Altium Designer

Advanced Training with Altium 365

ActiveRoute and Re-Routing

Altium.
TRAINING

Software, documentation and related materials:

Table of Contents

ActiveRoute and Re-Routing

1 Purpose

PCB routing is a repetitive process where existing routes are often adjusted as new routes are added and components are moved. Altium Designer interactive router has features like Loop Removal and Track Sliding, which help with re-routing while keeping track angles intact. The Push feature and Post-Route Glossing tools make routing more efficient and reduce errors, improving the quality and manufacturability of the PCB. The Retrace tool lets designers update track widths without re-routing.

ActiveRoute is an automated tool that efficiently routes multiple connections and allows designers to set a specific path for new routes. In this exercise, you will modify and re-route connections using ActiveRoute to demonstrate its capabilities.

2 Shortcuts
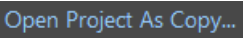
Shortcuts when working with ActiveRoute and Re-Routing

F1	Help
TAB	Open Properties panel during an active command
J » C	Jump Component
Shift+S	Single Layer Mode
U » T / Ctrl+W	Interactive Routing
Shift+D	Toggle Loop removal
CTRL+ALT+G	Gloss Selected
Alt+Selection	Selection of Unrouted connections
Ctrl+A	ActiveRoute
G	Grid
CTRL+S	Save Document

3 Preparation

1. Close all existing projects and documents.

2. Next, create a copy of the Training Project: `ActiveRoute and Re-Routing`.

3. Select **File » Open Project...** to open the *Open Project* dialog.

4. Enable the folder view button.

5. Navigate to the predefined Training Project `ActiveRoute and Re-Routing` `(Top\Projects\Altium Designer Advanced Training Course\...)`.

6. Select **Open Project as Copy...** Open Project As Copy....

7. In the new dialog *Create Project Copy:*

    a) Add your name to the project name: `ActiveRoute and Re-Routing - [Your Name]`.

    b) Add a description: `Altium Advanced Training - [Your name]`.

    c) Open the *Advanced* section.

    d) Select the **Ellipsis Button** from the *Folder* configuration to open the *Choose Folder* dialog.

       i) Select the folder with your name: `Project\For Attendees\[Your name]`.

       ii) Select **OK**.

    e) Change the **Local Storage** path if needed.

    f) Select **OK** to create the copy.

8. Wait until Altium Designer creates the copy of the project and opened the Project for you in the *Projects* panel, this may take up to 1 minute.

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

4 ActiveRoute

4.1 ActiveRoute Setup

You will configure ActiveRoute from within the PCB to allow for routing on multiple layers. In the default setup, where no layers are selected, the active layer is used.

> Hint: ActiveRoute is a plug-in that isn't installed by default. To download it, navigate to **Extensions and Updates…** menu located in the drop-down list under your profile icon in the top right corner of the interface. Next, select **Purchased**, scroll to **Software Extensions** section, find **ActiveRoute**, and click on the download icon ⬇.

4.2 Running ActiveRoute with a specified layer

9. Open the `SL1 Xilinx Spartan-IIE PQ208 Rev1.01.PcbDoc` document.

10. The PCB is opened in Single Layer mode, with Top Layer as active layer.

11. You can see that most of the routes have been placed, but some are still open, like U1 to HDR3.

12. Holding down the **ALT** key, use the left mouse button and sweep from right to left on all connection lines associated with `U1`, selecting them, as seen in Figure 1.

Figure 1. Select connection lines

13. Now, with 14 connections selected, select the **Panels** button in the lower right and open the *PCB ActiveRoute* panel. Note there are currently no layers enabled.

14. Activate the Bottom Layer, deactivate additional options like **Gloss Result** if they are active, see Figure 2 as reference.

Figure 2. ActiveRoute Panel with a Layer Selection

15. Select the **ActiveRoute** button at the top right in the panel **(or use Shift+A)** to run the ActiveRoute on the selected connections.

16. Active Routs starts, but after a few seconds the routing is finished without a result.

> Note: This routing will not show a result because U1 is a SMD component, Top Layer, and ActiveRoute was forced to use Bottom layer. ActiveRoute doesn't place or add vias. As a result, it could not route on the bottom side.

4.3 Running ActiveRoute in the Default Layer Mode

17. Holding down the **ALT** key, use the left mouse button and sweep from right to left on all connection lines associated with U1 selecting them, as seen in Figure 1.
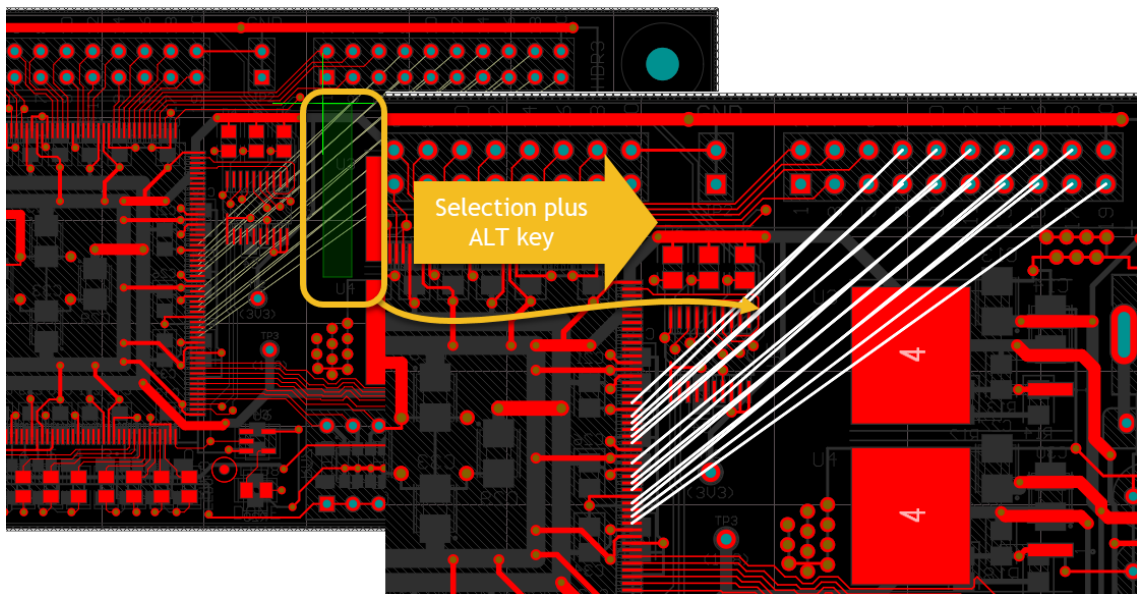
18. Now, with 14 connections selected again, select the **Panels** button in the lower right and open the *PCB ActiveRoute* panel. Deactivate the Bottom Layer. Now, ActiveRoute will use the active layer, for example the Top Layer, Figure 3.
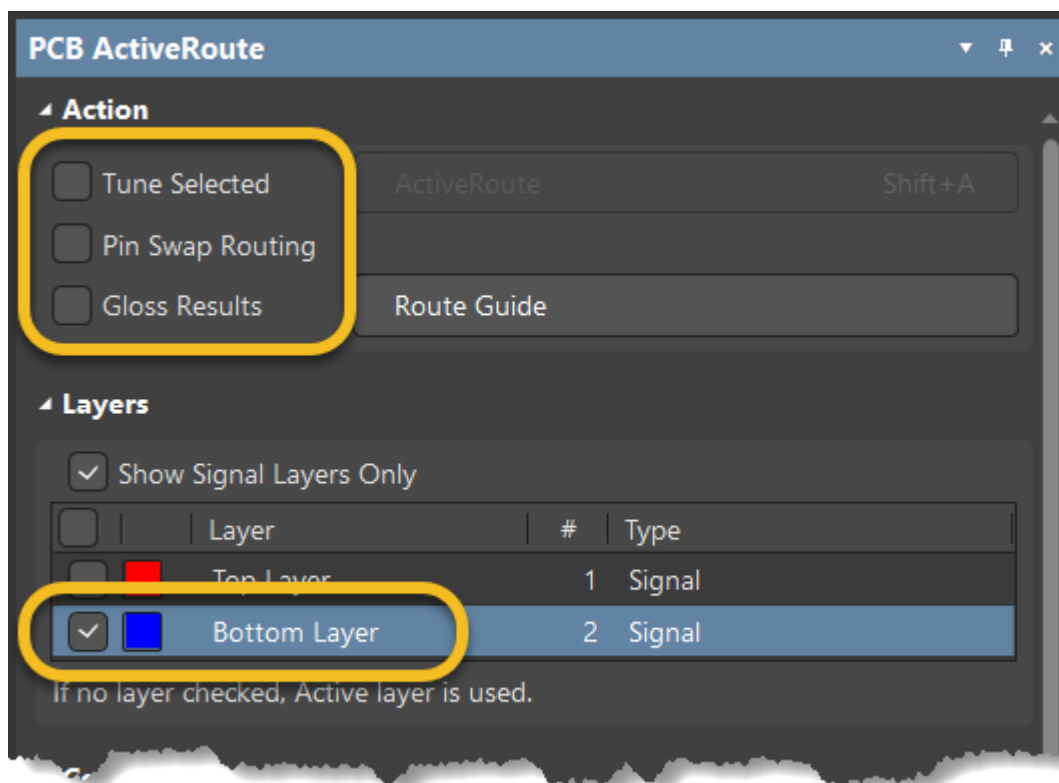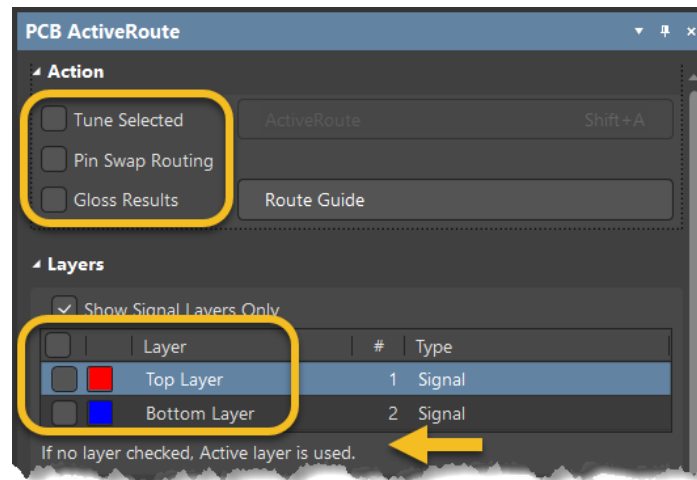

Figure 3. ActiveRoute Panel using the default layer

19. Select the **ActiveRoute** button at the top right in the panel (or use **Shift+A**) to run the ActiveRoute on the selected connections. This will route the 14 nets on the Top Layer, Figure 4.

Figure 4. ActiveRoute without Gloss

7

Altium.
TRAINING

4.4 Running ActiveRoute with Gloss

Next, we will use ActiveRoute for the same nets with active Gloss.

20. Before using the Gloss option, check a Preference setting and change it if needed. Use Figure 5 as reference.

    a) Open the *Preferences* ⚙ and navigate to the branch PCB Editor – Gloss and Retrace.

    b) Set Hugging Style to **45 Degree**.

    c) Set Gloss Parameter Effort to **Strong**.

    d) Close the preferences with **OK**.
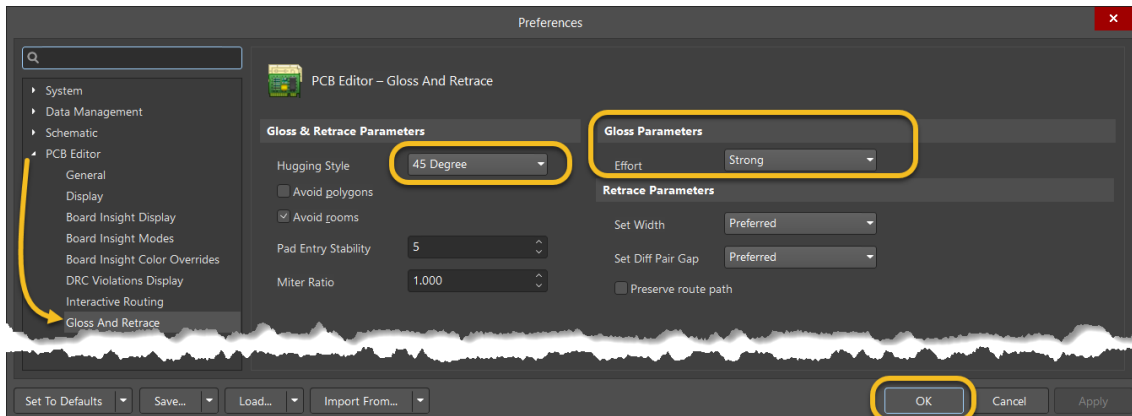
Figure 5. Preferences for Gloss and Retrace configuration

21. Back to the PCB, use Undo, **Ctrl+Z** to undo the last routing from ActiveRoute.

22. Holding down the **ALT** key, use the left mouse button and sweep from right to left on all connection lines associated with U1, selecting them.

23. Now, with 14 connections selected again, select the **Panels** button in the lower right and open the *PCB ActiveRoute* panel. Note there are currently no layers enabled.

24. Activate the option **Gloss Results,** Figure 6.
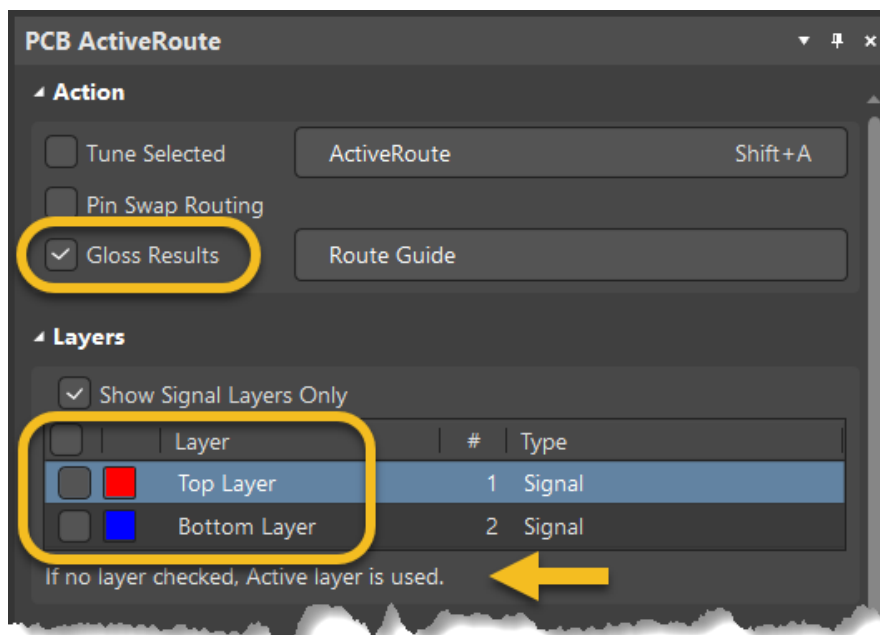
Figure 6. ActiveRoute with Gloss

25. Select the **ActiveRoute** button at the top right in the panel (or use **Shift+A**) to run the ActiveRoute on the selected connections. This will route the 14 nets on the Top Layer, Figure 7.
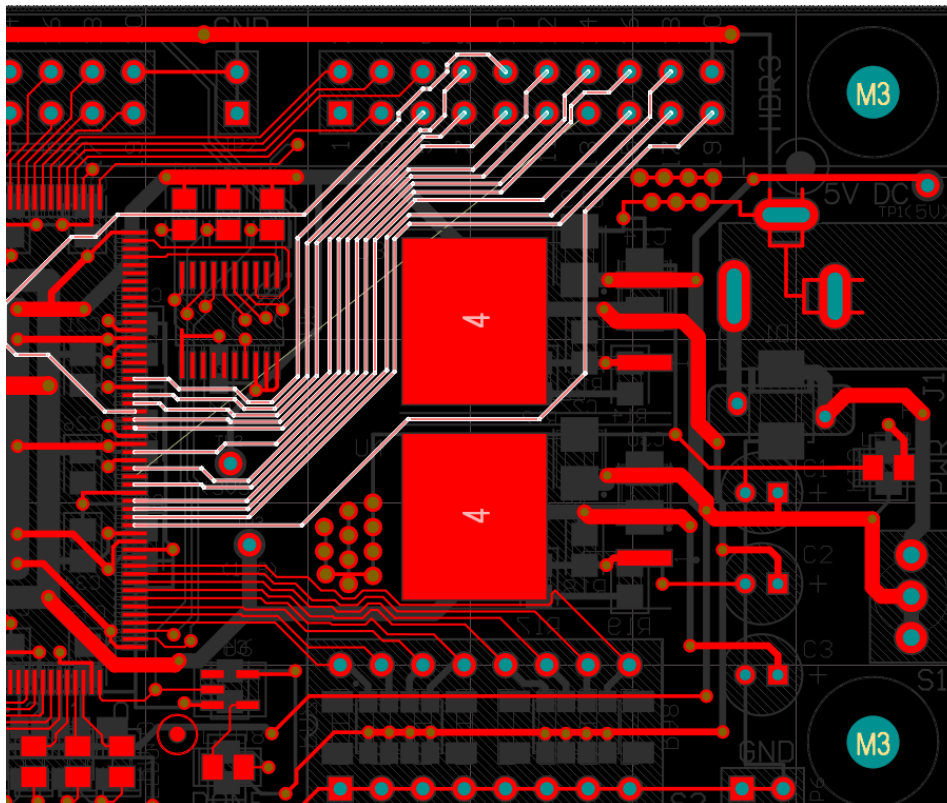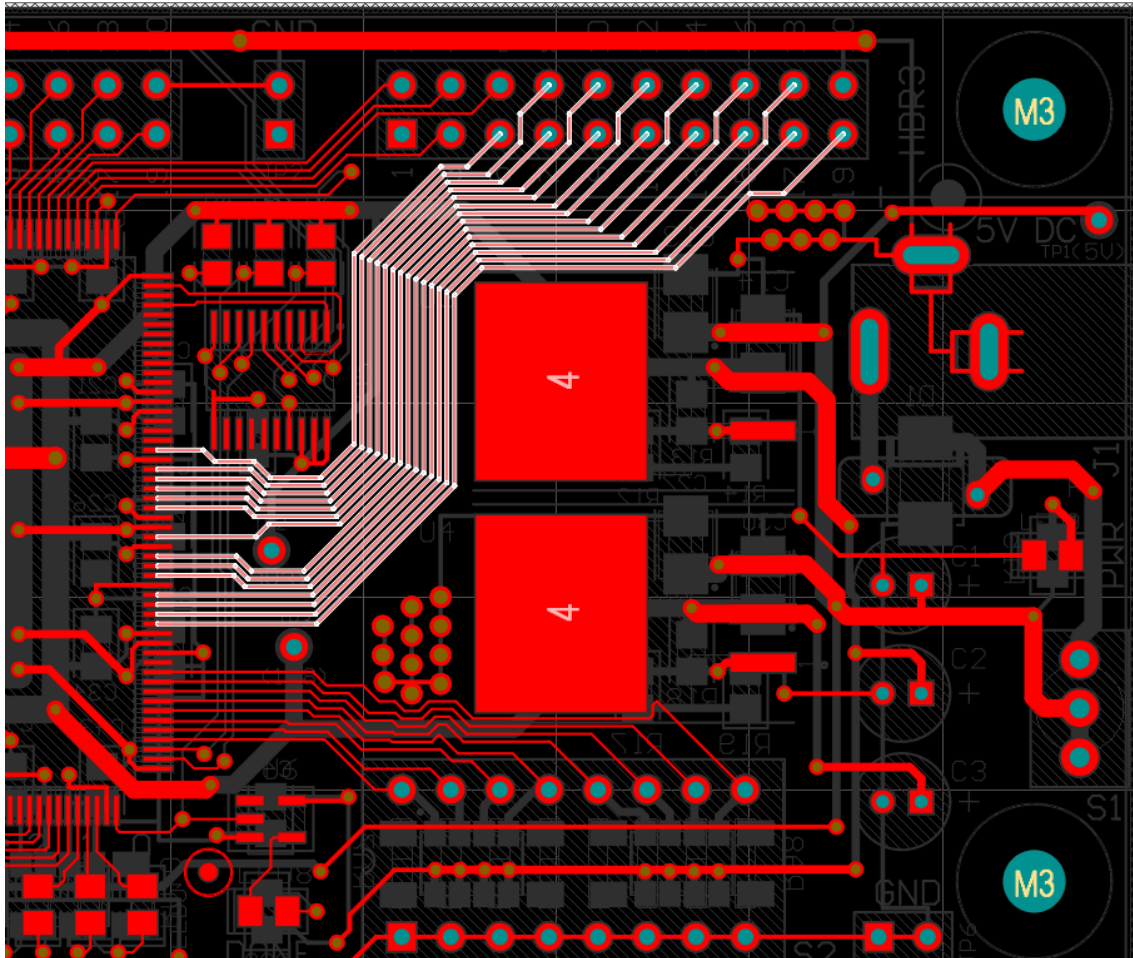

Figure 7. ActiveRoute with Gloss

4.5 Additional ActiveRoute Options

You will notice there is a second PCB in the project, called `Route_Guide_and_Tuning.PcbDoc`. In this PCB, you can further explore the power of the ActiveRoute feature when using the Route Guide and the Matched Length Tuning option.

26. Open the PCB document `Route_Guide_and_Tuning.PcbDoc`.

27. Holding down the **ALT** key, use the left mouse button and sweep from right to left on the lower bank of connections on U1, as shown in Figure 8.
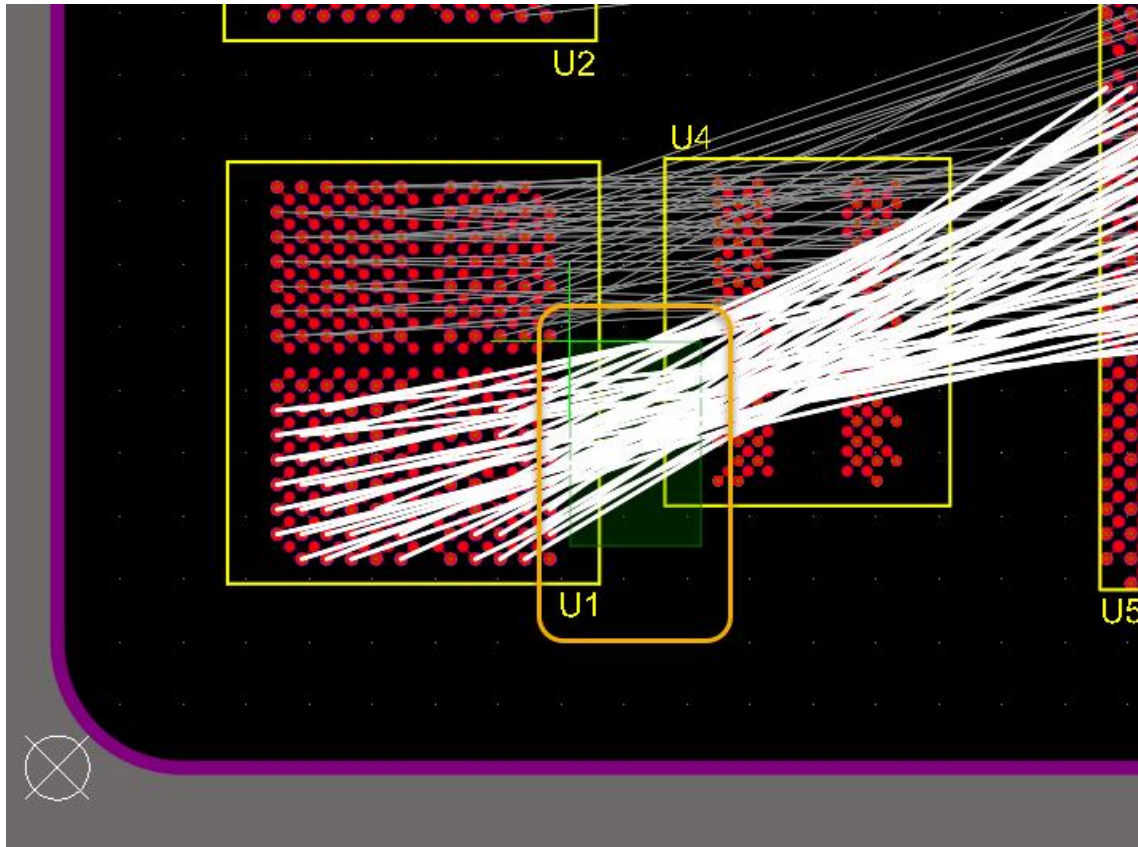
Figure 8. Select connections on U1

4.6 Route Guide

28. Next, draw the Route Guide as shown in Figure 9. A reference for the Route Guide is available on mechanical layer `99 Route Guide`.

    a) In the *PCB ActiveRoute* panel, select **Route Guide**.

    b) The unrouted connections are now attached to your cursor.

    c) Place the Route Guide path start points and corners with a left click. Finish the route guide Path with a right click.
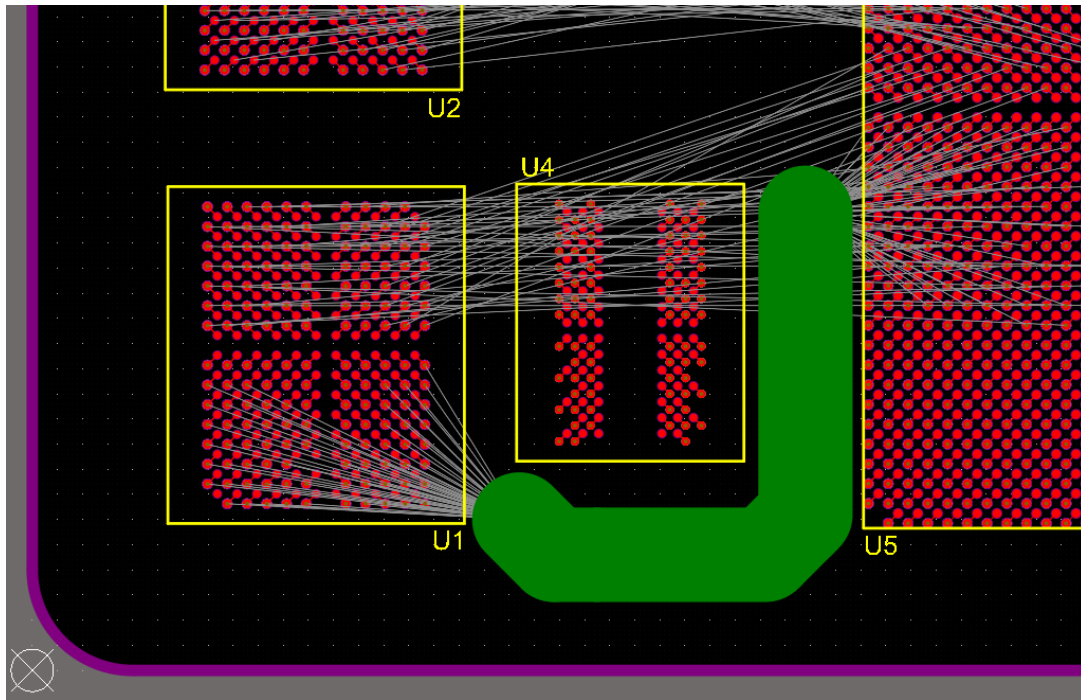
Figure 9. Route Guide

29. Next, select the **ActiveRoute (or use Shift+A)** from the *ActiveRoute* panel, the routed PCB should look similar to Figure 10.
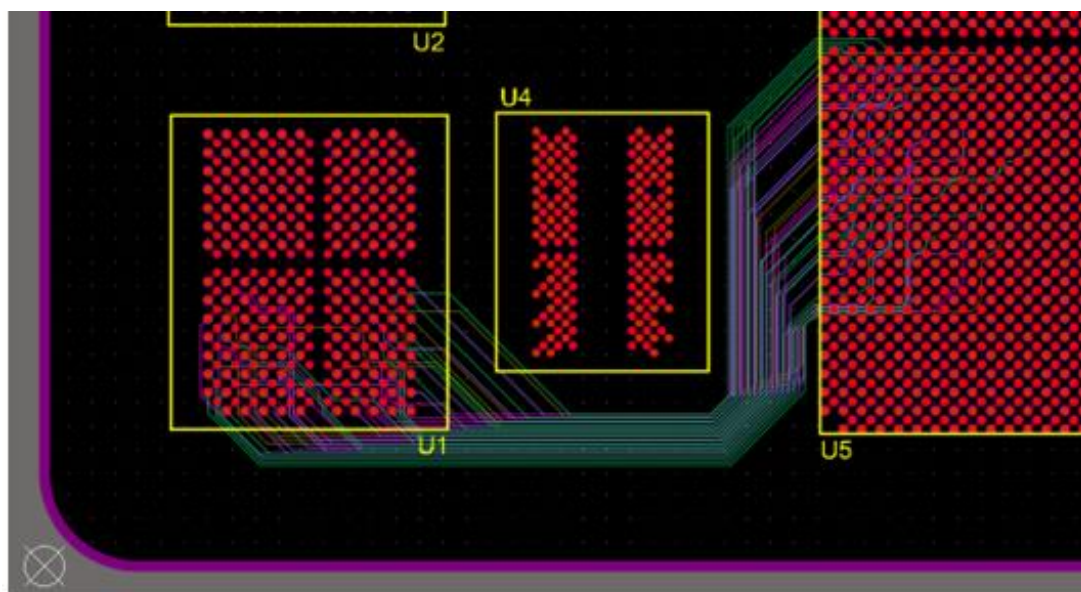
Figure 10. Routing using ActiveRoute Route Guide

4.7 Length Tuning

30. Open the *PCB Rules and Constraint Editor* **(D»R),** section *High Speed – Matched Lengths,* Figure 11. The rules MatchedLengths_4 to MatchedLengths_1 defines the length tuning for the four net classes from U2 to U5 we prepared for this exercise.

> Hint: More information on High Speed and Length Tuning can be found in module *Length Tuning Routing with Constraint Manager* and Altium **Online documentation**.
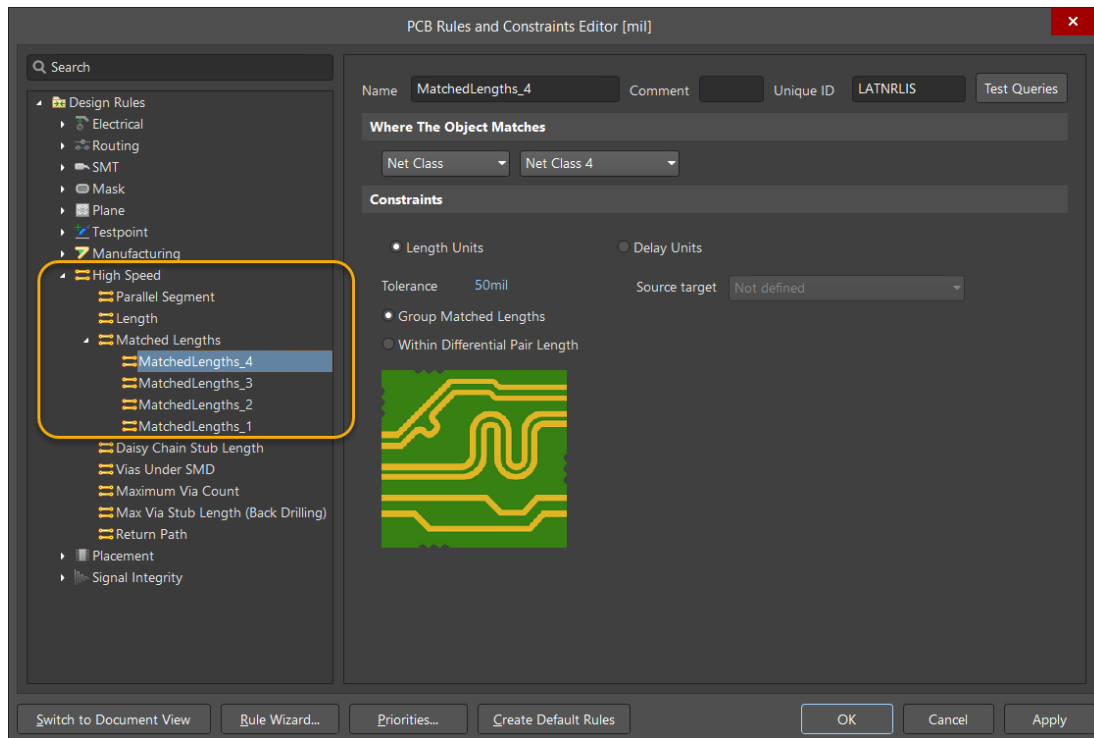
Figure 11. PCB Matched Length rule for ActiveRoute

31. **Close** the *PCB Rules and Constraint.*

32. Now let's explore ActiveRoute with length tuning. Using **ALT** key and using the left mouse button and sweep from right to left select the connections line between U2 and U5, as shown in Figure 12.
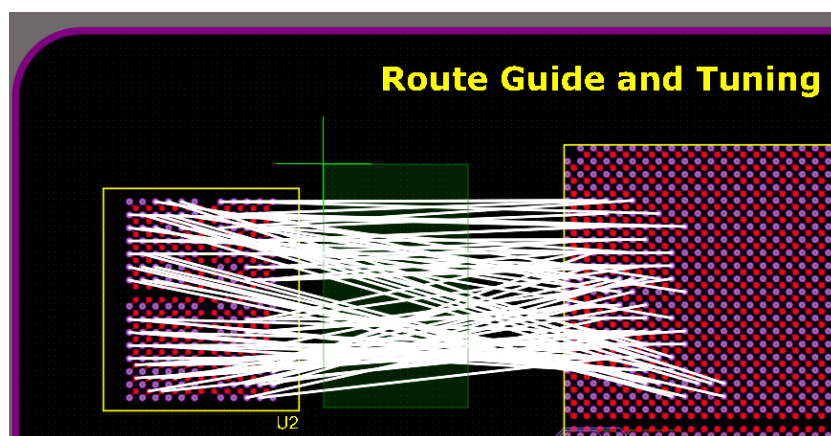

Figure 12. Connections selected

33. Next, ensure the tuning option is selected in the ActiveRoute panel, and then select ActiveRoute as shown in Figure 13. The finished routing and tuning should look similar to Figure 14.
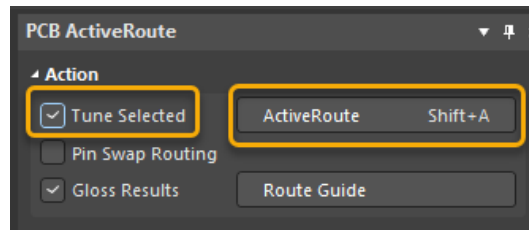
Figure 13. Tuning Option
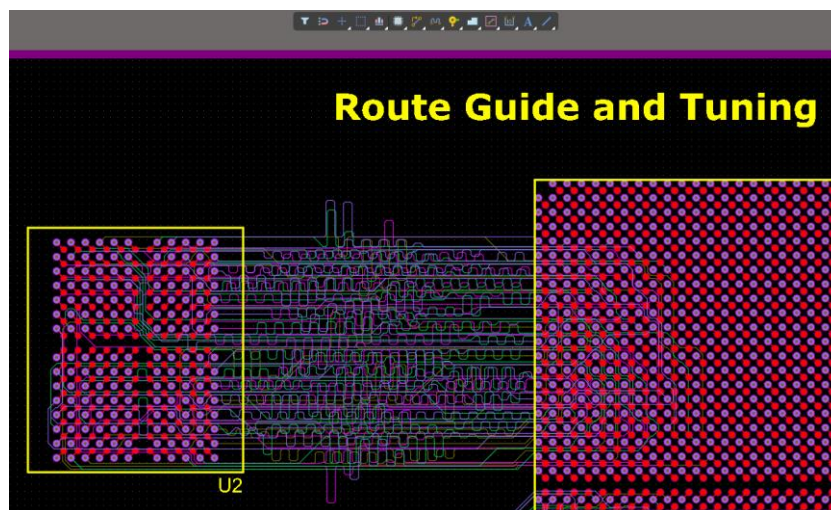
Figure 14. Completed routing with length matched tuning

34. When ready, save and close the PCB `Route_Guide_and_Tuning.SchDoc`.

Note: After ActiveRoute is finished, please remember to check and, if needed, update the length of the nets. Not every net may automatically achieve the desired length during the routing process. If any nets don't meet the specified target lengths necessary for proper tuning, you need to adjust them.

5 Re-Routing

5.1 Loop Removal

35. Next, you will continue to modify the PCB `SL1 Xilinx Spartan-IIE PQ208 Rev1.01.PcbDoc`.

36. You will now modify some of the routing of the connector HDR2. To jump to HDR2, press **J-C**, type in `HDR2`, and select **OK**. Navigate to the left-hand side of HDR2, zooming in as necessary.

37. If you aren't currently in Single Layer Mode (meaning only the Active Layer should be shown in color), switch to single layer mode by pressing the **Shift+S** keyboard shortcut. You can toggle Single Layer Mode by pressing **Shift+S** again. Another method to verify if you are in Single Layer Mode is in the Heads-up display, where **(Single)** is visible, as seen in Figure 15.

> Hint: This requires the Heads-Up option for **Current Layer** to be enabled under the Insight Modes section of **Preferences - PCB Editor - Board Insight Modes**.

Figure 15. Heads-up display showing Single Layer Mode

38. To edit an existing route, select **Route » Interactive Routing**. The connection that you will modify is HA2 (Figure 16).

   a) To start re-routing, click on the vertical section of the connection, where it passes between pad 1 and pad 3 of HDR2 (click anywhere in this segment).

   b) Route down and diagonally across to meet HA2, above where it connects to U1 pad 191, as shown in Figure 16. When you meet the old route, the redundant segments are removed.

> Hint: Loop removal doesn't require any special action from you if it's active in the Preferences. Run the Interactive Routing command, click to start re-routing of an existing route, define the new path, then come back to meet the old, existing routing. As soon as you create the new path, any redundant track segments, forming the old loop, are automatically removed.
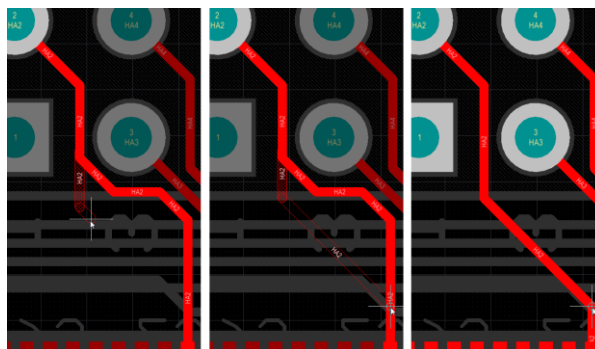

Figure 16. (a) Click on the existing routing, (b) then re-route to define a new path for connection HA2, (c) when you finish, the old loop is removed

> Hint: Loop removal can be used to re-route multi-layer connections layers, including vias. Note that loop removal shouldn't be used on nets that include intentional loops, such as ground nets. Because the entire net is analyzed for redundant loops, intentional loops on another part of the board can be removed. For these nets, Loop Removal can be selectively disabled, by double-clicking on the net name in the *PCB* panel.

5.2 Gloss

For ease of viewing and editing, ensure **Single Layer Mode** is active. You will use Glossing to clean up the routing between U1 and S2. Glossing is a sophisticated set of algorithms developed specifically to produce cleaner routing and pad entries, that respect the intent of the applicable design rules. Glossing attempts to reduce the path length, improve the shape of corners, and reduce the number of corners, resulting in neater routing created from fewer segments.

39. The track segments for nets SW0 to SW7 are just above U6, Figure 17. Press **J-C**, type in `U6`, and select **OK**.
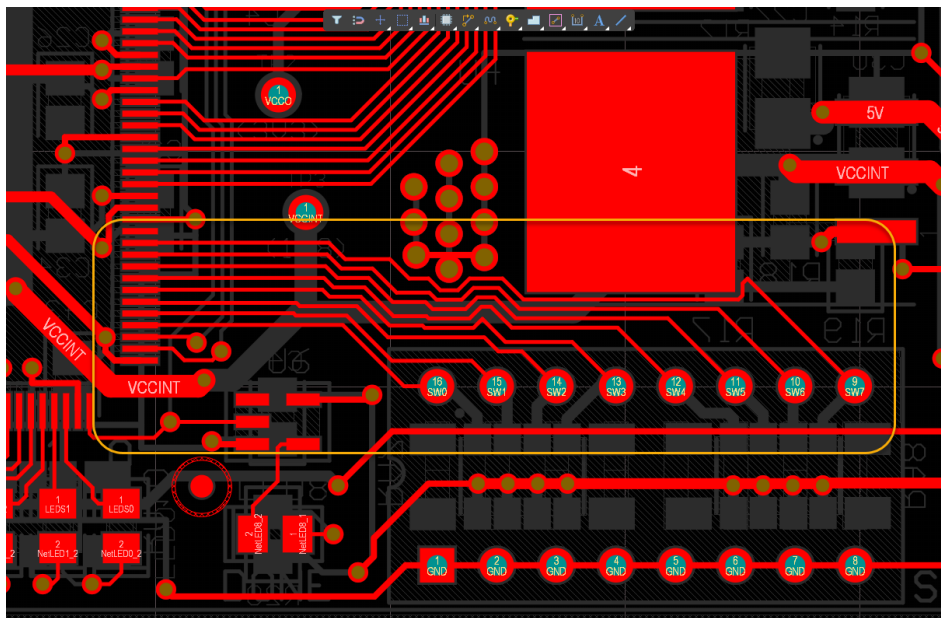

Figure 17. View between U1 and S2

40. For ease of selection, select the filter in the *Active Bar*, Figure 18. Next, follow the steps below:

    a) Select **All Objects** to toggle all options off
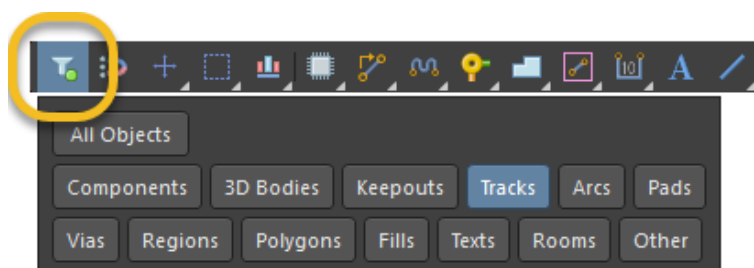
    b) Select **Tracks**.

Figure 18. Active Bar selection filter for Tracks only

41. Drag a selection box from lower right to upper left to select the parallel track segments, leaving U1 and routing to S2. This will select the first track segment of those parallel routed nets, refer to Figure 19 below.
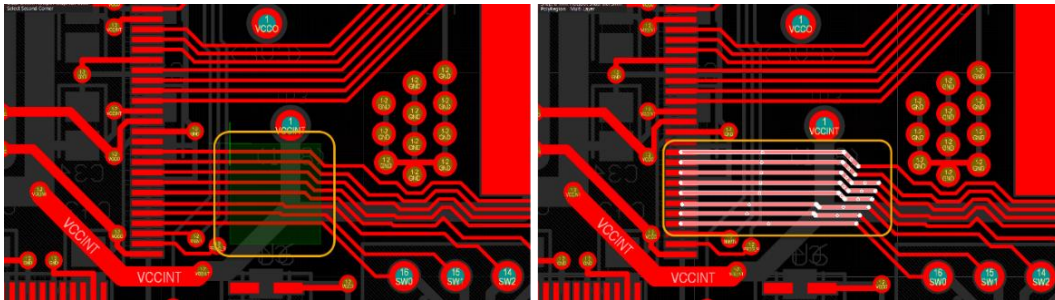

Figure 19. Selecting top routes

Hint: Drag the selection window from left-to-right, you will select all objects that fall completely within the bounds of the selection area. This behavior is the same as using the **Edit » Select » Inside Area** command. Drag the selection window from right-to-left, you'll select all objects that fall completely inside the selection area or are touched by its boundary. This behavior is the same as using the **Edit » Select » Touching Rectangle** command.

42. Press the **TAB** key to extend this selection to all connected tracks. Press the **TAB** key three more times to toggle through the selection modes and note how the selection changes.
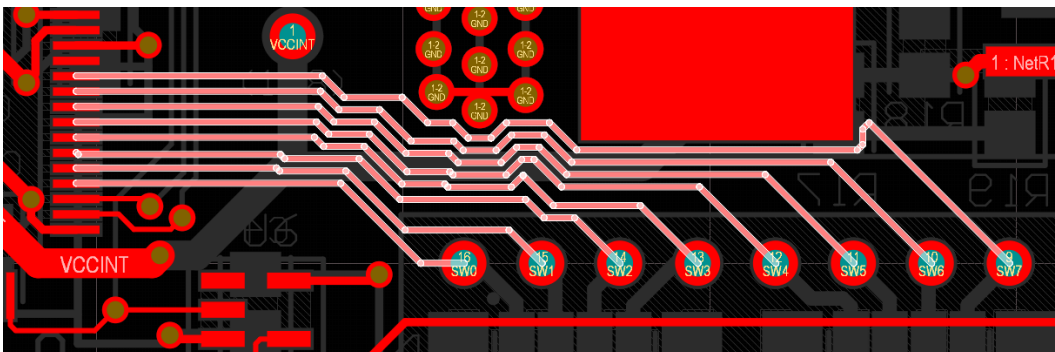

Figure 20. Results of pressing Tab to extend selection

43. With all of the tracks selected, as shown in the Figure 20 above, execute the Glossing Feature **Route » Gloss Selected** or press **CTRL + ALT + G**.

44. The Glossing tool will straighten up and remove unnecessary track vertices in a routed path, Figure 21.
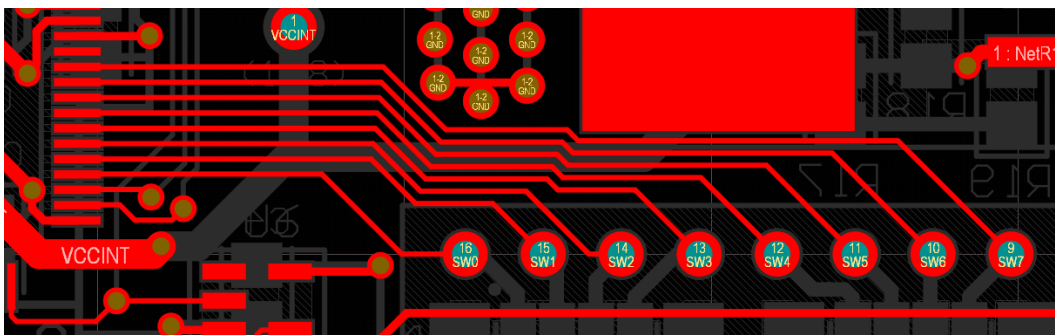

Figure 21. Results of applying Gloss command

5.3 Drag

The ability to drag a track segment is the next feature for modifying the existing routing.

45. To slide a track segment, first click on it to select it. Next, follow the steps below:

   a) Click once to select a diagonal segment of `SW0`, as shown in Figure 22.

   b) Position the cursor so it is over the selected track segment, it will change to a double headed arrow ✥, as shown in Figure 22.

   c) Click and hold the left mouse button, then move the cursor to the left. The diagonal track segment will slide, and the angles to the connected segments will be maintained. As you continue to move the diagonal segment to the left, the SW0 routing will automatically wrap around the VCCO via that is in the way.

   d) Position the segment anywhere to the left of its original location and release the mouse button.
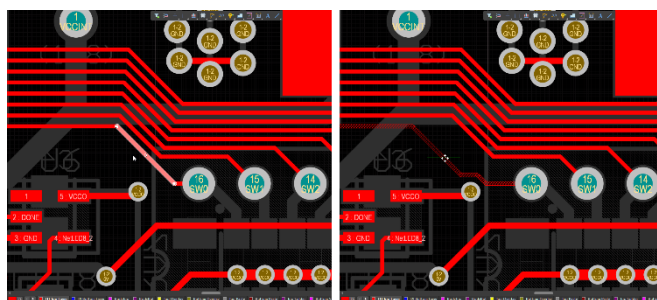

Figure 22. Click to select a segment, then click and hold to slide that segment

46. You can also slide multiple selected segments. The easiest way to select multiple segments is to use the Select Touching Line command. To launch this command, press **S** to activate the *Select* sub-menu, then press **L** to run the **Touching Line** command, Figure 23:

   a) Position the cursor below the diagonal segment of SW1, click once to start the selection line, then move the cursor up so that the selection line touches all the diagonal segments for SW1 through SW7. Then, click a second time to perform the selection, take a look at Figure 23 for a better idea of how this works.
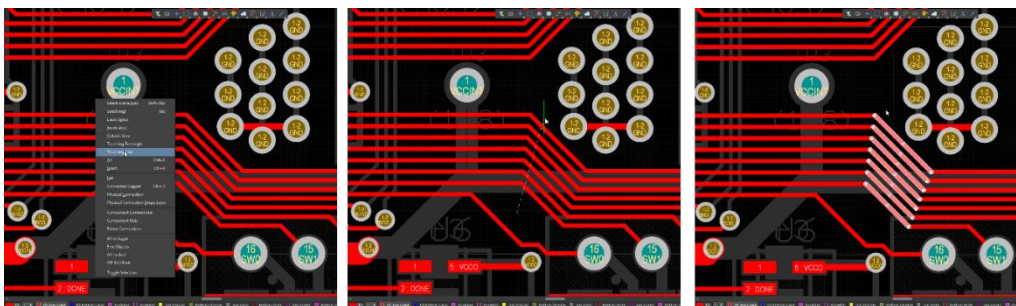
Figure 23. Press S, L then click to select segments touching that line

   b) Hover the cursor over any of the selected segments. The cursor will change to the double-headed arrow ✥.

   c) Click and hold to grab the segments, then move the mouse to slide them to the left. Note that you won't be able to position them on top of any existing track segments.

   d) Release the segments so that they're adjacent to SW0.

> Hint: Track sliding also uses the interactive router, so it can also push and shove existing tracks, vias, and jumper pads.

47. Alternatively, to slide multiple track segments, you can slide one and use it to push other segments. To slide all of the diagonal segments, click once to select SW7, as shown in Figure 24 below.

a) Click and hold on the selected segment, then move the mouse to the left. As you move the mouse, take note of the *Properties* panel. It displays the current mode, which is set to Push. If it's not in Push mode, press **Shift+R** to cycle to Push mode.

> Note: This requires that the *Routing Conflict Resolution* mode is enabled in the Interactive Routing Modes section of your Preferences: **Preferences » PCB Editor » Interactive Routing**.

b) The moving segment will push the other SW segments. Move all segments a small distance to the left, then release the mouse button.

Figure 24. Use the sliding segment to push other segments

> Hint: You can also launch the sliding mode without selecting the track segment first. To do this, hold the Ctrl key and then click on the segment. Start to move the segment and release the Ctrl key.
>
> The behavior for selected and unselected Tracks and Vias, if a Move or Drag is done, can be controlled in **Preferences » PCB Editor » Interactive Routing**, **Dragging section**.

5.4 Retrace

Retrace works exactly as its name implies, running along the selected routes, updating them to the current rule specifications. Because it does this on an individual net or pair level, it will attempt to maintain clearances, but is not able to push surrounding routes if more room is required. In this situation, the rule updates are applied only to those route segments that don't create a violation.

> Hint: The behavior we describe is based on the default settings within **Preferences » PCB Editor » Gloss and Retrace**. If you have other Preference settings, the behavior may differ. For the training, we suggest using the default settings.

For a demonstration of the retrace options, you will update the 5V net.

48. First, let's change some of the settings for better visibility:
    a) Deactivate the Single Layer Mode (**Shift+S** or *View Configuration* panel, tab *View Options*).
    b) Activate the Layer Set **Signal Layers** (**Design » Manage Layer Sets » Signal Layers**).

49. Now, select the 5V net:
    a) Open the *PCB* panel.
    b) Select **Nets** from the drop down.
    c) Activate **Select, Zoom, Clear Existing**. Also, feel free to select Mask or DIM.
    d) Select the net class `Power`.
    e) Select the Net `5V`.
    f) In the PCB, you can now see the selected 5V net, as seen in Figure 25.
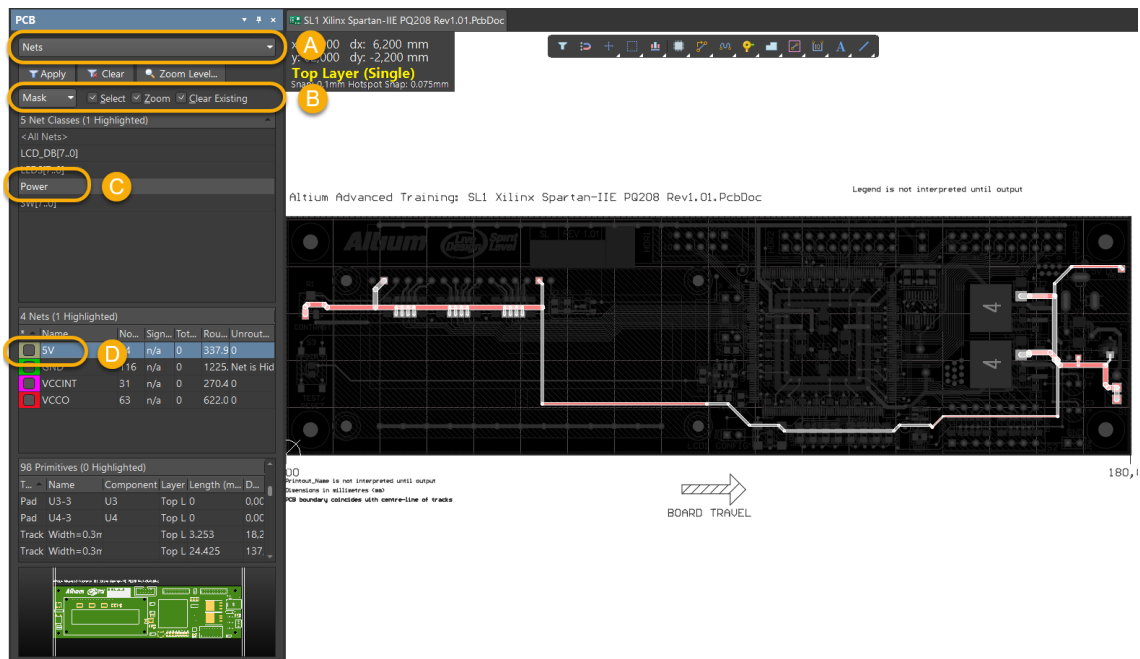

Figure 25. 5V Net

50. The 5V net is routed with different Track Width (0,3mm / 0,5mm / 1,0 mm).

    a) With **Jump » Component (J » C »** `RA3`), jump to RA3.

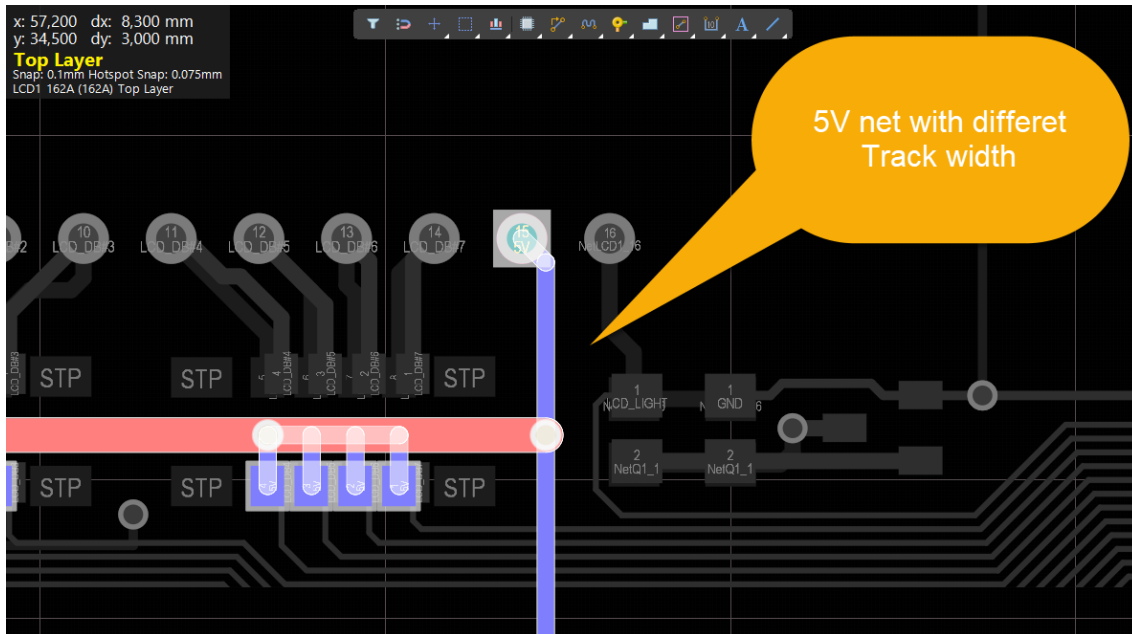   b) Zoom to that area to see the different track width for the 5V net, as shown in Figure 26.

Figure 26. 5V net between RA3 and R2 with different track width

51. Next, open the *PCB Rules and Constraint Editor*. Find and expand the **Routing** – **Width** and the rule **Width_Power** (**Design » Rules** or **D » R**) to see the rule for the Power Nets. The current values are: Minimum `0.3mm`, Preferred `0.3mm`, Maximum `2.0mm`.

52. Change the Preferred value to `0.7mm`, as seen in Figure 27.

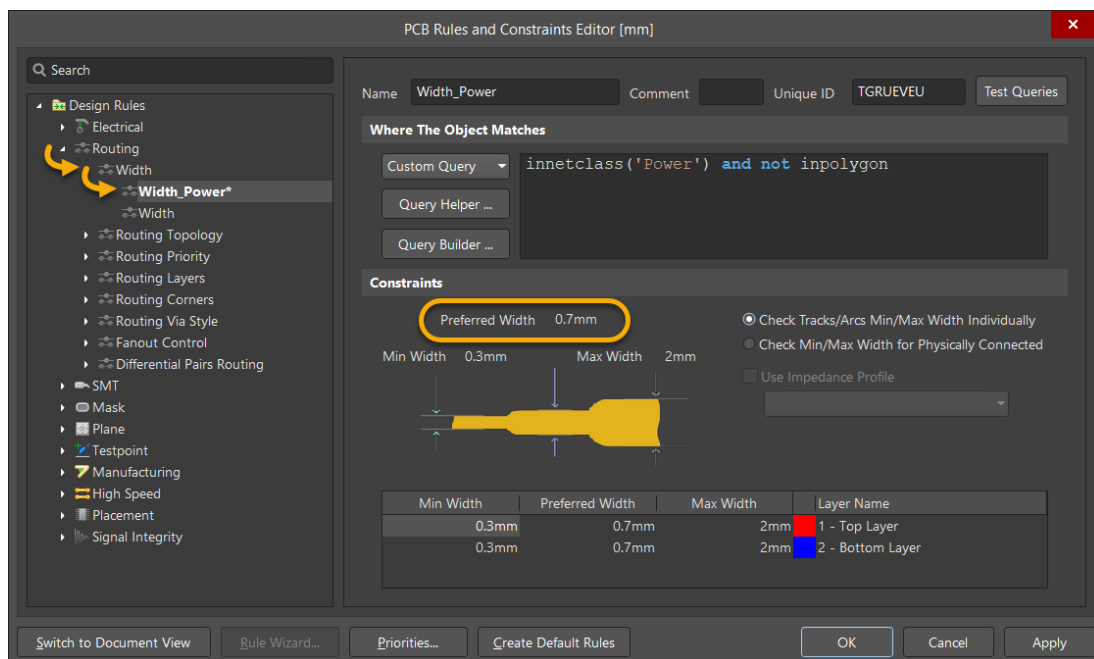53. Close the *PCB Rules and Constraint Editor* with **OK**.

Figure 27. Width rule with new Preferred Width of 0.7mm

54. To update the trace width of the 5V net, following the new default width, you will now run Retrace.

    a) Execute the command **Route » Retrace Selected (U » C)**.

    b) All selected tracks are updated with the new default width of 0,7mm, if the clearance would allow it, as seen in Figure 28.
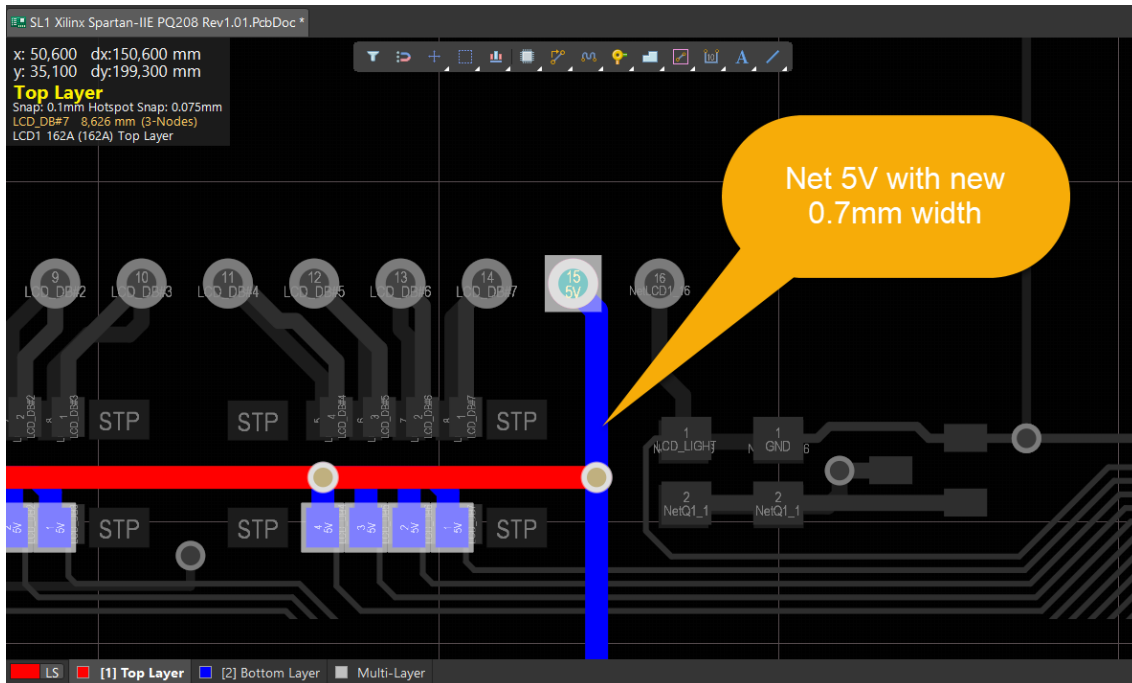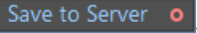

Figure 28. 5V net with new 0.7mm width

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

56. Save the modifications to the server:

    a) In the *Projects* panel, next to the Project name, you can 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 `ActiveRoute and Re-Routing - [Add Your Name] - Finished`.

       ii) Select **OK**.

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

Congratulations on completing the Module!

ActiveRoute and Re-Routing

from

Altium Designer Advanced Training
with Altium 365

Thank you for choosing **Altium Designer**

Altium
TRAINING