



Altium Designer

Advanced Training with Altium 365

Using xSignals in a High-Speed Design with
Constraint Manager

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

Using xSignals in a High-Speed Design with Constraint Manager	3
1 Purpose	3
2 Shortcuts	3
3 Preparation	4
4 Overview	5
5 Project Options for xNets	6
6 xSignals with Serial Termination	7
6.1 Creating the xNets with Constraint Manager	7
6.2 Creating xNet class with the Constraint Manager	8
6.3 Creating xSignals and xSignal Class from the xNets	9
6.3.1 xSignals	9
6.3.2 xSignal Class	13
7 Updating PCB	15
8 Length Tuning for xSignals with Serial Termination	16
8.1 PCB Panel	16
8.2 Initial Length Tuning	17
9 xSignals from a Net with T-Junctions	20
9.1 Creating xSignals from a Net with T-Junctions	20
9.2 Rules for xSignals from Net with T-Junction	23
9.3 Length Tuning with Accordion	24
10 Unions and Length Tuning	25
11 Length Tuning with Trombone	26
11.1 Deleting Accordion Length Tuning	26
11.2 Length tuning with Trombone	26
11.3 Modifying Trombone	28





Using xSignals in a High-Speed Design with Constraint Manager

1 Purpose

The designer's role is to convert their design needs, like keeping the route length within limits for timing, into specific design rules. For instance, designers create a Length rule to ensure timing and a Matched Length rule to spot timing issues.

The designer thinks of signals based on their function, such as routing an address signal from a connector to memory devices with a specific layout and termination resistors. Even if an address signal like A0 goes through a resistor, the designer still considers it as A0 after the resistor.

On the other hand, the PCB editor views signals as a series of connected pins, or nets. When a series termination resistor is added, it splits the signal into two different nets, complicating the specification of design rules like Length and Matched Length.

An xSignal is a designer-defined path between two nodes, which could be within the same net or across different nets separated by a component. xSignals help apply relevant design rules like Length and Matched Length during tasks like interactive length tuning.

2 Shortcuts

Shortcuts used when working with Using xSignals in a High-Speed Design with Constraint Manager

D » R	Constraint Manager (PCB)
D » G	Constraint Manager (SCH)
U » R	Interactive Length tuning
U » P	Interactive Differential Pair Length Tuning

Note: Altium recommends starting with the "Creating Classes from Schematic with Constraint Manager" module before this one. It provides an overview of the Constraint Manager.


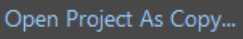

Caution: The Constraint Manager is available only with an Altium Designer Pro License, not with the Standard Subscription. For more information, check our [Online Documentation](#).

Caution: Using the Constraint Manager replaces setting Rules and Classes with Directives in the schematic for nets.





3 Preparation

1. Close all existing projects and documents.
2. Next, create a copy of the Training Project: Using xSignals in a High-Speed Design with Constraint Manager.
3. Select **File » Open Project...** to open the *Open Project* dialog.
4. Enable the folder view button .
5. Navigate to the predefined Training Project Using xSignals in a High-Speed Design with Constraint Manager (Top\Projects\Altium Designer Advanced Training Course\...).
6. Select **Open Project as Copy...** .
7. In the new dialog *Create Project Copy*:
 - a) Add your name to the project name: Using xSignals in a High-Speed Design with Constraint Manager - [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*.

Note: Due to file name length limits, the names of files in the workspace might differ from those in this document. We use CM for Constraint Manager and IP for Impedance Profile.



4 Overview

9. Open the schematic document Using xSignals in a High-Speed Design with CM.SchDoc. You will see two separate training circuits: a group of bus signals with a serial resistor and multiple components with T-Junctions connections.

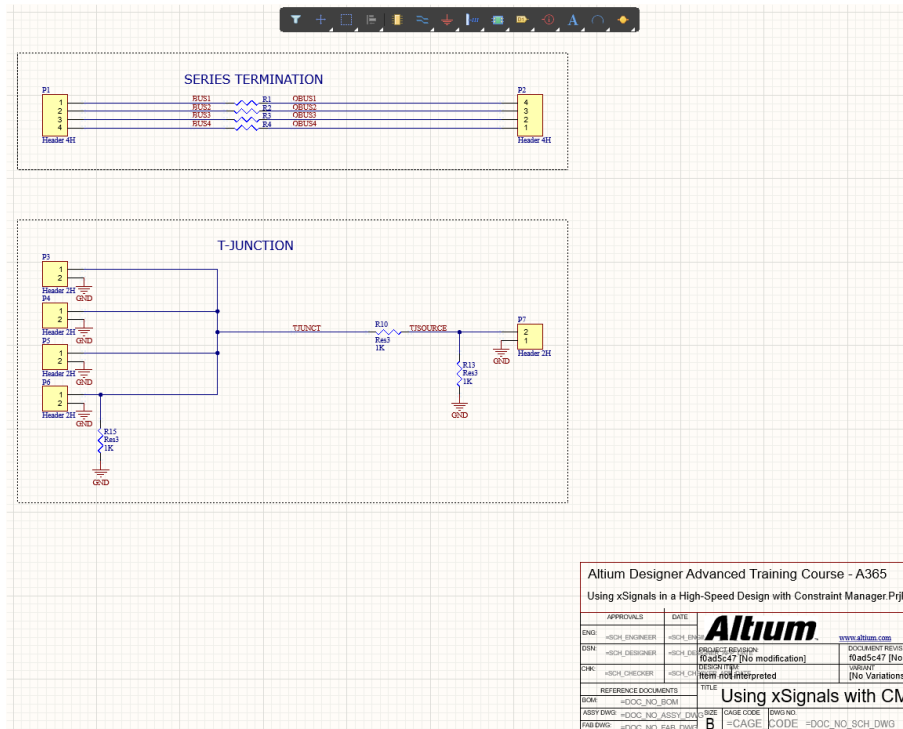


Figure 1. Example document for xSignals

10. Open the Using xSignals in a High-Speed Design with CM.PcbDoc PCB document. The component placement and routing are already complemented, Figure 2

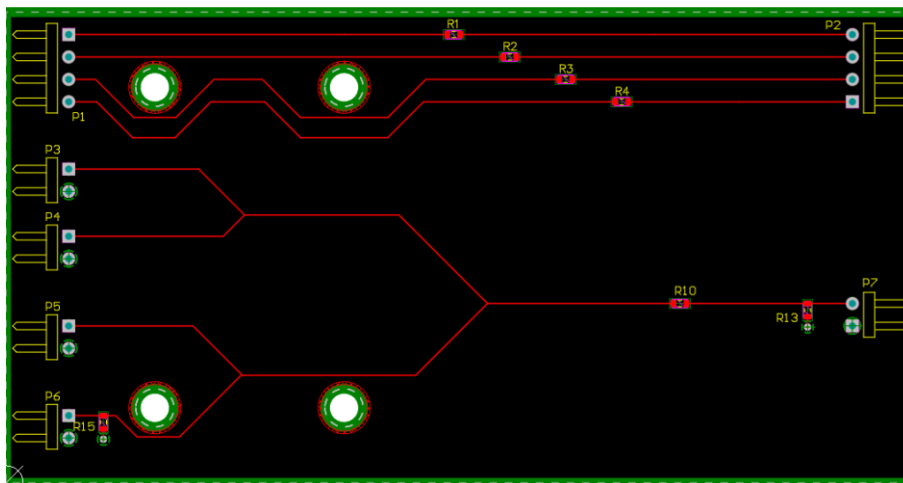


Figure 2. PCB without xSignals and Length Tuning

11. For the Series Termination and T-Junctions sections, you will do the following tasks:
 - a) Create the xSignal Nets, xSignals and the xSignal Class.
 - b) Create PCB Design Rules for Matched Net Lengths and Length.
 - c) Length tune the xSignals.



5 Project Options for xNets

12. Switch back to the Schematic document.
13. Open the *Project Options* with **C » O** and select the tab *Options*.
14. At the lower right, there's a section *xNets Creation*, Figure 3:
 - a) By default, the characters **U** and **X** are available, add the character **P** to support the headers from the training example.
 - b) Close the Project Options with **OK**.

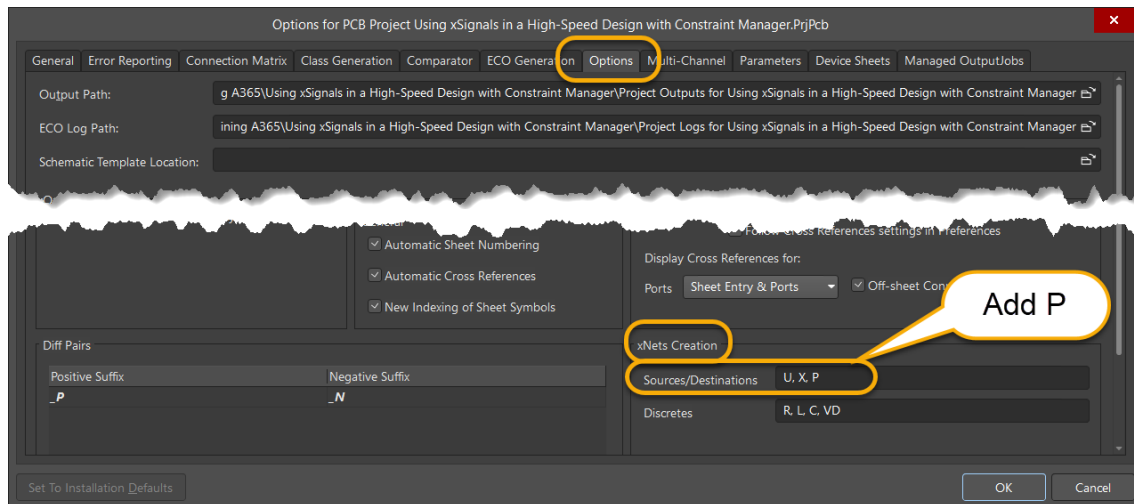


Figure 3. Project Options - Add Character P



6 xSignals with Serial Termination

Let's start with creating new xSignals between components P1 and P2 with the Constraint Manager, opened from the Schematic.

6.1 Creating the xNets with Constraint Manager

15. With the Schematic as active document: Open the *Constraint Manager* with **D»G** or by selecting **Design » Constraint Manager...**

Hint: For details about Constraint Manager GUI, the difference between Constraint Manager opened from SCH (Clearances, Physical, Electrical) and Constraint Manager opened from PCB (Clearances, Physical, Electrical, All Rules), see the Module Creating Classes from Schematic with Constraint Manager.

16. Select the view *Electrical* Electrical.

17. Create the xNets following the steps below:

- Select the eight Signals Bus1..Bus4 and OBUS1..OBUS4.
- By hovering over the selected net, right-click and select **xNets » Create xNets From Selected Nets**.
- The *Electrical* view is updated to show the created xNets.

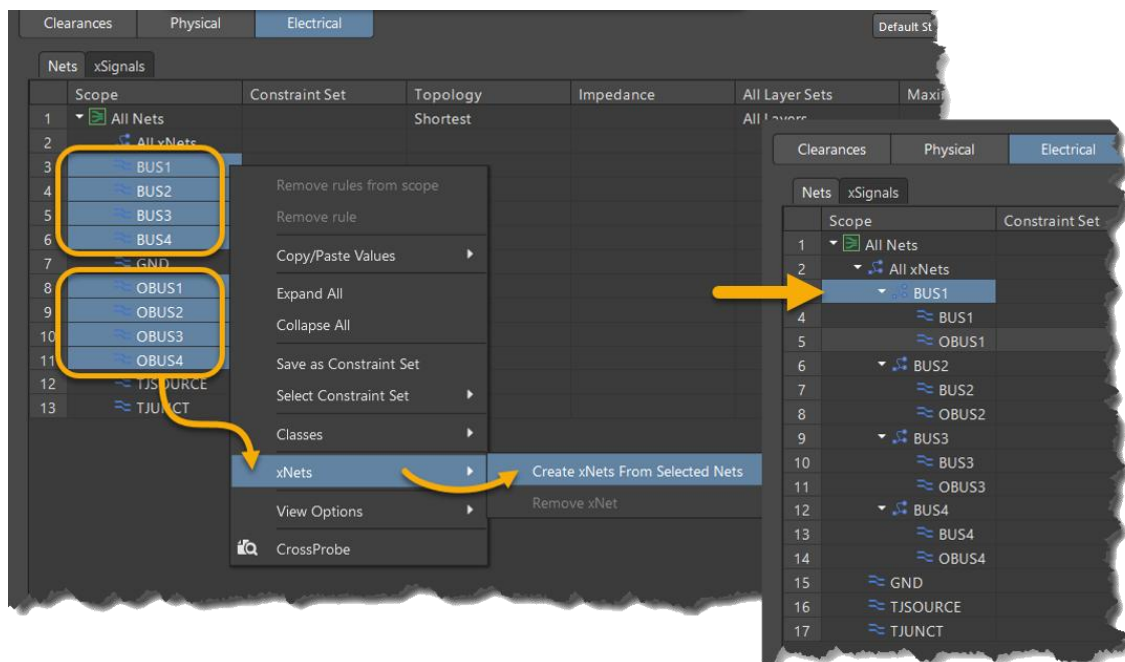


Figure 4. Create xSignal for BUS / OBUS

6.2 Creating xNet class with the Constraint Manager

18. With an xNet Class, you can group one or more xNets, making it possible to apply design rules and settings specific to that group.

19. Change the *Constraint Manager* view to the *Physical* Physical view, Figure 5.

- Select the four xNets `Bus1..Bus4`.
- By hovering over the selected xNet, right-click in the grid area and select **Classes » Add Class » xNet Class**.
- In the new dialog *Add Class*, as illustrated in Figure 6 on the following page:
 - Add the Class name `Series-Term`.
 - Transfer the four xNets from the Non-Members to the Members area >>.
 - Close the dialog with **OK**.
- Back in the *Constraint Manager*, the view is updated to present the new xNet class.

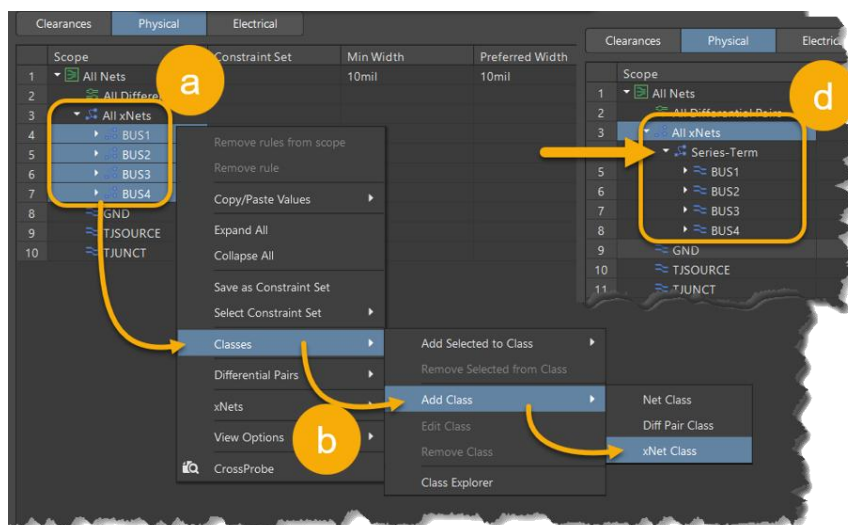


Figure 5. Create xNet Class – Figure A, B, D

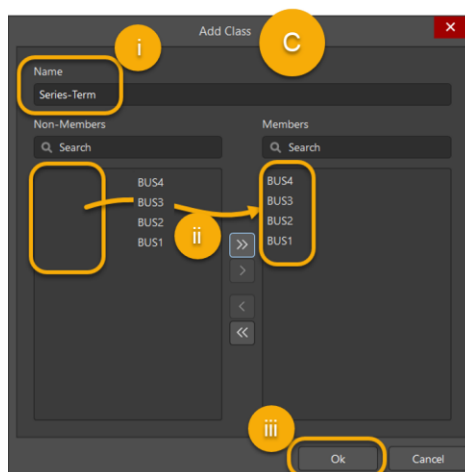


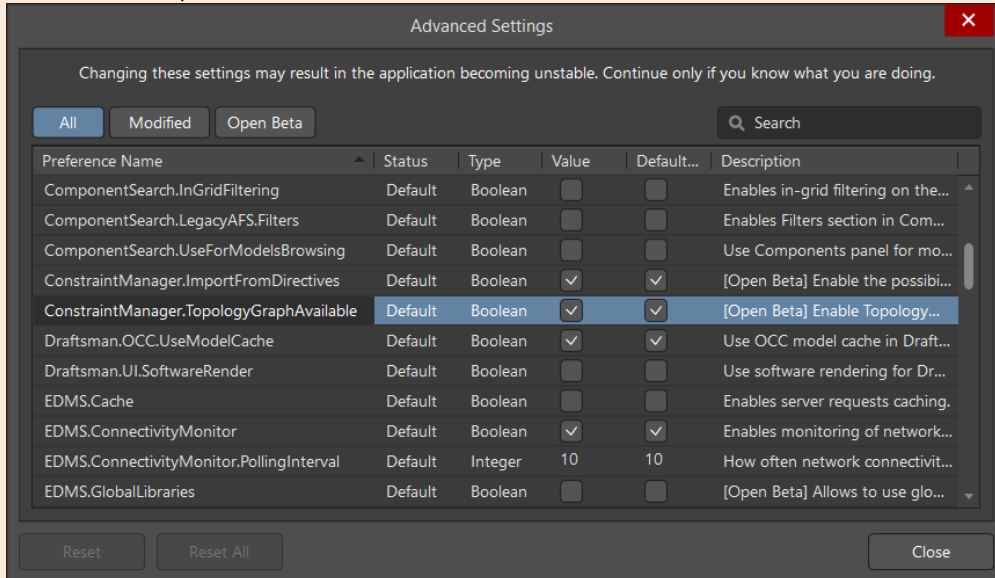
Figure 6. Create xNet Class – Figure C

Hint: You can also create the xNet class inside the *Electrical* view.
To create the class inside the *Electrical* view, select the xNets, right-click anywhere in the grid area, and select **Classes » Add Selected to Class**.

6.3 Creating xSignals and xSignal Class from the xNets

Caution: The topology graph functionality is used for this training.

This functionality is available when the `ConstraintManager.TopologyGraphAvailable` option is enabled in the [Advanced Settings dialog](#). (**Preferences » System » General » Advanced...** button). When the option is disabled, the table user interface is used.



6.3.1 xSignals

20. Inside the Constraint Manager, follow the steps below, Figure 7:

- Choose the Electrical **Electrical** view.
- From the xNet class `Series-Term`, select the cell `Topology` of the xNet `BUS1`.
- At the bottom, the topology graph area, you can see the graphic for the topology `Shortest`, change the topology to **Custom**.

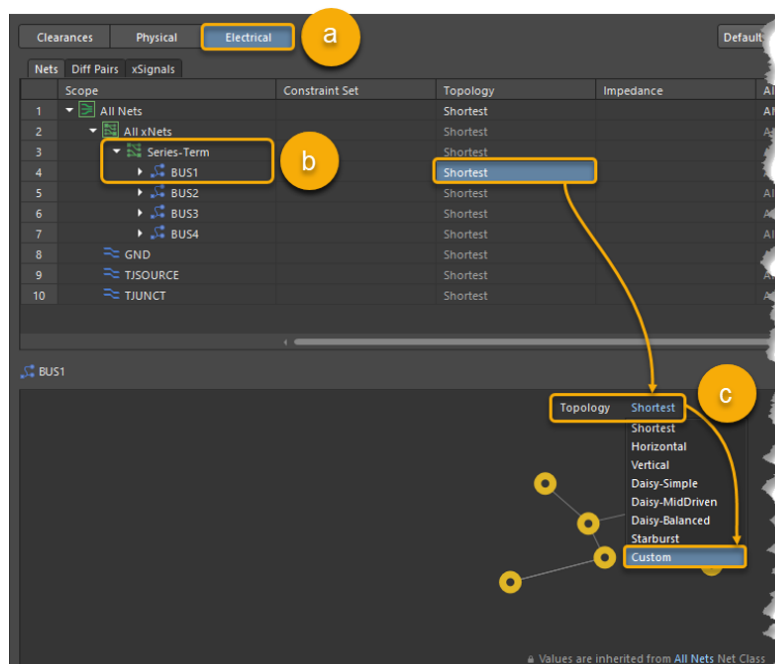


Figure 7. Change Topology for xNet



21. In the topology graph area, Figure 8:

- a) Right-click in the currently empty topology graph area at the bottom left.
- b) Select the command **Add Node » P1-1** for the first node of the topology graph and place it on the left side.
 - i) Right-click again in the topology graph area and place the second node – R1.
 - ii) Right-click again in the topology graph area and place the last node – P2-4.
- c) You can see a chain of 3 nodes: P1-1 – R1 – P2-4. If needed, select a node and move it to a new position.
- d) Activate the checkboxes for the *Proposed xSignal* you see on the right-hand side.
- e) Save the modifications **File » Save** or **CTRL+S**.

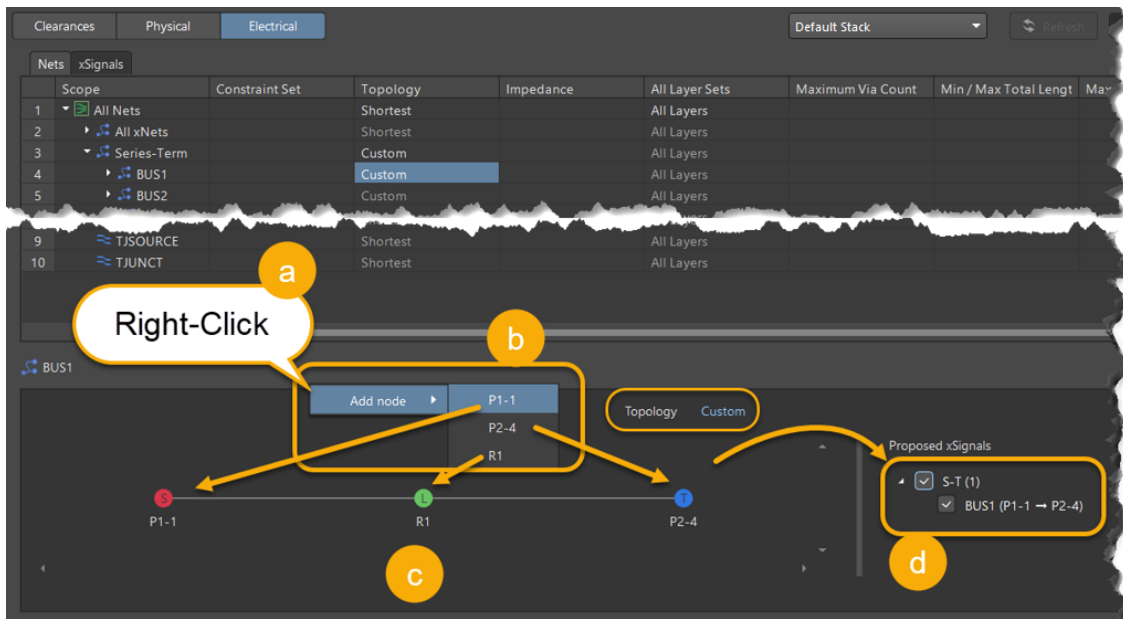


Figure 8. Signal Topology and xSignal

Hint: You could now repeat Step 16 for the remaining three xNets. Instead of repeating the previous step, create a Constraint Set and assign this to xNets Bus2...Bus4.





22. Hovering over the xNet name **BUS1**, Figure 9:

- a) Right-click and select the command **Save as Constraint Set**.
- b) Add the Name **Series-Term** and select **Create**.
- c) The **Properties** panel shows the constraint set you've just created.

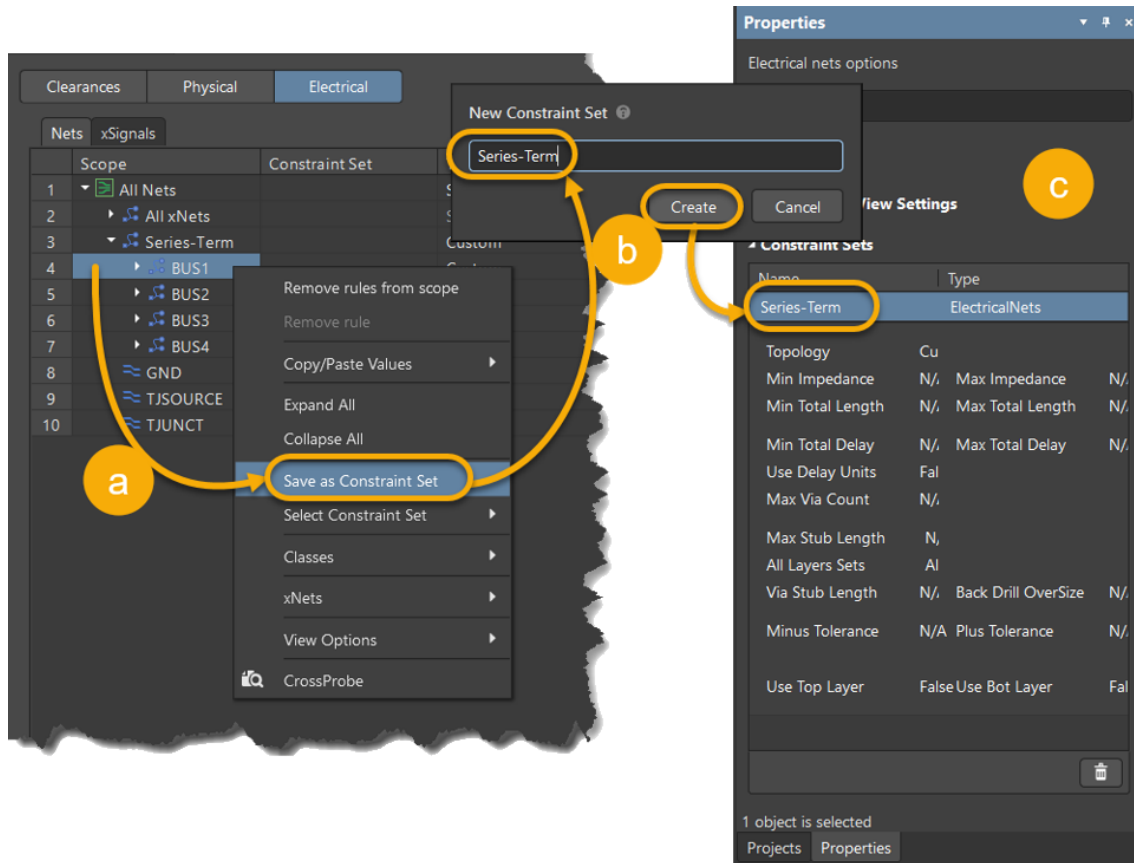


Figure 9. Create a Constraint Set





23. Next, let's reuse the constraint set for the three xNets Bus2...Bus4, Figure 10.

- a) Select the three xNets: Bus2 up to Bus4.
- b) Right-click on the xNet name `BUS2` and select the command **Select Constraint Set » Series-Term**.
- c) The cells from the column *Constraint Set* are updated with a selected constraint set.

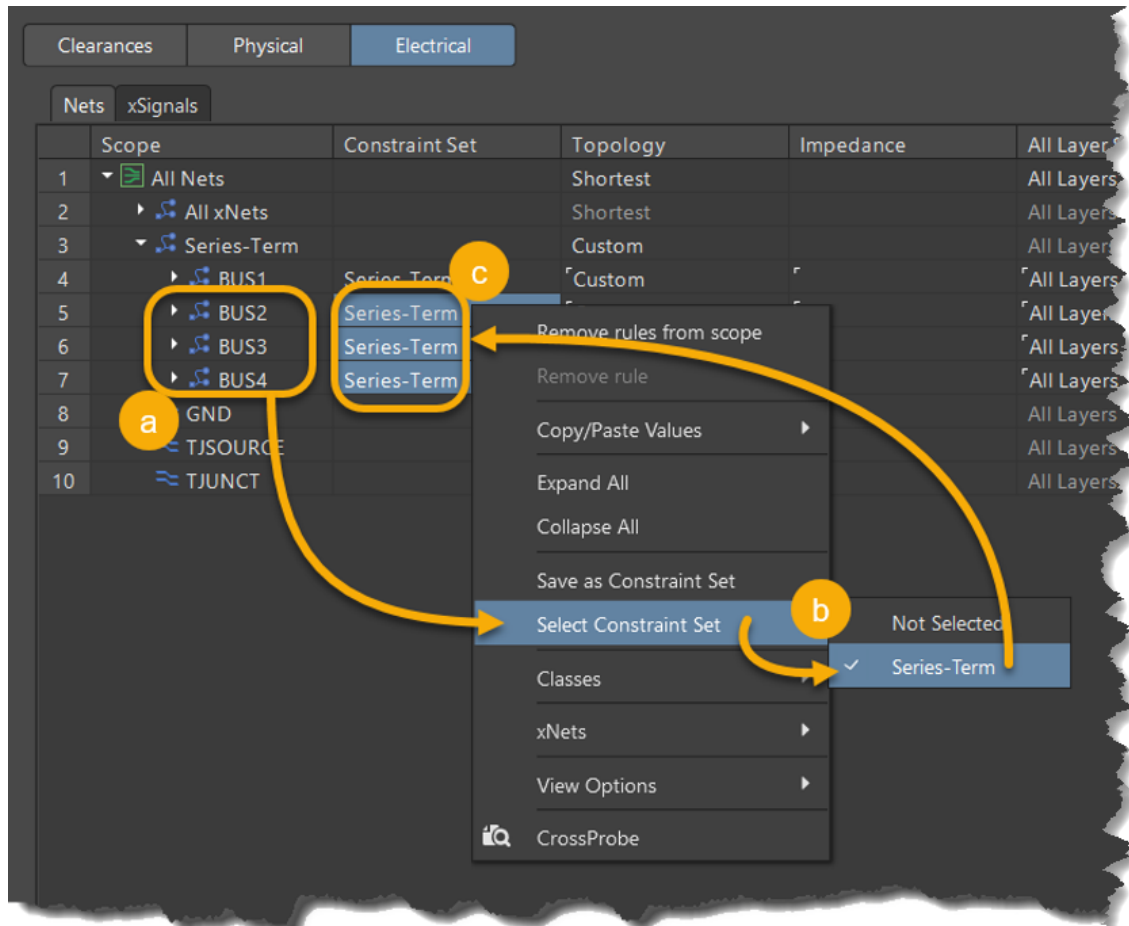


Figure 10. Update BS2...Bus4 with a Constraint Set

24. Select the cell *Topology - Custom* for `BUS2`. The topology is configured the same way as for `BUS1`, and the xSignal is activated. Feel free to check the configuration for `BUS3` and `BUS4`.





6.3.2 xSignal Class

With an *xSignal* Class, you can group one or more xSignals to set specific design rules and settings for that group.

25. Let's create the xSignal Class for the four created xSignals, Figure 11.

- Still in the Electrical **Electrical** view of the Constraint Manager, select the tab *xSignals* at the top.
- Select the four xSignals based on the xNets BUS1...BUS4.
- Right-click and select the command **Classes » Add Class**.
- In the new dialog *Add Class*, add the name *Series-Term*.
- Transfer the four xSignals from the Non-Members to the Members area.
- Close the dialog with **OK**.
- The Constraint Manager is updated with the new class information. Save the modifications with **File » Save** or **CTRL+S**.

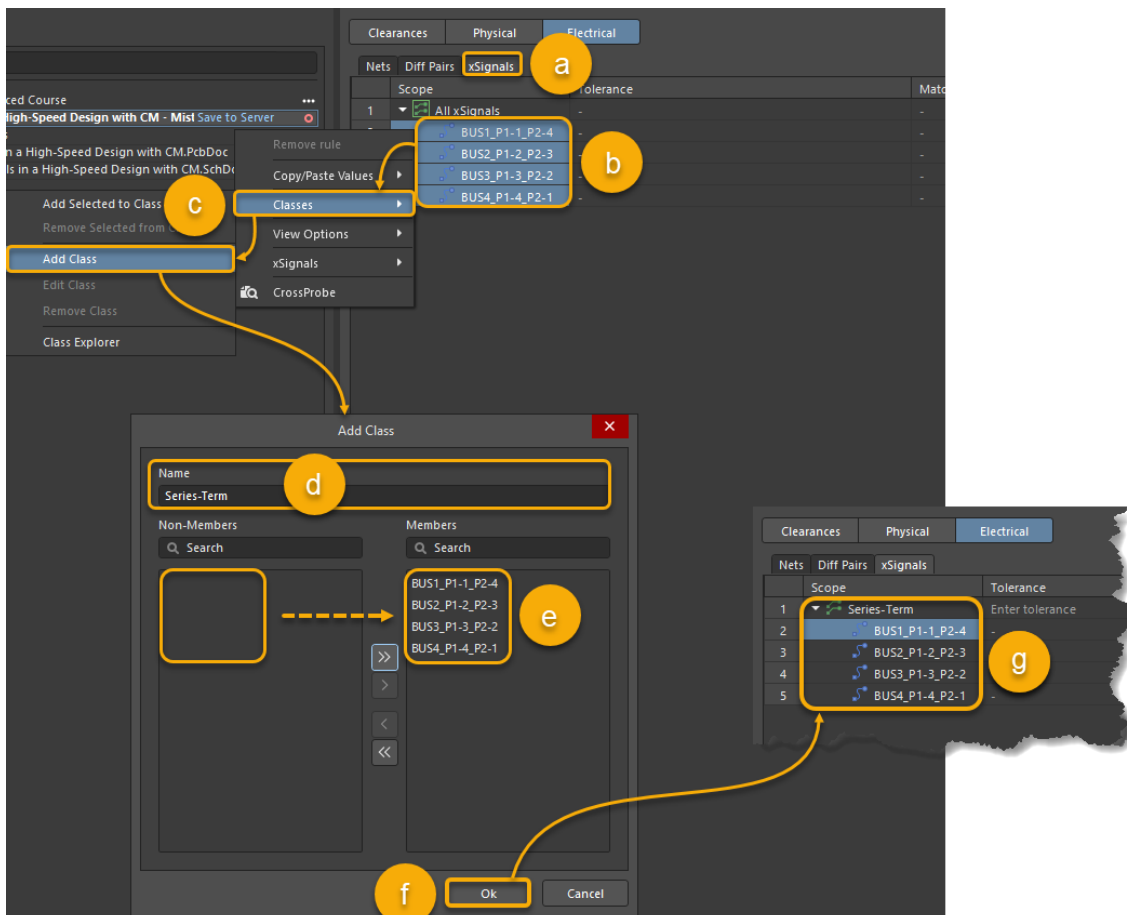


Figure 11. Creating an xSignal class



26. Now you can add a tolerance to the xSignals class. This tolerance will be transferred to the PCB, as Matched Length rule for Length Tuning. Additionally, you can define a net as Matching Target net. For the training example, the Signal BUS4 is used as a reference. Please, follow the steps below, Figure 12:

- Select the cell with the Note `Enter tolerance` and add the Value `10mil`.
- Select the cell with the Note `Not Defined` and select the Net `BUS4...`.
- Save the modifications with **File » Save** or **CTRL+S**.

27. **Close the Constraint Manager.**

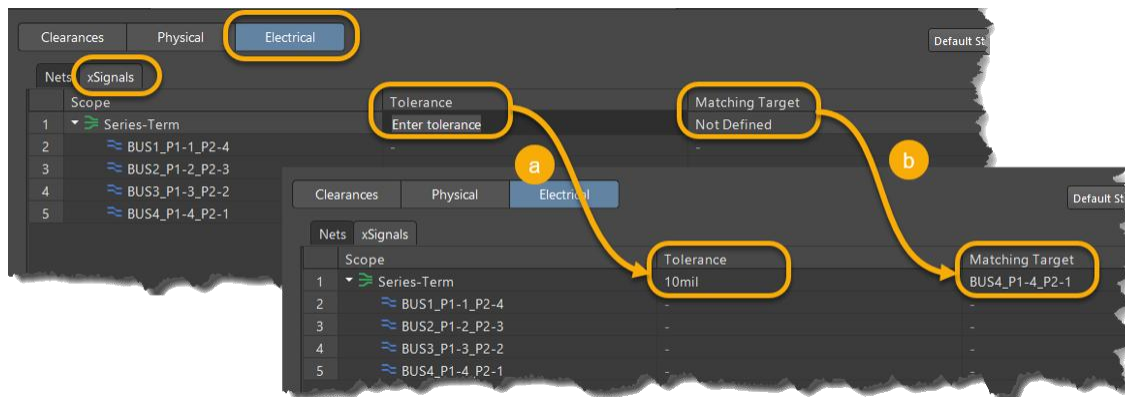


Figure 12. Adding Tolerance and Reference Net for xSignal class



7 Updating PCB

After modifying the constraints in the Schematic, you will now update the PCB with the new information.

28. Make the schematic the active document in the design area. Execute the command **Design » Update PCB Document ...**, this will open the ECO dialog.
 - a) Select **Execute Changes** to import the modification.
 - b) Select **Close** to close the ECO dialog.

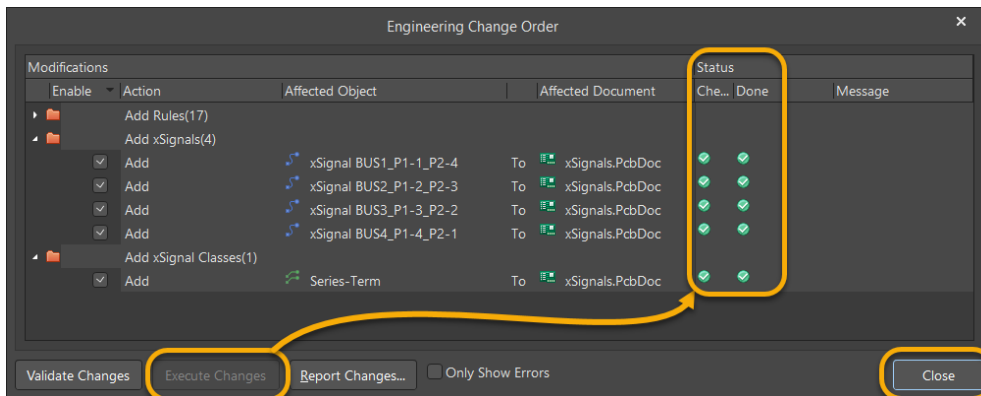


Figure 13. ECO

29. Open the PCB document or change the focus to the PCB, if it's already open.
30. Open the *Constraint Manager* from the PCB with **D»R**, you see four views Clearances, Physical, Electrical, All Rules.
31. Select the All Rules All Rules view, available for the Constraint Manager for the PCB.
 - a) Open the Rule class *High Speed*.
 - b) One Basic Rule is available, the one you've created in the Schematic. Select the rule to see its details.

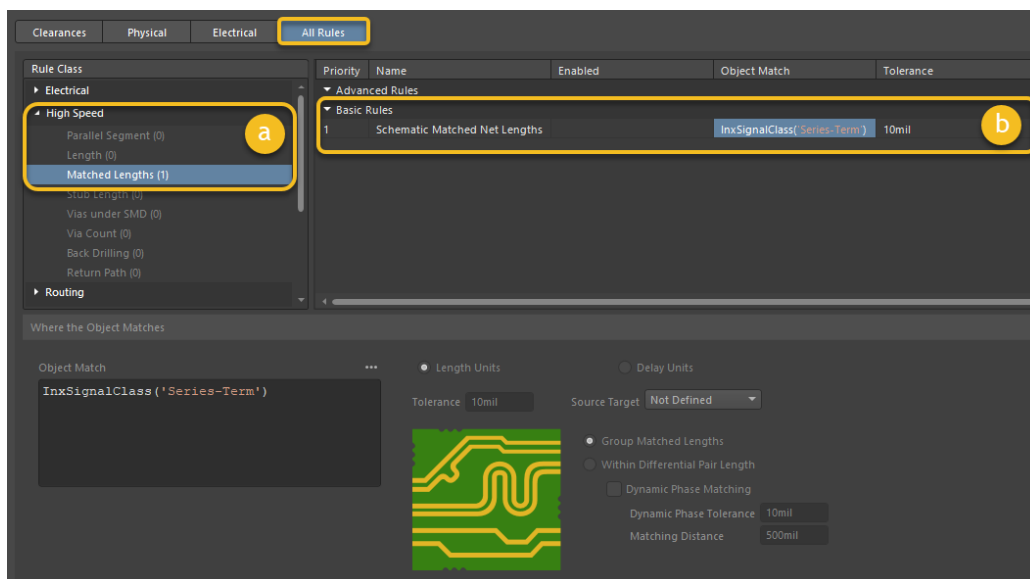


Figure 14. Constraint Manager PCB, rule class High Speed

32. Close the Constraint Manager.





8 Length Tuning for xSignals with Serial Termination

8.1 PCB Panel

33. Open the *PCB* panel or change the focus to the *PCB* panel if it's already open.

34. From the drop-down menu in the *PCB* panel, select **xSignals**, as shown in Figure 15.

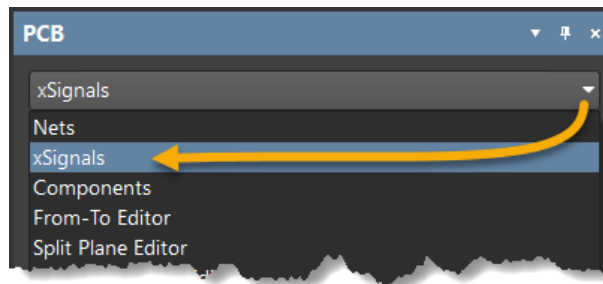


Figure 15. xSignals information from the PCB panel

35. With *xSignals* selected in the drop-down list, note:

- The xSignal class *Series-Term* with four xSignals.
- Two of the xSignals violate the Length rule, as shown by the orange highlights in Figure 16. The two signals are too short compared to the Target net.

Hint: Signals are also shown with a Violation if they are longer than allowed. Check the Margin Column for positive Values. In this situation, you need to check your routing or your Rule Values.

- The Target net information.

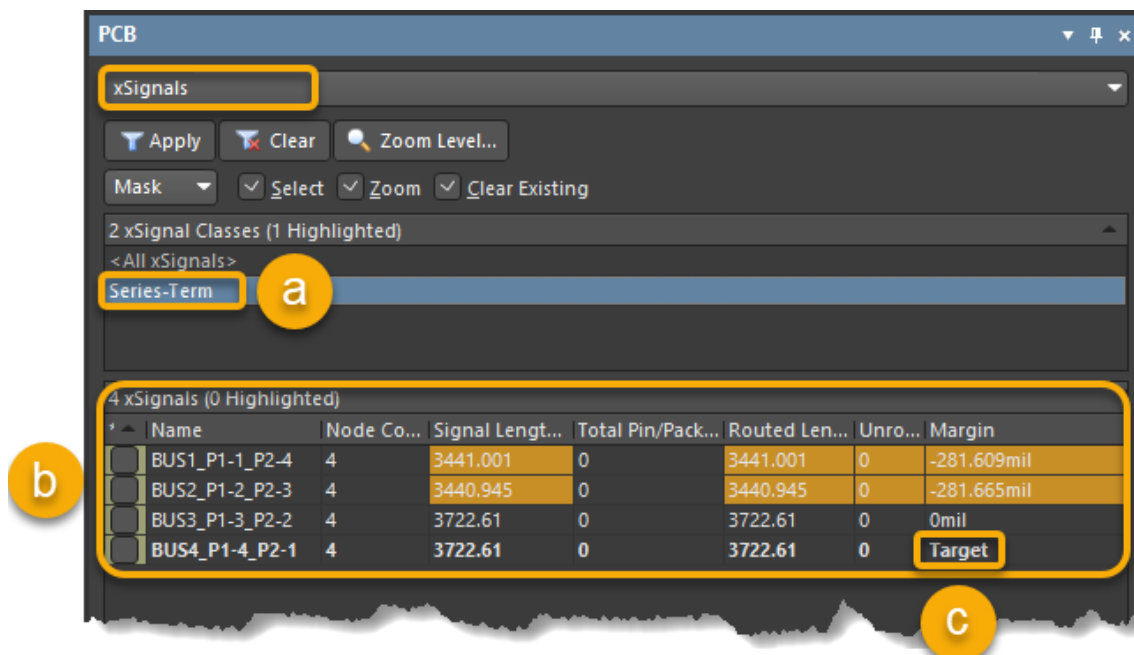



Figure 16. PCB panel with xSignal view





8.2 Initial Length Tuning

Next, you will length tune the two xSignals `Bus1`, `Bus2` so that the routed lengths adhere to the created *Length* rule. This can be done on either side of the resistors.

36. Start the Length Tuning command by going to the **Route** menu and selecting **Interactive Length Tuning, U»R**. This is also available from the ActiveBar .
37. Your cursor will turn to a crosshair. Press the **TAB** key to pause the length tuning and open the *Properties* panel.
38. Use Figure 17 and the instructions below to change the following values for the Length Tuning properties:
 - a) Set the *Pattern* style to **Accordion**.
 - b) Set the *Style* as **Rounded**.
 - c) Set the *Max Amplitude* to 50mil with a *Step* size of 10mil.
 - d) Set the *Space* to 20mil with a *Step* size of 5mil.

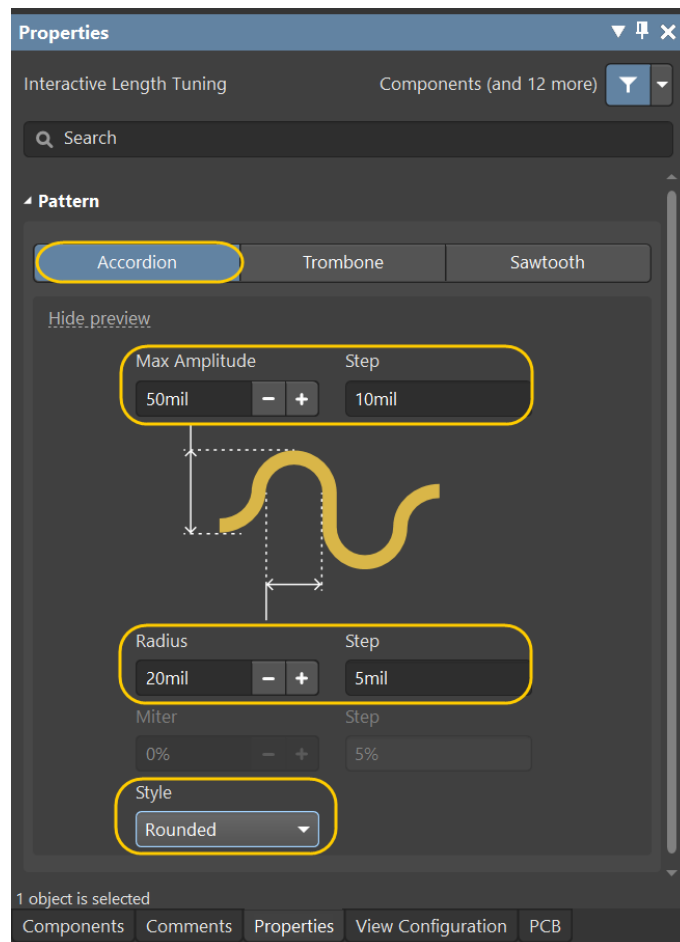



Figure 17. Accordion parameter for first length tuning

39. Select the Pause icon  to continue the length tuning.
40. Click on one of the track segments, either the `BUS1` or `OBUS1` net at the top of the PCB document. If needed, press **G** to change the Grid.
41. Move your cursor along the track to start adding accordions.





42. Hit the **TAB** key to open the Length Tuning properties.
 - a) From the *Properties* panel, in the *Target* section, ensure the *Source* is set to **From Rules**, as shown in Figure 18.
 - b) Enable **Clip to Target** so that the length adheres to the Length Rule.

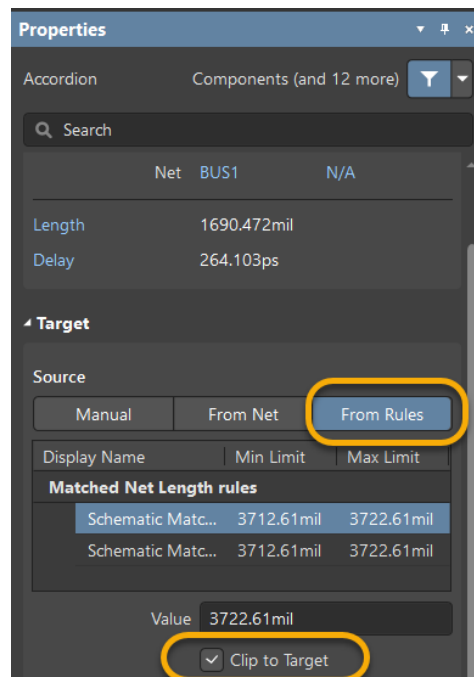



Figure 18. Length Tune from the Length Rule

Hint: The **Clip to Target** option adjusts the length of the xSignal to meet a 10mil Tolerance. When the Length Guide Indicator turns green, the xSignal meets the length requirement.

43. Select the Pause icon  to continue the length tuning.
44. To ensure that you're adhering to the Length rule, you can use the *Length Guide Indicator* to help you achieve the proper length. This guide can be enabled and disabled using the **SHIFT+G** shortcut keys while tuning, as shown in Figure 19.

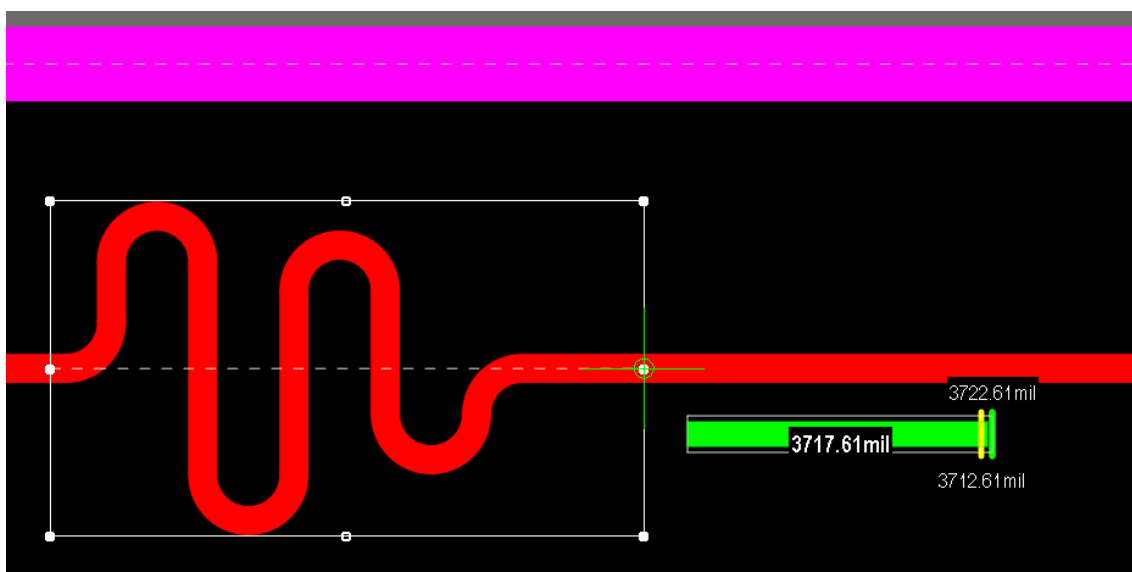


Figure 19. Using the Length Guide Indicator





45. Left-click to complete the length tuning for the first xSignal BUS1.
46. Repeat the length tuning for the second xSignals, similar to what is shown in Figure 20. Add one part at the BUS signal and the other part at the OBUS signal.

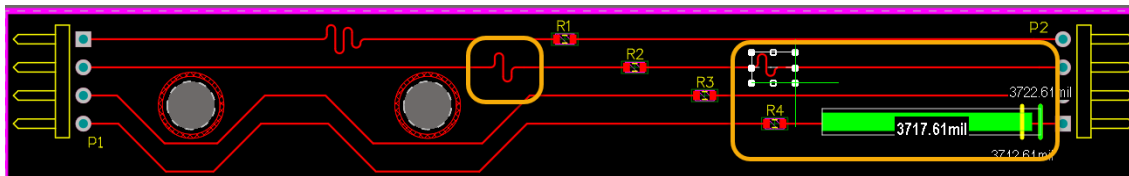


Figure 20. Series-Term xSignals with length tuning

47. Feel free to check the PCB panel for the xSignals Length and Margin.
48. Save the modifications **File » Save** or **CTRL+S**.





9 xSignals from a Net with T-Junctions

Next, you will create xSignals for a signal with T-Junction connections, set a rule for it, and adjust the length as needed. As before you will use the Constraint Manager from the Schematic to add this information.

9.1 Creating xSignals from a Net with T-Junctions

49. Change the focus back to the Schematic document.
50. Open the Constraint Manager with **D»G** or by selection **Design » Constraint Manager...**, you see the Constraint Manager with three view (**Clearances**, **Physical**, **Electrical**).
51. Select the Electrical **Electrical** view.
52. From the tab **Nets**, select the Signal **TJUNCT**.
 - a) Select the topology **Custom**.
 - b) Define the topology as shown in Figure 21 by **Right-click » Add Node**
 - i) Source: R10-2
 - ii) Destination: P3-1, P4-1, P5-1, P6-1. The node R15-2 won't be used in the exercise.
 - c) Activate the checkboxes for all **Proposed xSignals**.
53. Save the modifications **File » Save** or **Ctrl+S**.

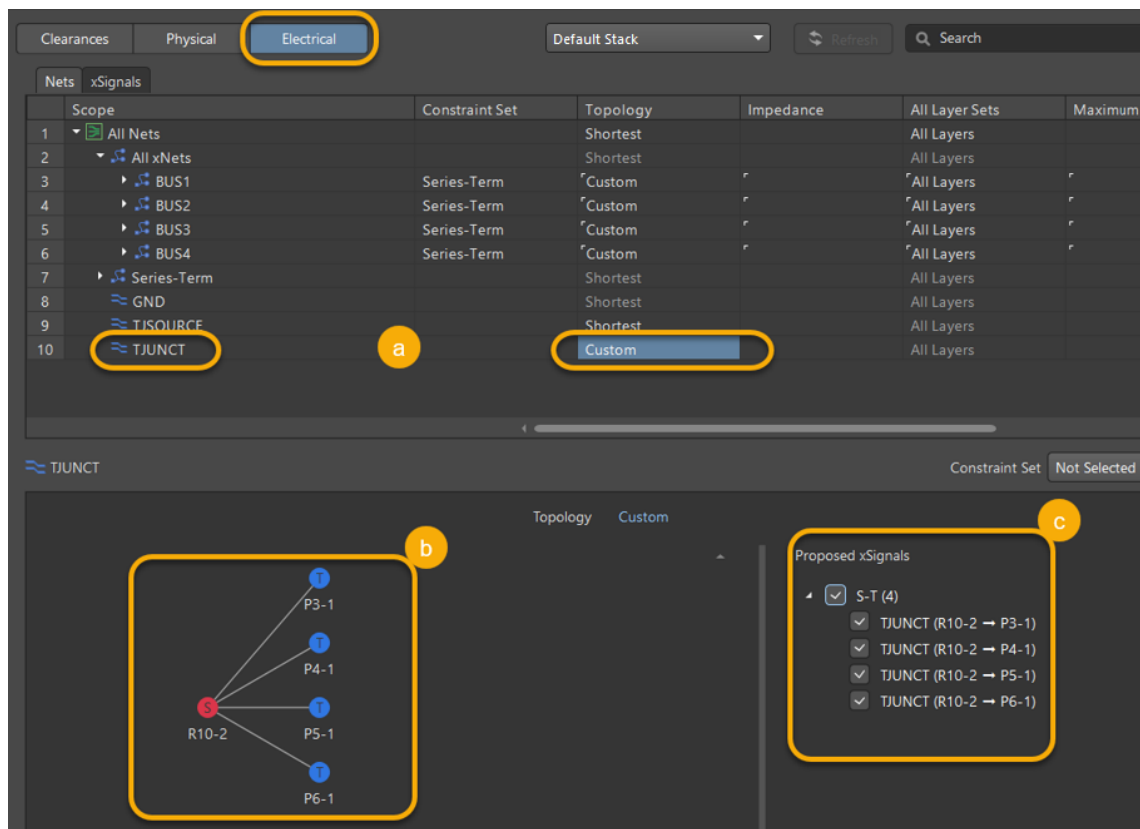


Figure 21. Topology for TJUNCT





Now that you have defined the Topology with the Proposed Signals, you can check the xSignal list.

54. Select the Tab xSignals and configure the new xSignals, as seen in Figure 22.

- Select the four TJUNCT xSignals and create a xSignal Class with **Right Click » Classes » Add Selected to Class » New Class**
- Add the Class Name T-Junction.
- Add the Tolerance of 10 Mil.
- Save the modifications **File » Save** or **CTRL+S**.
- You can close the Constraint Manager.

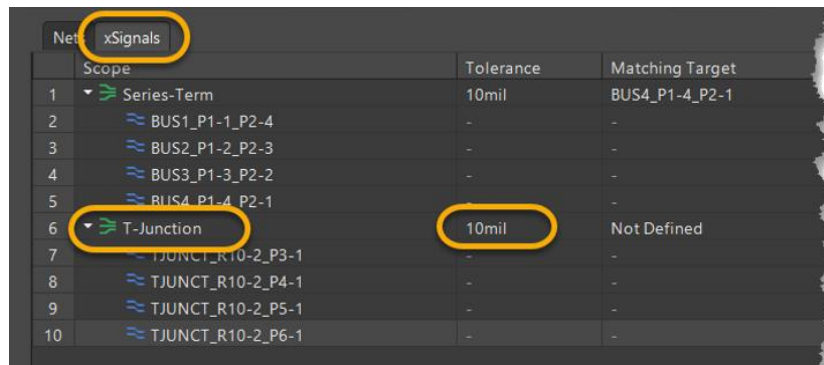


Figure 22. Constraint Manager with xSignal Class T-Junction and Tolerance

55. Now that you have modified the constraints, you will update the PCB:

- Make the Schematic the active document.
- Execute the command **Design » Update PCB....** This will open the *ECO* dialog.
- Select **Execute Changes** to push the modifications to the PCB.
- Select **Close** to close the *ECO* dialog.

56. You can explore the xSignals classes by going to the **Design** menu and selecting **Classes**. Use Figure 23 as a reference.

57. Press **OK** to close the *Object Class Explorer*.

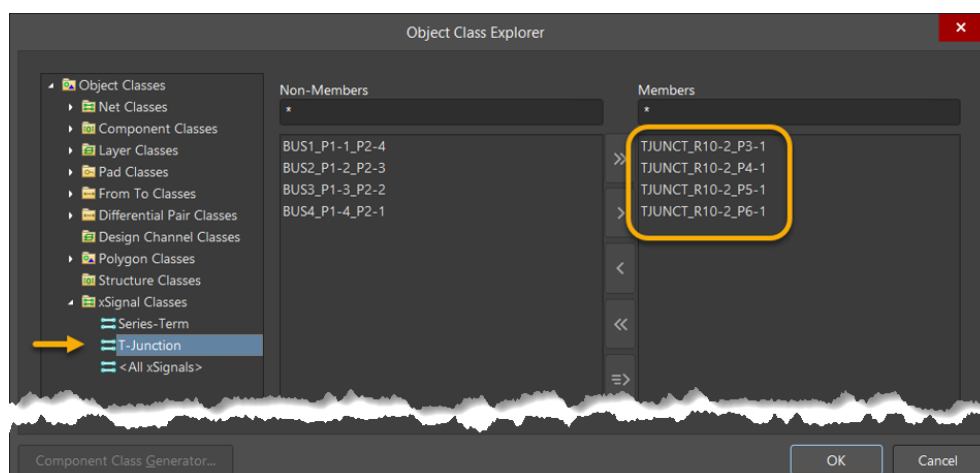


Figure 23. New xSignal class for the T-Junction net



58. Change your focus back to the PCB document and the *PCB* panel.

- Near the top of the panel, change the visibility drop-down from **Normal** to **Mask**.
- Ensure that the **Select** option is also enabled.
- With the xSignals selected from the drop-down menu, select the T-Junction class we created earlier.
- In the following xSignals pane, simply select each individual xSignal from TJUNCT__R10-2_P3_1 to TJUNCT__R10-2_P6_1 to see the signal path, as shown in Figure 24.
- Note that even if you don't define a target net, Altium Designer selects the longest existing routed path as target net for length tuning.
- Select the button **Clear** or, back in the PCB, **Shift+C** to remove masking.

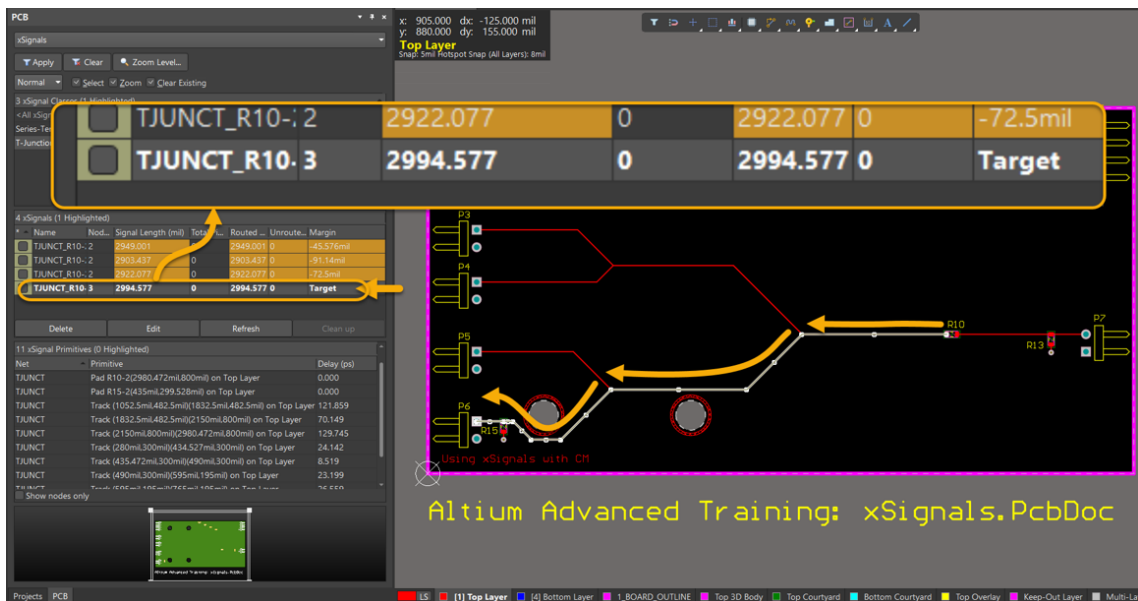


Figure 24. One signal path from the T-Junction xSignal class

9.2 Rules for xSignals from Net with T-Junction

Next, you can add an additional Length rule for the T-Junction example in the constraint manager, opened from the PCB.

59. Open the *Constraint Manager* from the **Design** menu or with **D»R.** , you see the Constraint Manager with four view (**Clearances** , **Physical** , **Electrical** , **All Rules**).
 - a) Select the **All Rules** **All Rules** view.
 - b) The *High Speed - Matched Length* section now has two rules, one for the class *Series-Term* and the new rule for T-Junction.
60. Next, you will create a new *Length* rule for the T-Junction xSignals.
 - a) Ensure **All Rules** tab is selected.
 - b) In the *High-Speed* section, open the *Length* rule category. At the moment no Length rule exist.
 - c) Right-click anywhere in the rule area and select the command **Add Custom Rule**. A new **Advanced Rule** is created.
 - d) Select the new rule and change the name to *Length_TJunction*. Now, you can configure its settings, as seen at the bottom of the **Constraint Manager**.
 - e) Configure the rule with the following parameters, using Figure 25 as a reference.
 - i) In the Object *match* section, type in `InxSignalClass('T-Junction')`.
 - ii) Change the Values in the *Constraint* area to:
 - Enable the **Length Units** option: *Minimum*: 3900mil, *Maximum*: 4100mil

Hint: You can use the tool suggestions after typing the first few characters, or view the list of existing classes after adding `InxSignalClass`. Alternatively, click the ellipsis button to use the Builder or helper.

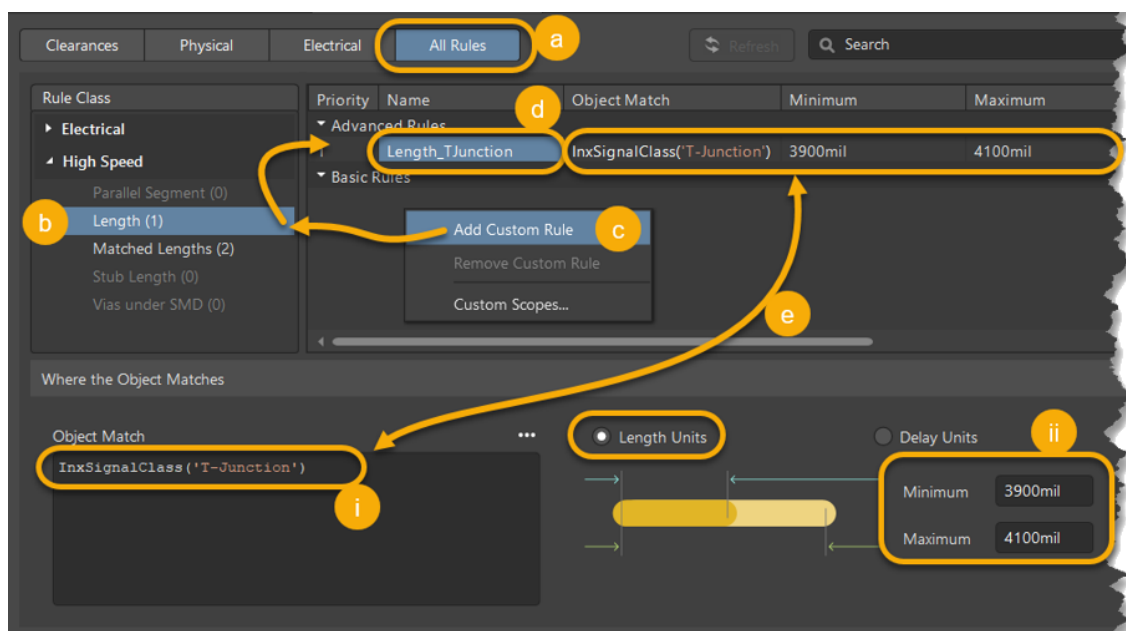



Figure 25. *Length_TJunction* rule

61. Save the modifications.



62. Change the focus to the PCB and the *PCB* panel with the **xSignals** still selected in the drop-down list.
 - a) Select the xSignal class *T-Junction*.
 - b) Note how all four xSignals now break the length rule, as they're highlighted in orange.
 - c) Select the **Clear** button at the top of the *PCB* panel to clear any selections.

9.3 Length Tuning with Accordion

63. You will length tune all four *TJUNCT* xSignals, as shown in Figure 26.
64. From either the **Route** menu or from the *ActiveBar*, start the **Interactive Length Tuning (U»R)** command.
65. With the command active, hit the **TAB** key and ensure that the followings values are set for the Length Tune accordions that will be added:
 - a) *Style*: Rounded
 - b) *Max Amplitude*: 120mil with *Step*: 10mil
 - c) *Space*: 30mil with *Step*: 10mil
66. Select the Pause icon  to continue the length tuning. Complete length tuning on all 4 xSignals so that they now adhere to the length rule, as shown in Figure 26.
67. Right-click to exit the command once completed.

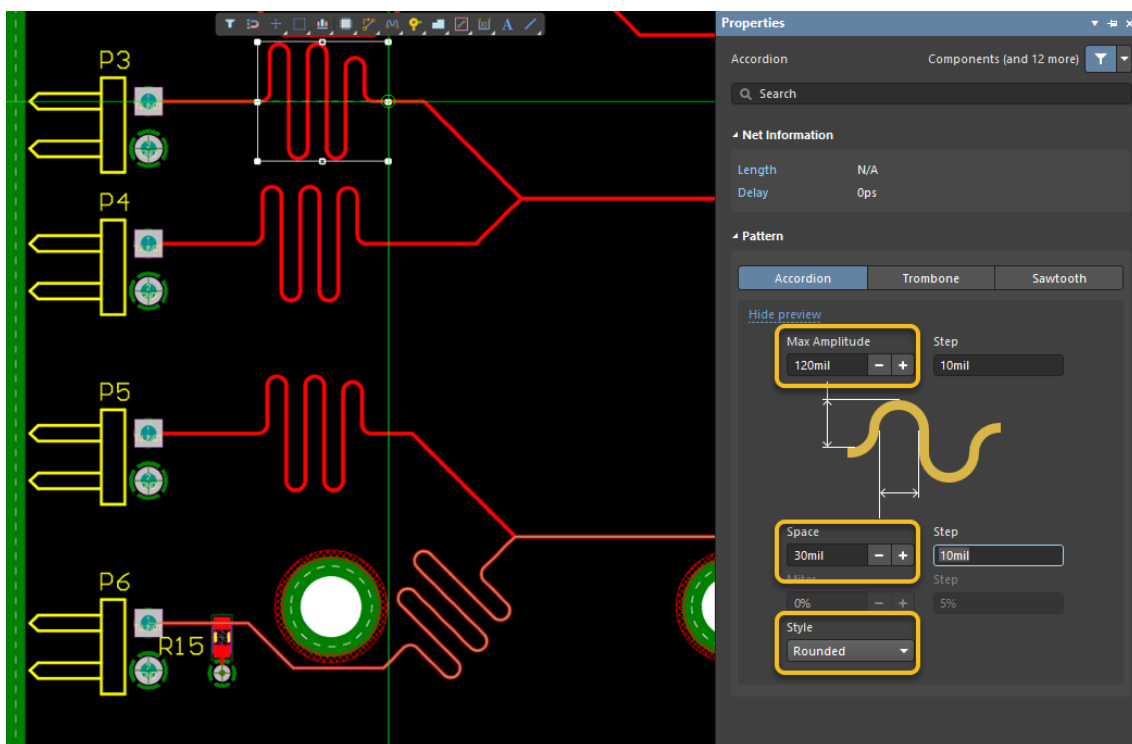


Figure 26. Tuned T-Junction xSignal Class





10 Unions and Length Tuning

When Length Tuning is added, all tuned routes will be listed in the PCB panel, making it easy to review them.

68. In the *PCB* panel, select **Unions** from the drop-down menu, as shown in Figure 27.

69. In the *Union Type* pane, select **Interactive Length Tunings**.

70. In the next pane, you will find all of the length tuned objects that you've placed during this exercise.

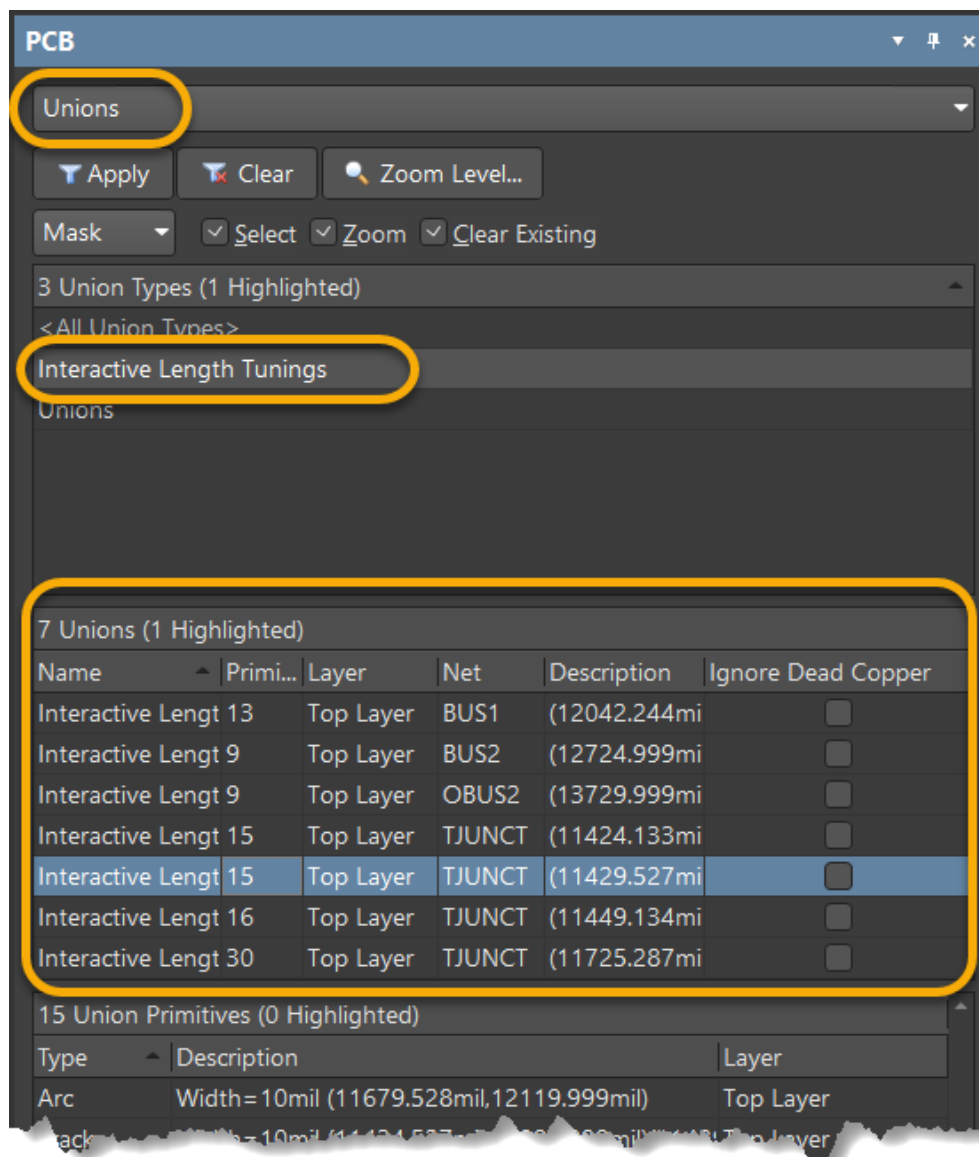


Figure 27. Union PCB Panel

Hint: If you placed a Length Tuning object without a pattern, unnecessary tuning, or incorrect values like radius or amplitude, you can easily find this "empty" Length Tuning object in the PCB panel by selecting **Unions - Interactive Length Tunings**.






11 Length Tuning with Trombone

Next, you can length tune the T-junctions signal paths again. Instead of using the *Accordion* style, you will use the *Trombone* pattern.

11.1 Deleting Accordion Length Tuning

71. Select the *Accordion* length tuning objects for the net TJUNCT.
72. Press **DEL** on your keyboard to delete the length tuning objects.

11.2 Length tuning with Trombone

73. Open the *PCB* panel and select the xSignal class *T-Junction*. Note how all xSignals break the signal Length/Routed Length rule again.
74. Select **Clear** in the *PCB* panel.
75. Start the **Interactive Length Tuning** command from the **Route** menu, *ActiveBar*  or press **U»R**.
 - a) Your cursor will turn into a crosshair. **Don't** start tuning yet.
 - b) Press the **TAB** key to open the *Properties* panel. Use Figure 28 as a reference to change the following settings:
 - i) *Pattern*: Trombone
 - ii) *Style*: **Rounded**
 - iii) *Radius*: 50mil
 - iv) *Step*: 10mil

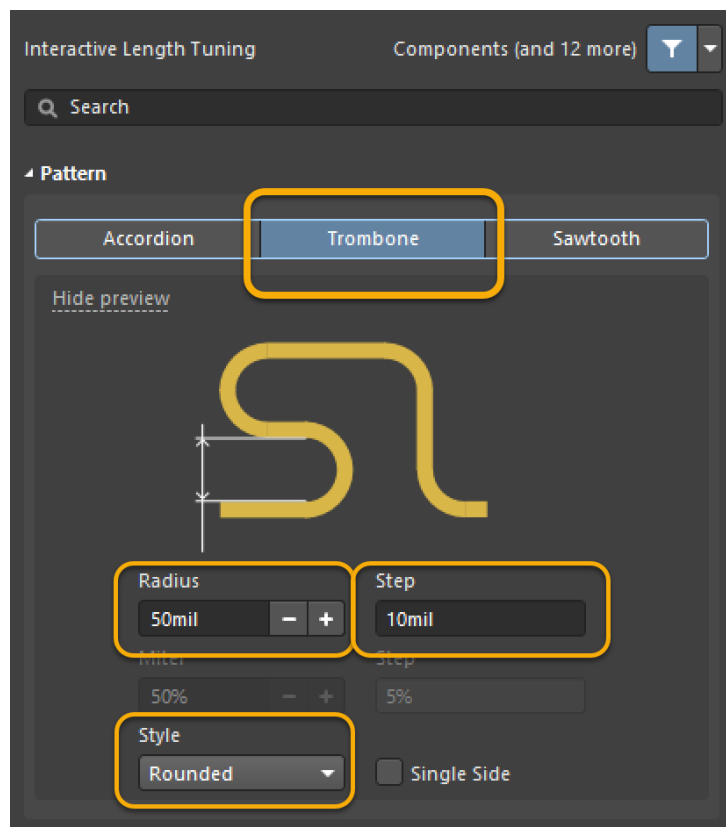



Figure 28. Trombone pattern





76. Select the Pause icon  to continue the length tuning.
77. Begin length tuning each route for components P3, P4, and P5, as shown in Figure 29. When the Length Guide Indicator turns green, the routes follow the rule.

Hint: The Trombone doesn't need to be perfect. After it has been placed, you can optimize it by selecting and moving as you wish.

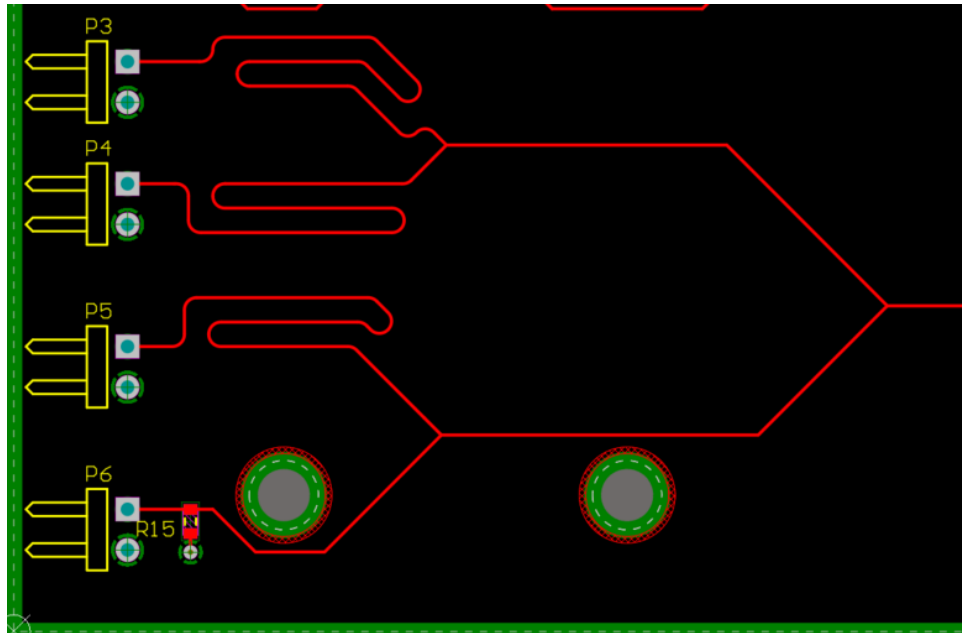


Figure 29. First Trombone length tuning





11.3 Modifying Trombone

The last signal path for P6 is too close to a board cutout. This means, you won't have enough room to length tune the path in this area. You need a *Trombone* pattern with more loops.

78. Use the Length Tuning command again to tune the signal path for P6, to the right of the board cutout, as shown in Figure 30. The Signal Path will be still too short, but we will change that with the next steps.

79. **Left-click** to place the *Trombone* and **right-click** to exit the command.

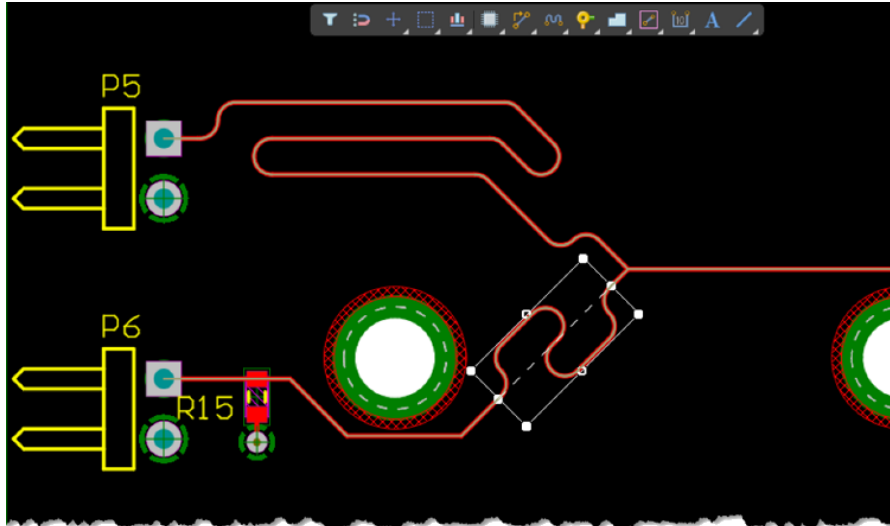


Figure 30. First Trombone for Signal Path P6

80. Select the Trombone you just placed. You will notice that there's a bounding box around it. It's called the Sleeve.

81. Once selected, you will see the vertex points, as shown in Figure 31.

82. Click and drag the lower vertex point away from the board cutout. By changing the size of the bounding box, Altium will either add, remove, or change the length of the existing loops to update the signal length.

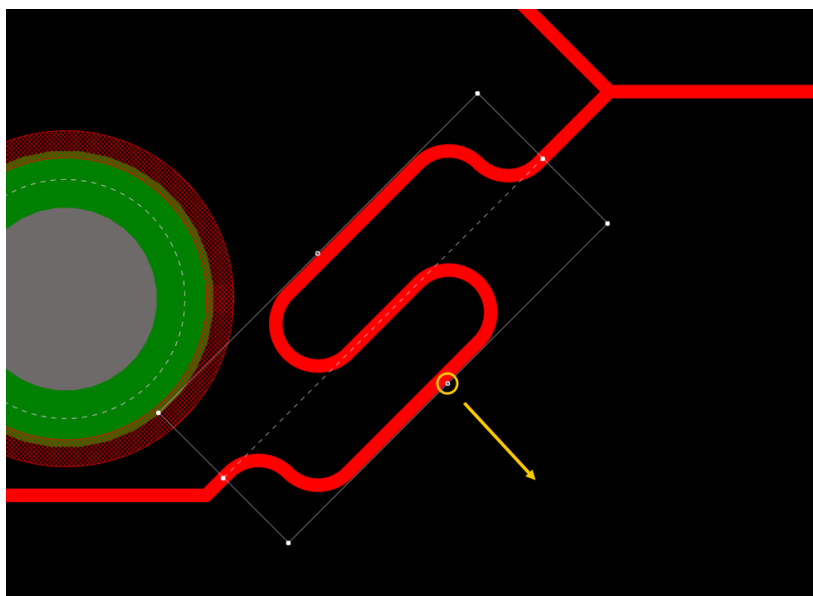


Figure 31. Adjusting the bounding box of the Length Tune





83. Keep dragging the vertex until enough loops are added to meet the length rule, as shown by the Length Guide Indicator in Figure 32.

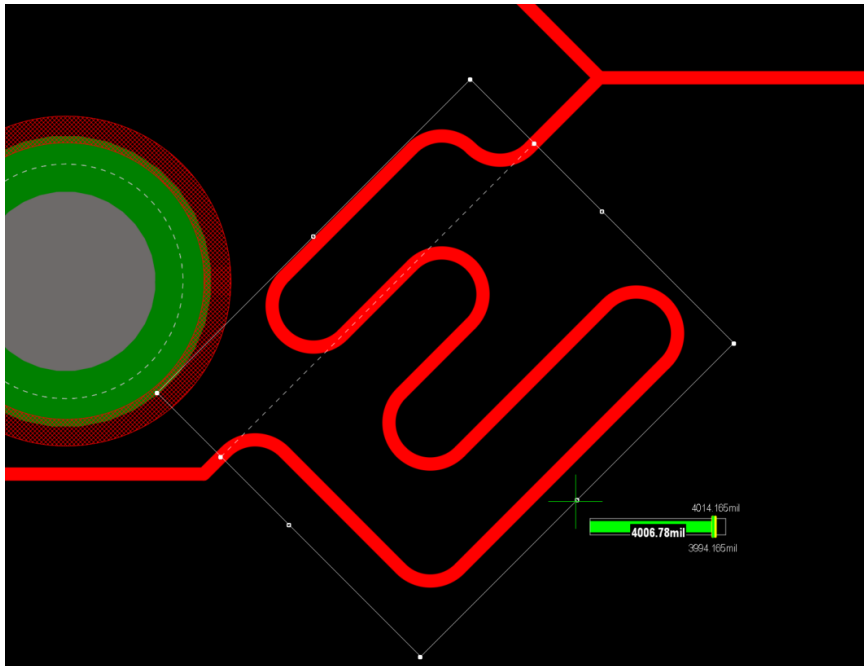
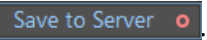


Figure 32. Adjusting the Length Tune to adhere to the Length rule

84. Open the *PCB* panel to ensure that neither of the xSignal classes have any errors, and that all signal routes adhere to your rules.
85. Save all documents using **File » Save All**.
86. Save the modifications to the server:
- In the *Projects* panel, next to the Project name you find the command **Save to Server**

 - Select **Save to Server**.
 - In the dialog *Save [Project Name]*:
 - Add the comment Using xSignals in a High-Speed Design with Constraint Manager - [Add Your Name] - Finished.
 - Select **OK**.
87. When ready, close the project and any open documents, **Window » Close All**.





Congratulations on completing the Module!

Using xSignals in a High-Speed Design with
Constraint Manager

from

**Altium Designer Advanced Training
with Altium 365**

Thank you for choosing **Altium Designer**

