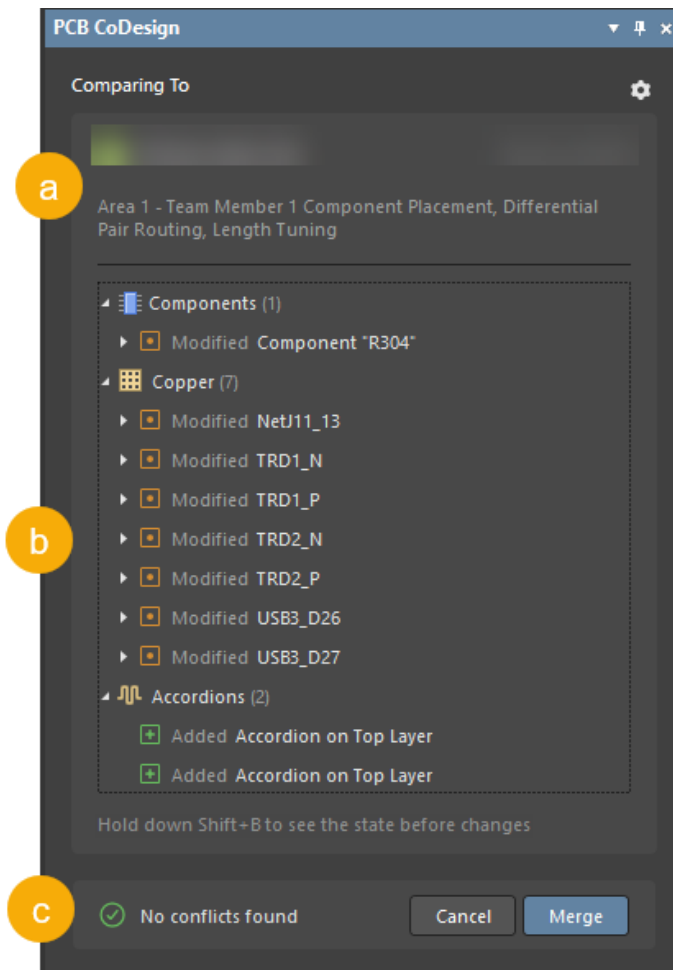


Figure 18. PCB Codesign merge information

Hint: When comparison results are shown in the PCB CoDesign panel, and another user saves the project to the Workspace, or you modify the PCB locally, a notification about the need to update the comparison appears in the PCB CoDesign panel, with a button to refresh the comparison data.

45. Expand the branches from the Copper objects select a net, for example, USB3_D27.
46. Select the information of the modified connection to see in the PCB the original and modified path for the net USB3_D27.

Hint: You can configure the colors for new, modified, and deleted Copper objects in the View Configuration panel.





7.3 Merging

47. Next, you will merge your local PCB with the PCB saved as head revision.

48. In the *PCB CoDesign* panel, Figure 19:

- Select the command **Merge** from the *PCB CoDesign* panel to merge your local PCB with the PCB from the server.
If Team Member1 and Team Member2 followed the training flow, no conflict will be detected.
- Wait until the merge is completed.
- Repour the polygons to take into account the merged changes with **Tools » Polygon Pours » Repour All**.
- The *PCB CoDesign* panel is updated, a new command **Save to Server** is available.

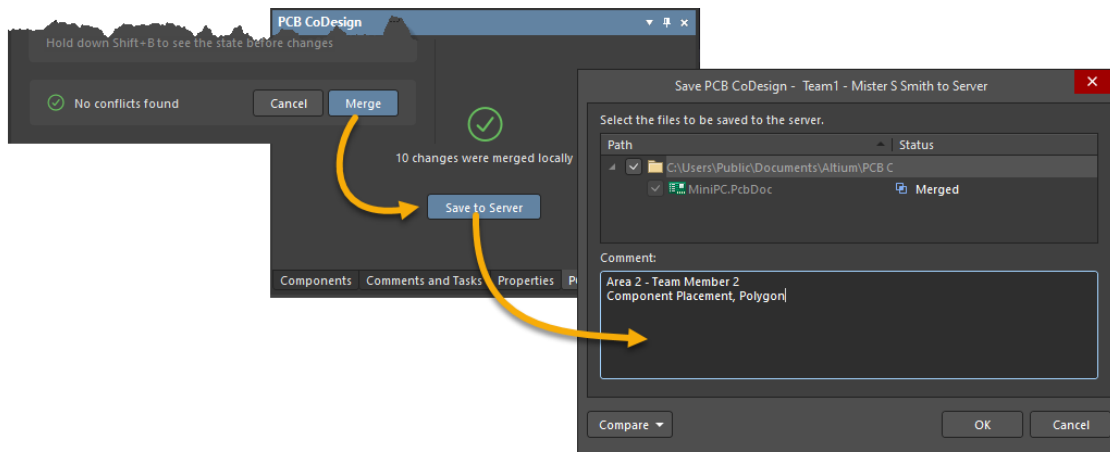


Figure 19. Save to Server

This is Step 5 of the flow, where Mia merged her changes with the changes from John.

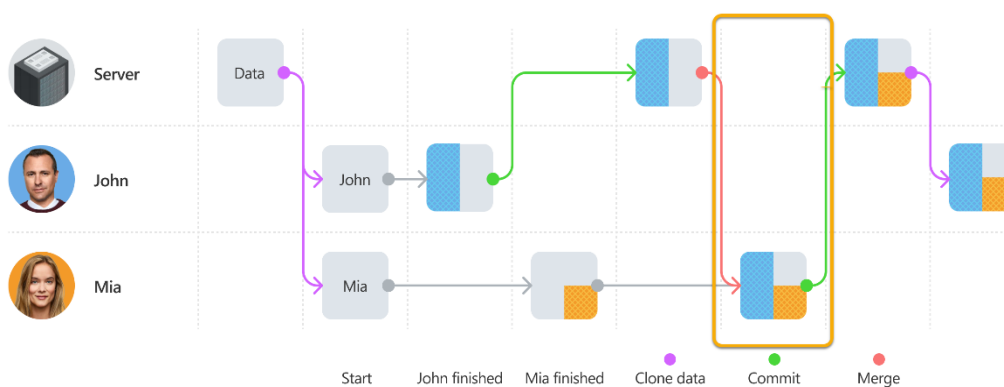


Figure 20. Training Flow - Step 5: Merge the PCB





7.4 Saving to Server

49. Now please select the polygon on the Top Layer and shelve it by using the context menu (Right click) and selecting **Polygon Actions » Shelf Selected**.
50. Next, you will save the modifications to the server, using one of the methods below:
 - a) Start saving in the *PCB CoDesign* panel by selecting **Save to Server**.
 - b) Right-click the icon in the *Projects* panel and select **Save to Server**.
51. In the *PCB CoDesign* dialog, as shown in Figure 19 above:
 - a) Select the **Save to Server**.
 - b) In the *Save PCB CoDesign* dialog, add a comment and select **OK** to save.
52. This is Step 6 of the flow, where Mia has uploaded the Data to the server.

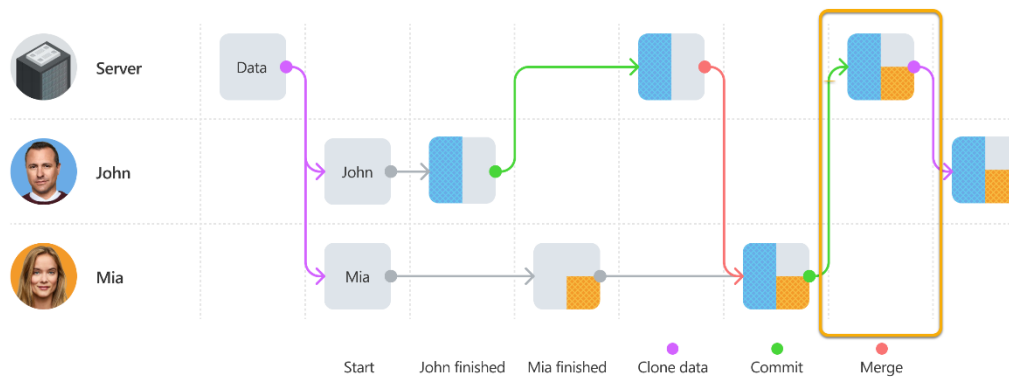


Figure 21. Training Flow – Step 6: Mia uploaded merged data





Team Member 1:

7.5 Downloading Head Revision

Now that **Team Member 2** merged and updated the design, you can update your design with the last information saved to the server from **Team Member 2**.

53. Open the *PCB CoDesign* panel, if not already open.

54. Altium detects that the local files are out of date, Figure 22.

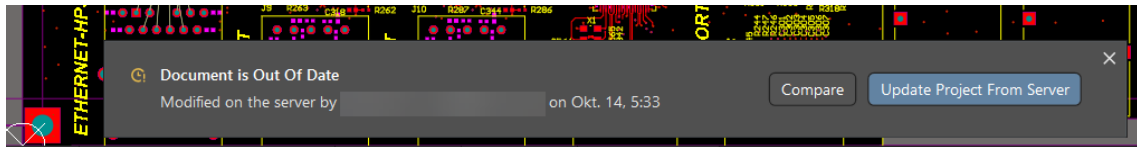


Figure 22. File is out of date

55. From the Pop-Up dialog:

- select the command **Compare and Merge** to update your local PCB file.
- Wait until the compare is finished.

56. In the *PCB CoDesign* panel, Figure 23.

- You can see the found differences between the local files and the head revision.
- Select the command **Update Project From Server**.
- Wait until the Altium Designer downloaded all files. When the download is finished, the *PCB CoDesign* panel is updated with the following information: *You have an up-to-date version of the project*.

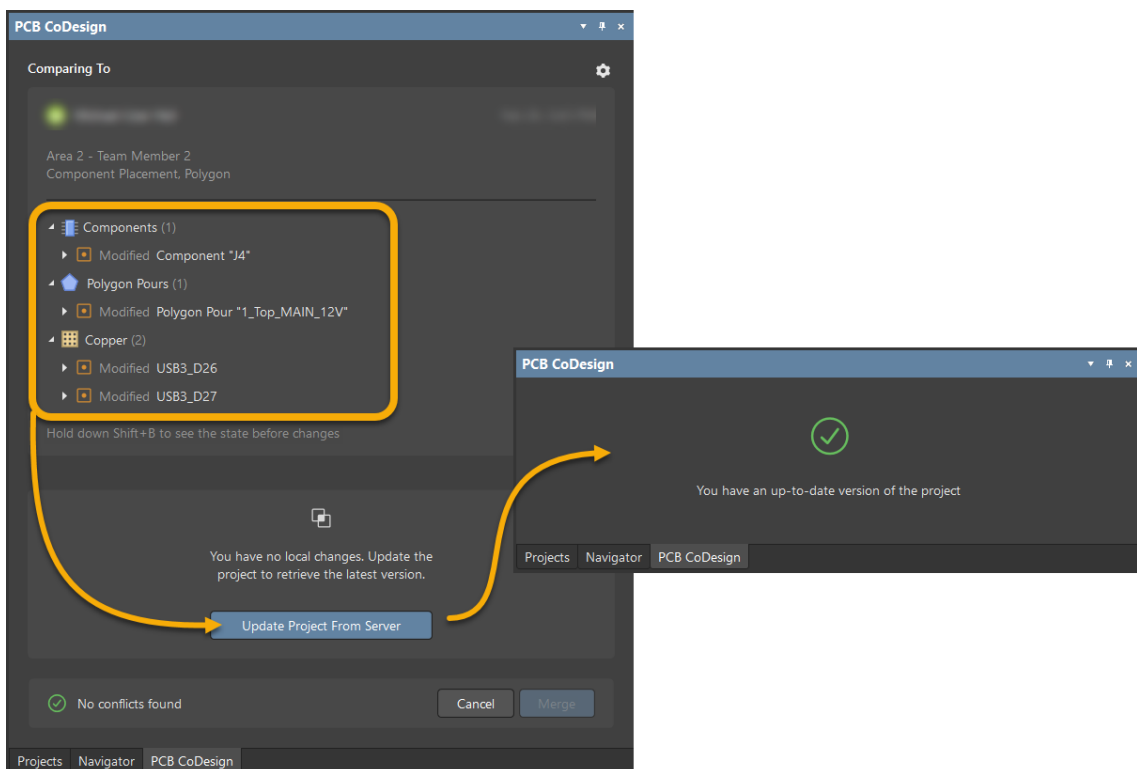


Figure 23. PCB CoDesign Update from server



57. We are in step 7 in the flow, where John downloads the latest head revision, uploaded from Mia.

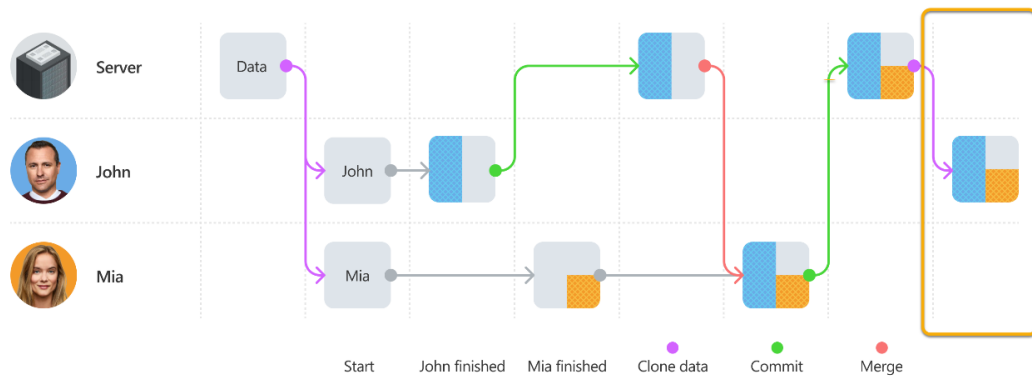


Figure 24. Training Flow – Step 7: John updates his local copy with the latest head revision

Team Member 1 and Team Member 2:

58. Updating the project from server is now complete.



Congratulations on completing the Module!

PCB CoDesign

from

**Altium Designer Advanced Training
with Altium 365**

Thank you for choosing **Altium Designer**

