

Altium Designer

Advanced Course

Module:
Using xSignals in a High-Speed Design

Software, documentation and related materials:

Copyright © 2022 Altium LLC

All rights reserved. You are permitted to use this document provided that (1) the use of such is for personal use only and will not be copied or posted on any network computer or broadcast in any media, and (2) no modifications of the document are made. Unauthorized duplication, in the whole or part, of this document by any means, mechanical or electronic, including translation into another language, except for brief excerpts in published reviews, is prohibited without the express written permission of Altium LLC. Unauthorized duplication of this work may also be prohibited by local statute. Violators may be subject to both criminal and civil penalties.

ACTIVEBOM®, ActiveRoute®, Altium 365™, Altium Concord Pro™, Altium Designer®, Altium Vault®, Altium NEXUS™, Autotrax®, Camtastic®, Ciiva™, CIIVA SMARTPARTS®, CircuitMaker®, CircuitStudio®, Codemaker™, Common Parts Library™, Draftsman®, DXP™, Easytrax®, EE Concierge™, xSignals®, NanoBoard®, NATIVE 3D™, OCTOMYZE®, Octopart®, 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 subsidiaries. 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	3
1.1 Purpose	3
1.2 Shortcuts	3
1.3 Preparation	3
1.4 Overview	3
1.5 Using xSignals with a Serial Termination	4
1.5.1 Creating the xSignals	4
1.5.2 Working with xSignal Classes	7
1.5.3 Creating Design Rules for the Serial Termination xSignals	9
1.5.4 Length Tuning for xSignals with serial termination	11
1.5.4.1 Initial Length Tuning	11
1.6 xSignals from a net with T-Junctions	13
1.6.1 Create xSignals from a net with T-Junctions	13
1.6.2 Rules for xSignals from net with T-Junction.....	16
1.7 Creating xSignals using the xSignals Wizard	18
1.7.1 Preparation.....	18
1.7.2 Running the xSignal Wizard	18
1.7.3 Creating Rules after using the xSignal Wizard	19
1.7.4 Length Tuning.....	19
1.7.4.1 Length tuning with Trombone	19
1.7.4.2 Modify Trombone.....	20
1.8 Unions and Length Tuning	22

Using xSignals in a High-Speed Design

1.1 Purpose

The designer's job is to translate their design requirements, such as the maximum route length allowed to meet the timing budget, into a set of design rules, such as a Length rule to ensure that the timing is met and a Matched Length rule to detect potential timing mismatches.

The designer sees the signals in terms of their function – “Here is an address signal and it must be routed from this connector to each memory device. To achieve that, I'll route using a fly-by topology with a termination resistor at the end. I might also require a series terminator at the source”. Even though address A0 passes through a termination resistor, to the designer, that signal is still A0 on the other side of that resistor.

But the PCB editor sees each signal simply as a set of connected pins (commonly referred to as a net) — Net A0 goes from this connector pin to this memory component pin, then to this memory component pin, and so on. As soon as a series termination resistor is added, that address line becomes two different nets. This makes it difficult for the designer to specify key design requirements, such as Length and Matched Length design rules.

An xSignal is essentially a designer-defined signal path between 2 nodes - they can be 2 nodes within a net, or they can be 2 nodes in associated nets separated by a component. The xSignal can be used to scope relevant design rules such as Length and Matched Length, which will then be obeyed during design tasks, such as interactive length tuning.

1.2 Shortcuts



Shortcuts when working with xSignals and Length Tuning:

Open the xSignal Manager: **D » X » W**

Create xSignals between components: **D » X » C**

Interactive Length tuning **U » R**

1.3 Preparation

1. **Close all existing projects and documents.**
2. Open the `Using xSignals in a High-Speed Design.PrjPcb` project found in its respective folder of the Advanced Training.

1.4 Overview

3. Open the `xSignals.SchDoc` schematic document. You'll see two separate circuits. A group of bus signals with a serial resistor, as well as multiple components with T-Junctions connections.
4. Open the `xSignals.PcbDoc` PCB document. You'll notice the component placement and routing is already complemented as shown in Figure 1 below.

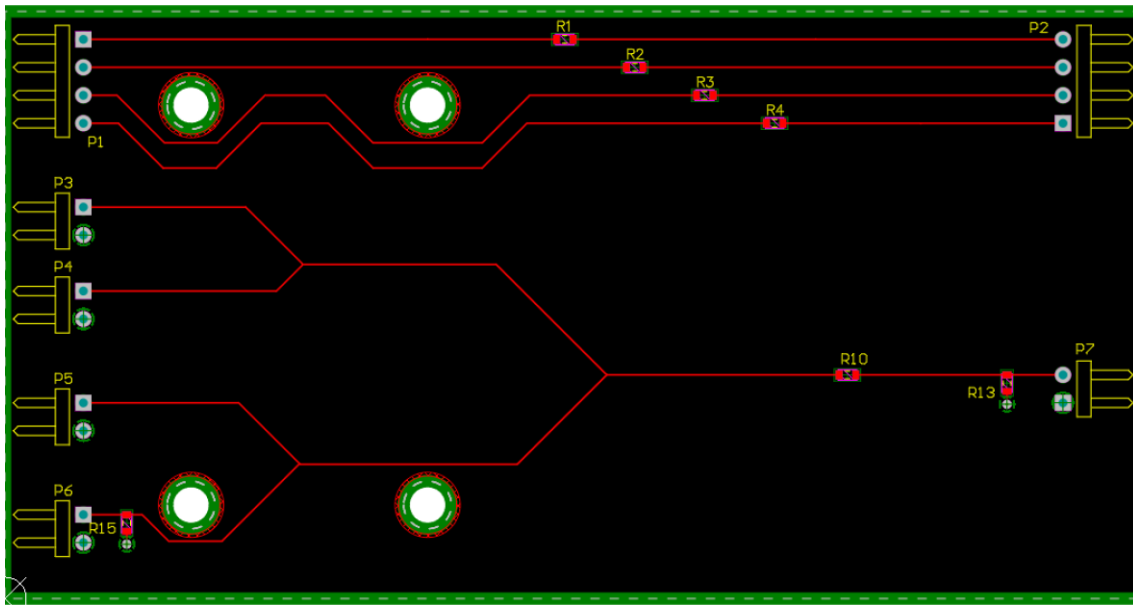


Figure 1. PCB without xSignals and Length Tuning

5. In summary, here is what you will do for the Series Termination and T-Junctions sections:
 - a) Create the xSignals and the xSignal Class.
 - b) Create PCB Design Rules for *Length* and *Matched Net Lengths*.
 - c) Length tune the xSignals.

1.5 Using xSignals with a Serial Termination

1.5.1 Creating the xSignals

As a first step, we will create new xSignals between components P1 and P2.

6. Open the *PCB* panel from the **Panels** button.
 - a) From the drop-down menu in the *PCB* panel, select **xSignals** as shown in Figure 2.

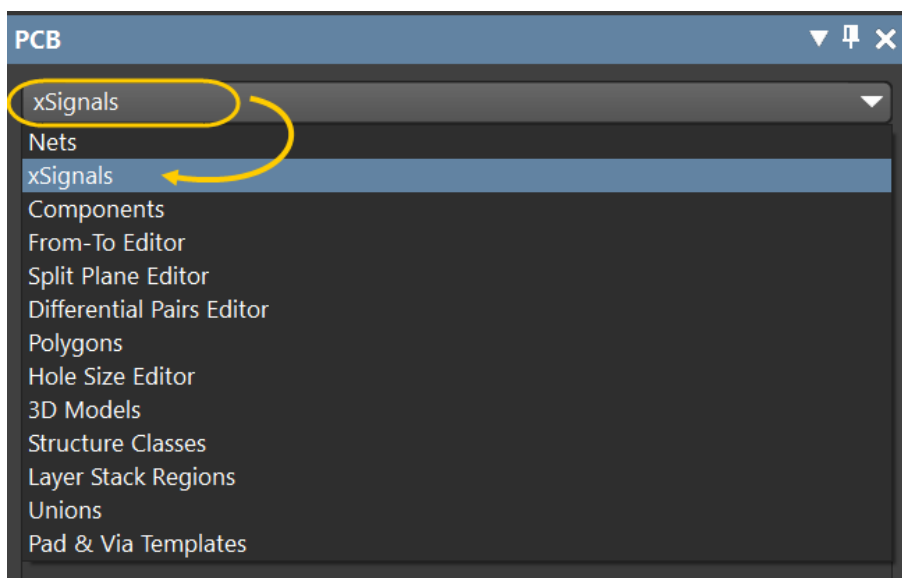


Figure 2. xSignals information from the PCB panel

- b) The *xSignal Class* pane will list all of the xSignals classes in our design. One *xSignals Class* is created by default as shown in Figure 3.
- c) The next pane for *xSignals* is empty as there's currently no *xSignals* defined in the PCB.

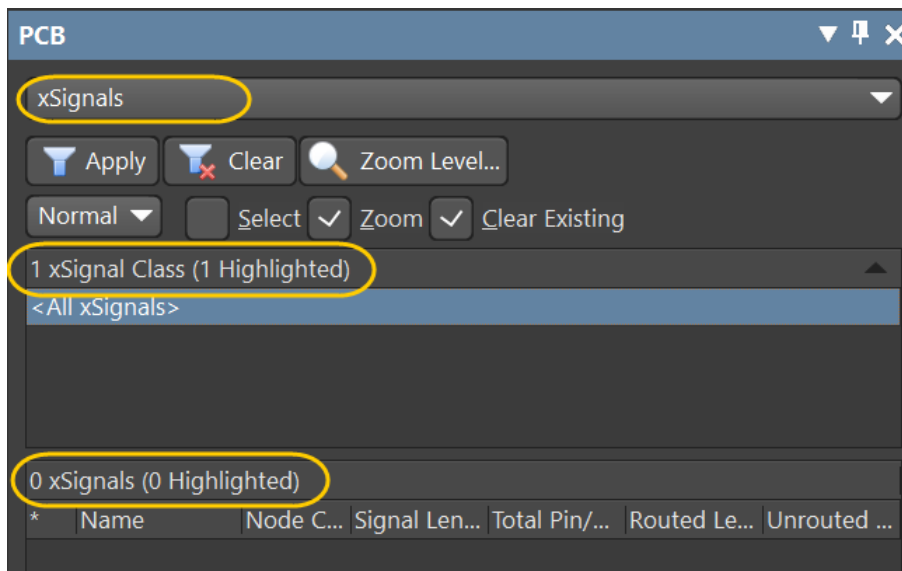


Figure 3. xSignals pane information from the PCB panel

7. While still in the *PCB* Panel:
 - a) Change the menu drop-down from **xSignals** to **Components**.
 - b) Ensure that both the **Select** and **Zoom** options are enabled as shown in Figure 4.

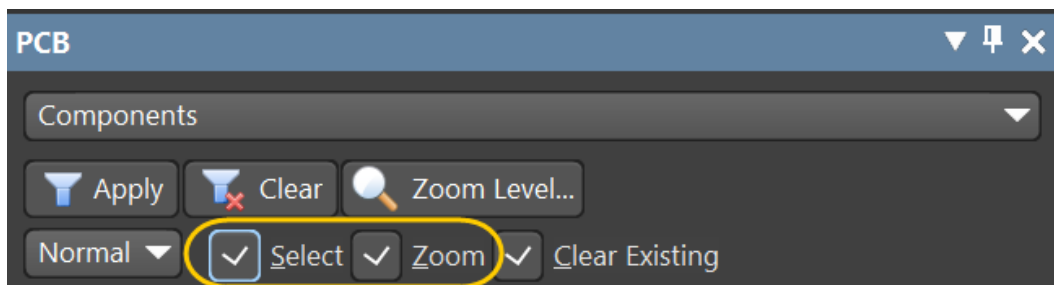


Figure 4. Select & Zoom options in the PCB panel

- c) From the *Component Class* pane, select *<All Components>*. The *Components* pane will now list all of the components in the PCB.
- d) Select components R1 through R4 using **Shift+Select** to see the 4 components we will use for our first set of xSignals.

8. Right-click on either of these components in the PCB.
9. From the right-click menu, select **xSignals » Create xSignals From Connected Nets** as shown in Figure 5. This method is one of the most common methods to create multiple xSignals at once.

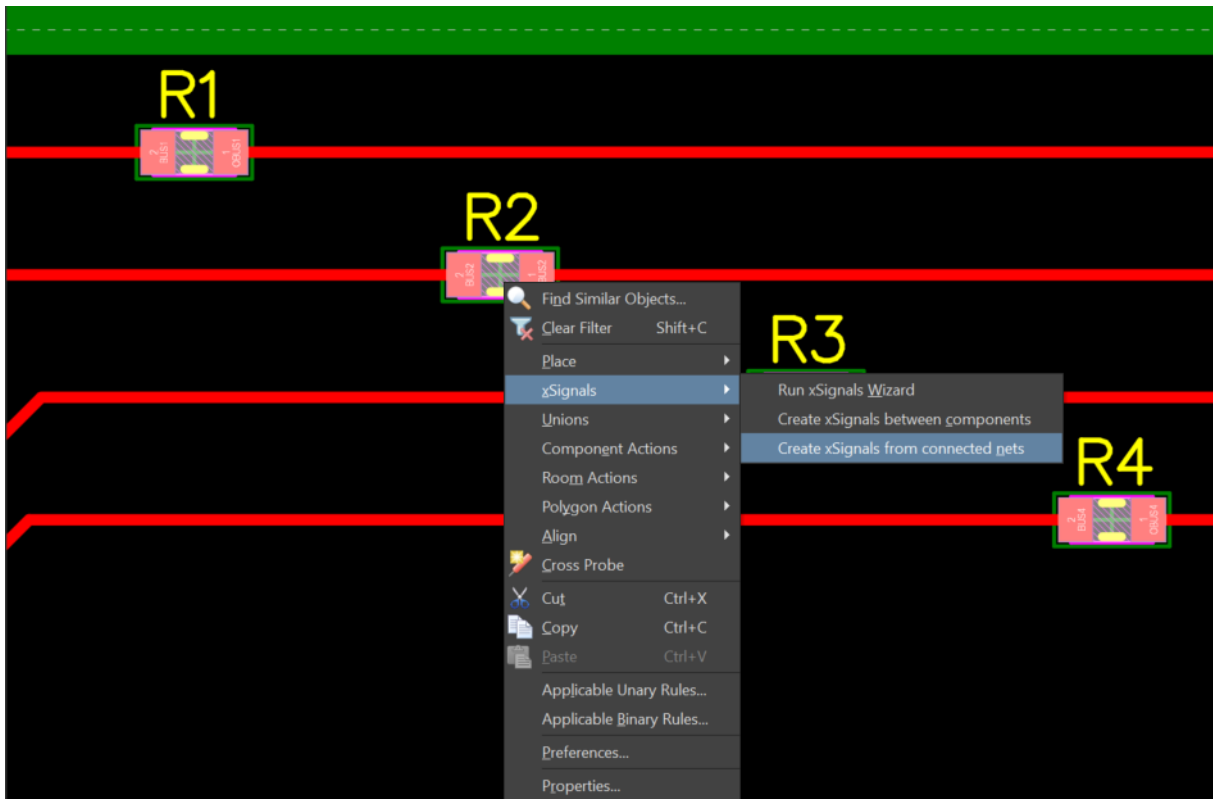


Figure 5. Create xSignals between components dialog

10. In the *Create xSignals From Connected Nets* dialog that appears, use Figure 6 below to help you with the following instructions:
 - a) Ensure that the 4 resistors are selected from the *Source Component* pane.
 - b) Ensure that all of the *Source Component Nets* are selected.
 - c) Click **Analyze**. The software will identify all possible xSignals between the chosen components, including the selected nets, and it will list them in the *xSignals* region of this interface. All potential xSignals will be generated. In this case, it's going to create an xSignal for each of the connected nets.
 - d) Click **OK** to close this interface and the 4 detected xSignals will be created.



Clicking on the drop-down arrow next to the Analyze button allows you to specify if the connections are direct, multinet, or through one or two series components. In this example we used the default option for 1 series component.

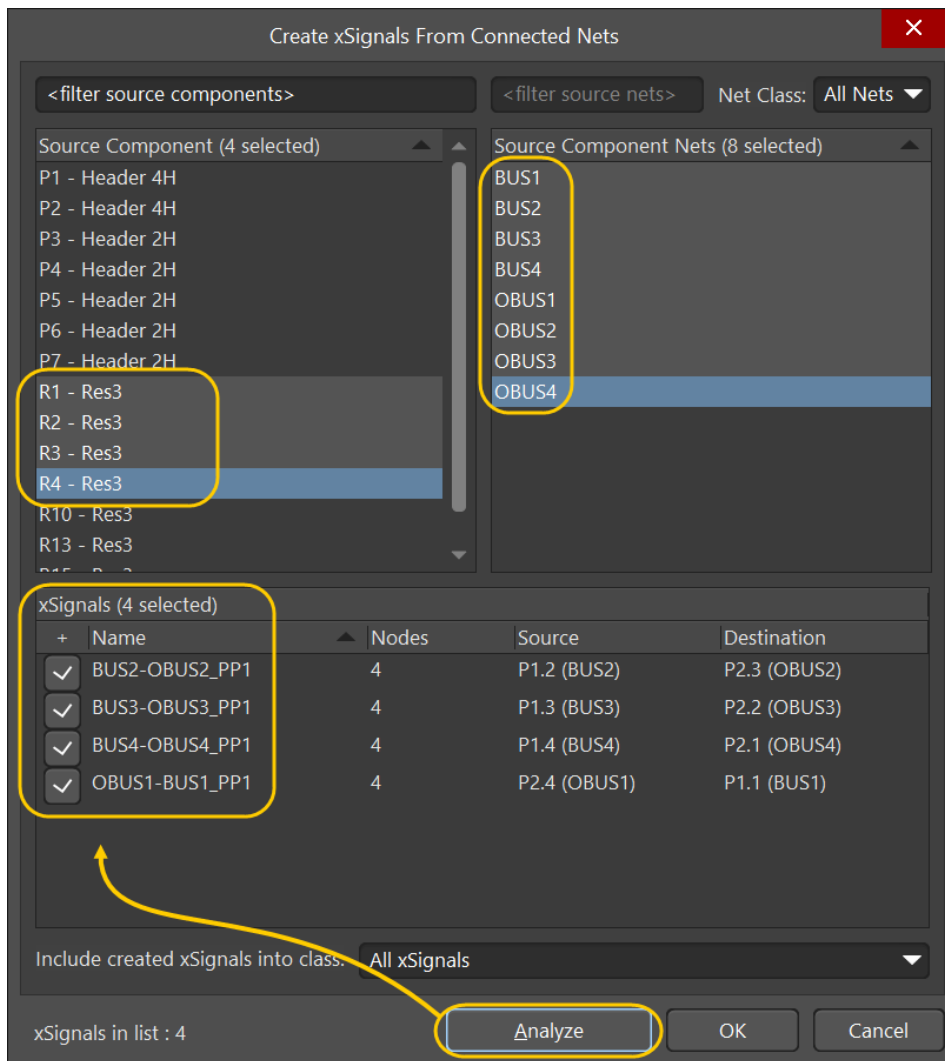


Figure 6. Dialog to create xSignals between components

1.5.2 Working with xSignal Classes

An *xSignal Class* would allow us to assign one or multiple xSignals into a specific group. This would also allow us to create Design Rules and other specifications for the xSignals in that group only.

11. Change the focus back to the *PCB* panel.

- Select **xSignals** from the drop-down list at the top of the panel.
- The *xSignal Class* pane will list all of the xSignals classes, which is currently only the <All xSignals> class.
- With the <All xSignals> class selected, we can see the 4 xSignals generated from the previous steps in the *xSignals* pane.

12. Right-click anywhere in the *xSignal Class* pane and select **Add Class** as shown in Figure 7.

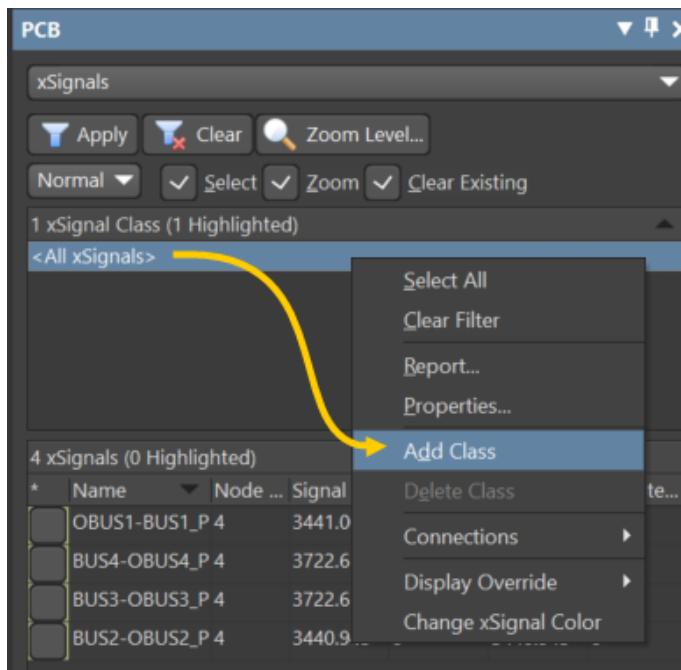


Figure 7. A new xSignal Class

13. In the *Edit xSignal Class* dialog that appears:

- Change the name to *Series-Term*.
- From the *Non-Members* section, select all 4 *BUS* nets.
- Then, click on the **>>** button to add these 4 xSignals to the *Members* group. Your dialog should now look like Figure 8.
- Click on the **OK** button to close the dialog.

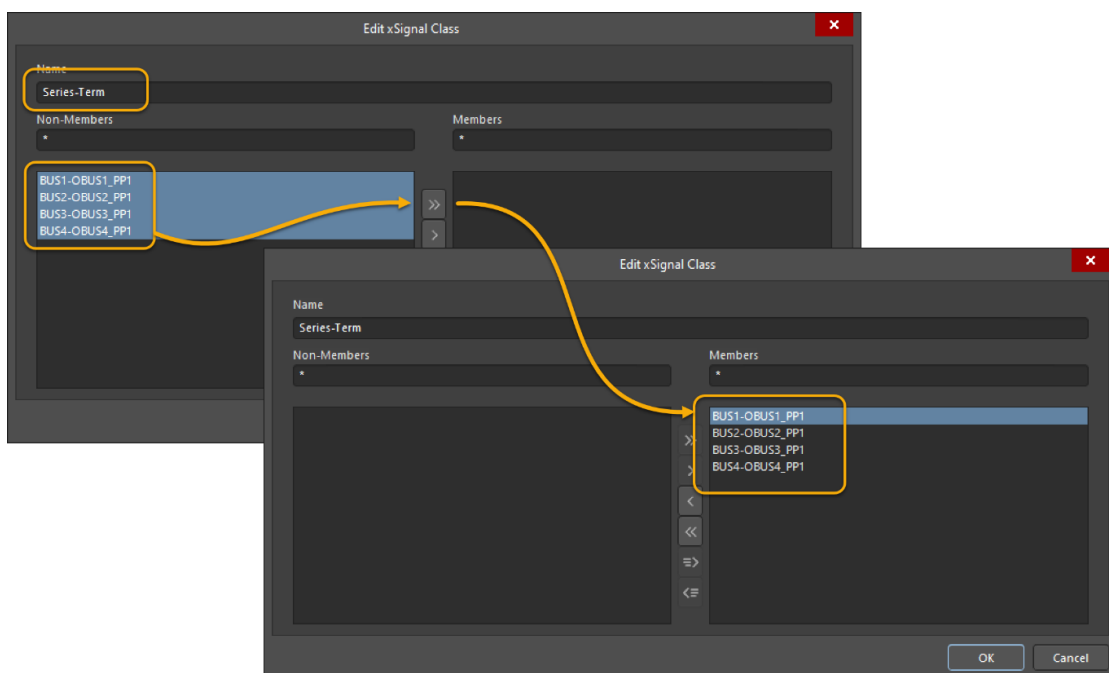


Figure 8. Adding xSignal members to a xSignal class

14. In the *PCB* panel, you will now see the *xSignal Class* *Series-Term*, with the 4 nets that we assigned as members. This is shown in Figure 9.

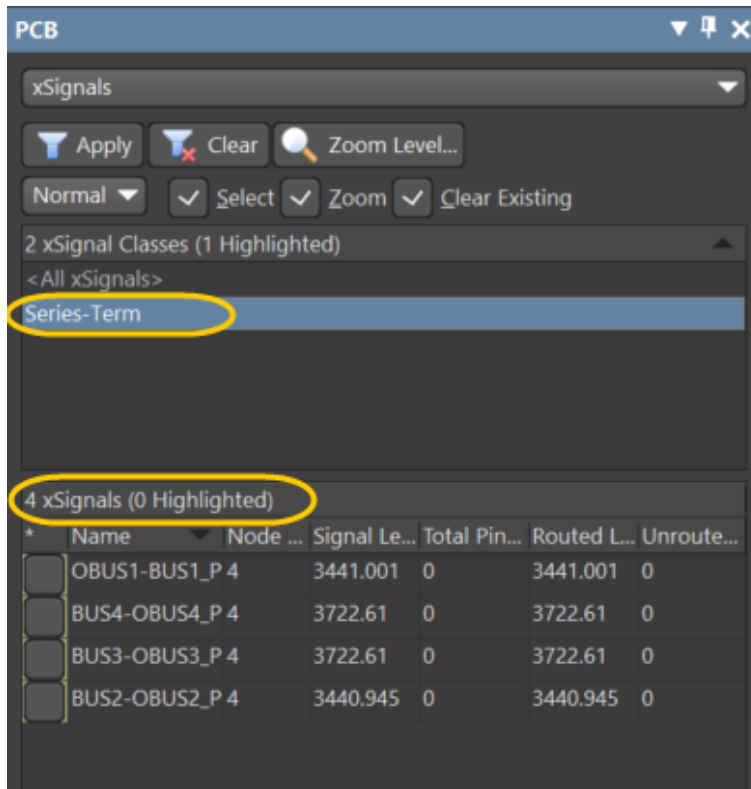


Figure 9. New *xSignals* Class with *BUS* nets

1.5.3 Creating Design Rules for the Serial Termination *xSignals*

15. From the Design menu, select **Rules....** The shortcut for this is **D » R.**
16. Using Figure 10 as reference, we will create a new design rule and change its settings:
- From the left-side of this interface, expand the *High Speed* rule category.
 - Right-click on the *Length* rule category and select **New Rule...**
 - Double-click on the new *Length* rule so that we can configure its settings.
 - Change the Name to `Length_Series-Term`.
 - In the *Query* section, under *Where the Object matches*, select **xSignal Class** from the drop-down menu.
 - From the 2nd drop-down menu, select the **Series-Term** *xSignal Class*.
 - Change the values in the *Constraints* area to:
 - Minimum: 3645mil
 - Maximum: 3655mil
 - Close the *PCB Rules and Constraint Editor* by clicking **OK**.

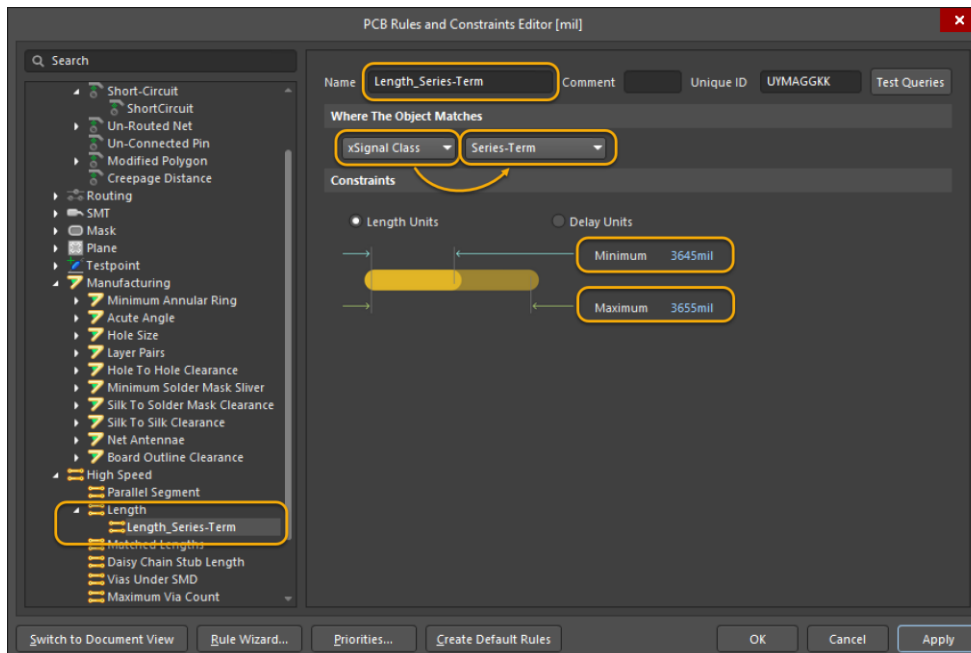


Figure 10. Length rule for the Series-Term xSignal class members

17. Change the focus back to the *PCB* panel.
18. With xSignals still selected in the drop-down list, note how all four xSignals violate the *Length* rule as shown by the orange and red highlights in Figure 11. Two Signals are too short and two signals are too long. Since we will be length tuning the routes in a later step, we will ensure they're all below the minimum length rule for now.

2 xSignal Classes (1 Highlighted)

<All xSignals>

Series-Term

4 xSignals (0 Highlighted)

* Name	Node ...	Signal Le...	Total Pin...	Routed L...	Unroute...
OBUS1-BUS1_P 4		3441.001	0	3441.001	0
BUS4-OBUS4_P 4		3722.61	0	3722.61	0
BUS3-OBUS3_P 4		3722.61	0	3722.61	0
BUS2-OBUS2_P 4		3440.945	0	3440.945	0

Delete Edit Refresh Clean up


Figure 11. xSignal Length rule violations

19. From the **Design** menu, re-open the Length_Series-Term rule.
 - a) Change the *Minimum* length to: 3845mil
 - b) Change the *Maximum* length to: 3855mil
 - c) Click **OK** to close the *PCB Rules and Constraint Editor*.
20. Back in the *PCB* panel, we can see that all signal paths are now shorter than the *Minimum* length defined in the rule.
21. Hit the **Clear** button at the top of the *PCB* panel to clear any existing selection.

1.5.4 Length Tuning for xSignals with serial termination

1.5.4.1 Initial Length Tuning

Next, we will length tune the four xSignals so that the routed lengths adhere to the *Length* rule we just created. This could be done on either side of the resistors.

22. Start the Length Tuning command by going to the **Route** menu and select **Interactive Length Tuning**. This is also available from the ActiveBar .
23. Your cursor will turn to a crosshair. Press the **TAB** key to pause the length tuning and open the *Properties* panel.
24. Using Figure 12 as a reference, as well as the instructions below, 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 *Radius* to 20mil with a *Step* size of 5mil

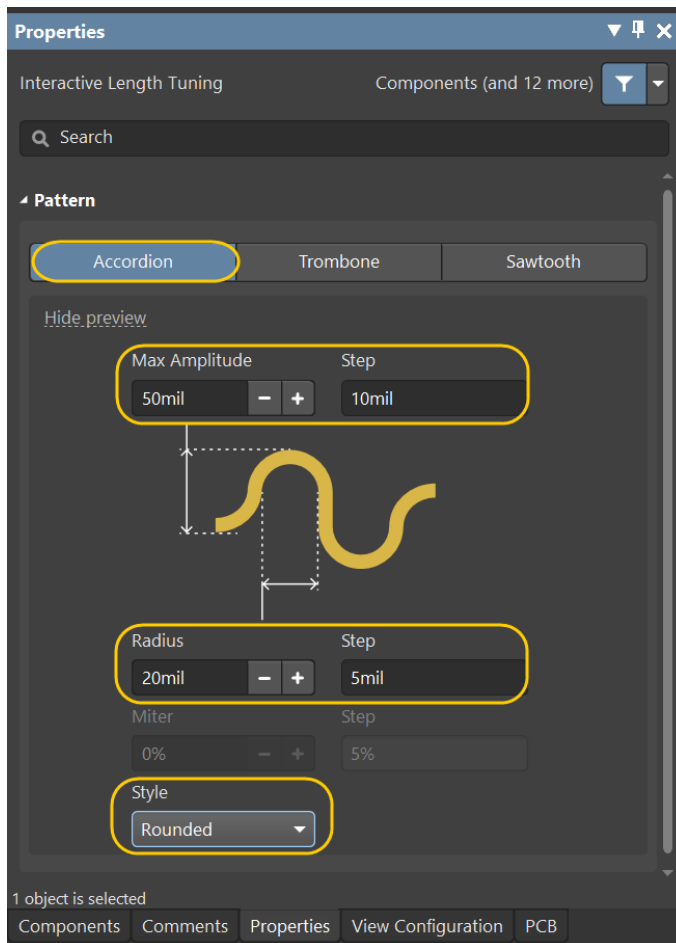



Figure 12. Accordion parameter for first length tuning

25. Click on the Pause icon  to continue the length tuning.
26. Click on one of the track segments, either the **BUS1** or **OBUS1** net at the top of the PCB document.
27. Move your cursor along the track to start adding accordions.

28. Hit the **TAB** key to open the Length Tuning properties.
29. From the *Properties* panel, in the *Target* section, ensure the *Source* is set to **From Rules** as shown in Figure 13.
30. **Clip to Target** must also be enabled so that the length adheres to the Length Rule.

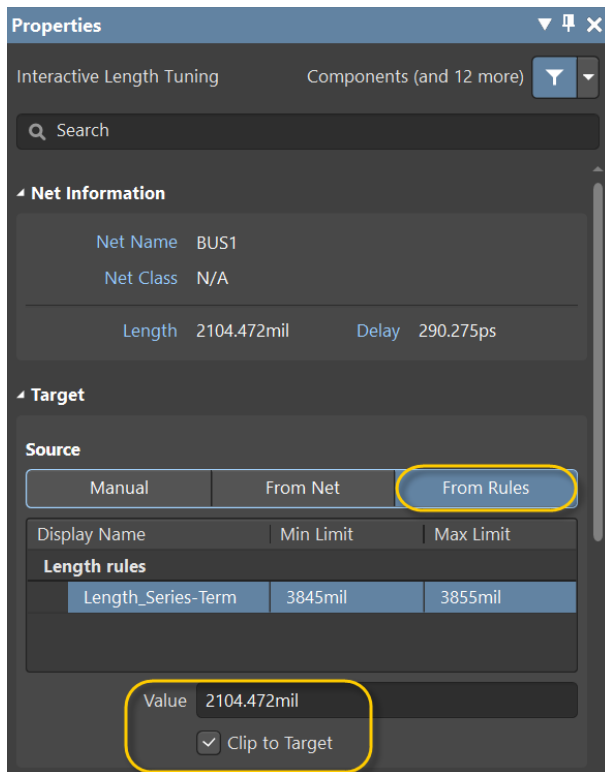



Figure 13. Length Tune from the Length Rule



The **Clip to Target** option will automatically provide the length necessary to tune the xSignal to a value between 3845mils and 3855mil. This is the value that's shown in the Length Guide Indicator. When the indicator turns green, this means the xSignal meets the Length rule, but the value shown is for that single net only.

31. Click on the Pause icon  to continue the length tuning.
32. 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/disabled using the **SHIFT+G** shortcut keys while tuning as shown in Figure 14.

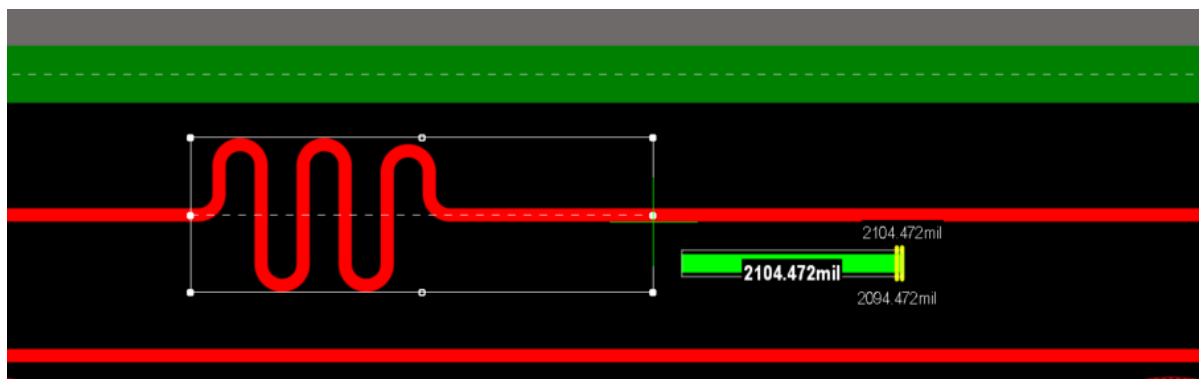


Figure 14. Using the Length Guide Indicator

33. Left-click to complete the length tuning for the first xSignal.
34. Repeat the length tuning for the three remaining xSignals similar to what is shown in Figure 15
 - a) BUS2-OBUS2_P4
 - b) BUS3-OBUS3_P4
 - c) BUS4-OBUS4_P4 on the right side of component R6.

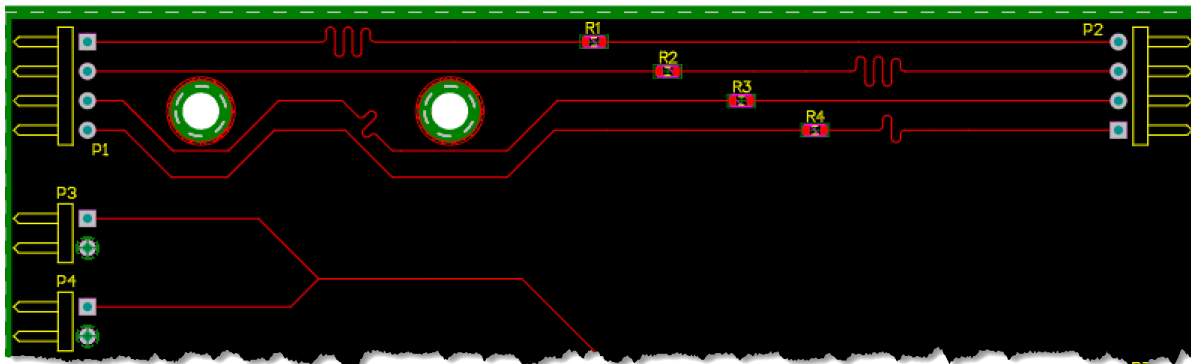


Figure 15. Series-Term xSignals with length tuning

1.6 xSignals from a net with T-Junctions

Next we will create xSignals for a signal that contains T-Junction connections. We will also define a rule for this xSignal and perform the appropriate length tuning.

1.6.1 Create xSignals from a net with T-Junctions

35. In the PCB document, select R10 and P3.
36. Right-click on one of the selected components. From the right-click menu, select **xSignals » Create xSignals between components**.
37. Using Figure 16 as a reference, in the *Create xSignals Between Components* dialog, we will change the following settings:
 - a) Select R10 as the *Source Component*.
 - b) Select P3, P4, P5 and P6 as the *Destination Components*.
 - c) Leave the *Source Component Nets* at the default (2 selected).
 - d) Click on **Analyze**.
 - e) In the *Create xSignals between Components*, note the four xSignals (TJUNCT_PP1 to TJUNCT_PP4) that are going to be created.
 - f) Click **OK** to save and close the dialog.

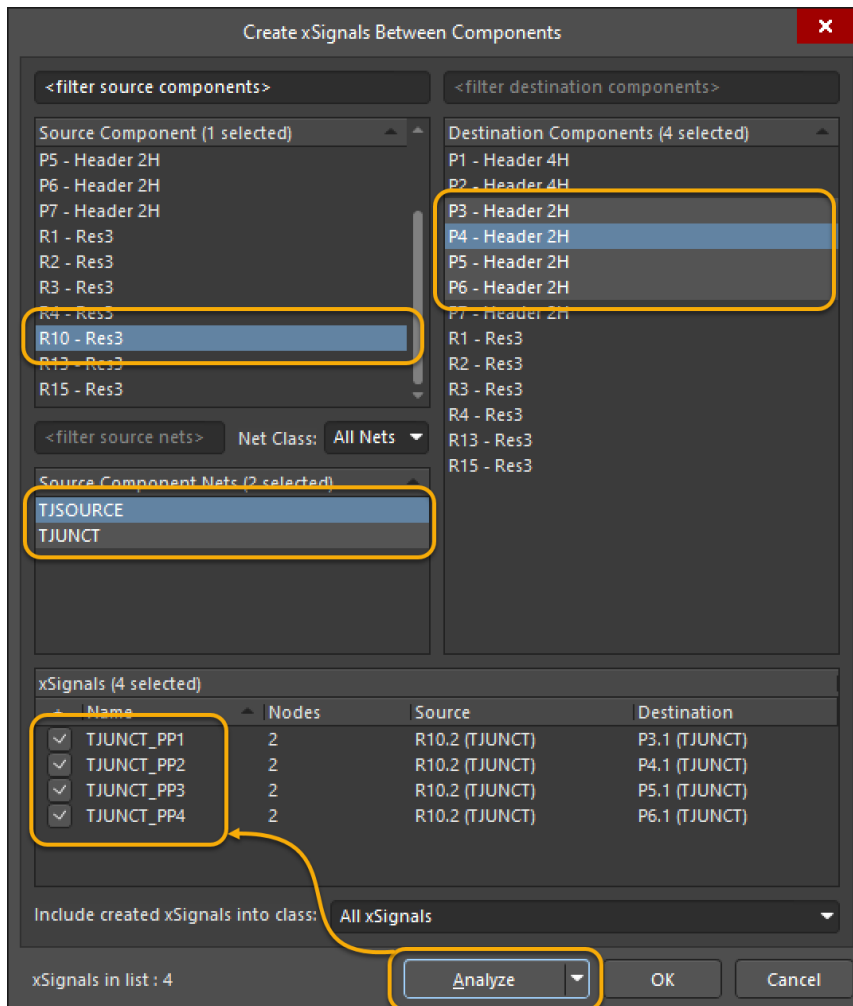



Figure 16. xSignals creation for the net with T-Junctions



To simplify the example, we defined the Signal paths between R10 and the Headers P3 to P6. It would also be possible to start the signal path from P7, with R10 as serial resistor, and stop the Signal Paths at the headers P3 to P6.

38. We will explore our xSignals classes by going to the **Design** menu and select **Classes**.
39. Using Figure 17 as a reference:
 - a) Select the *xSignal Classes* section.
 - b) Right-click on it and **Add Class**.
 - c) Rename the class as T-Junction.
 - d) Select and add the new xSignals TJUNCT_PP1 to TJUNCT_PP4 to the class using the  button to add them as *Members*.
 - e) Press **OK** to close the *Object Class Explorer* once the xSignals have been added.

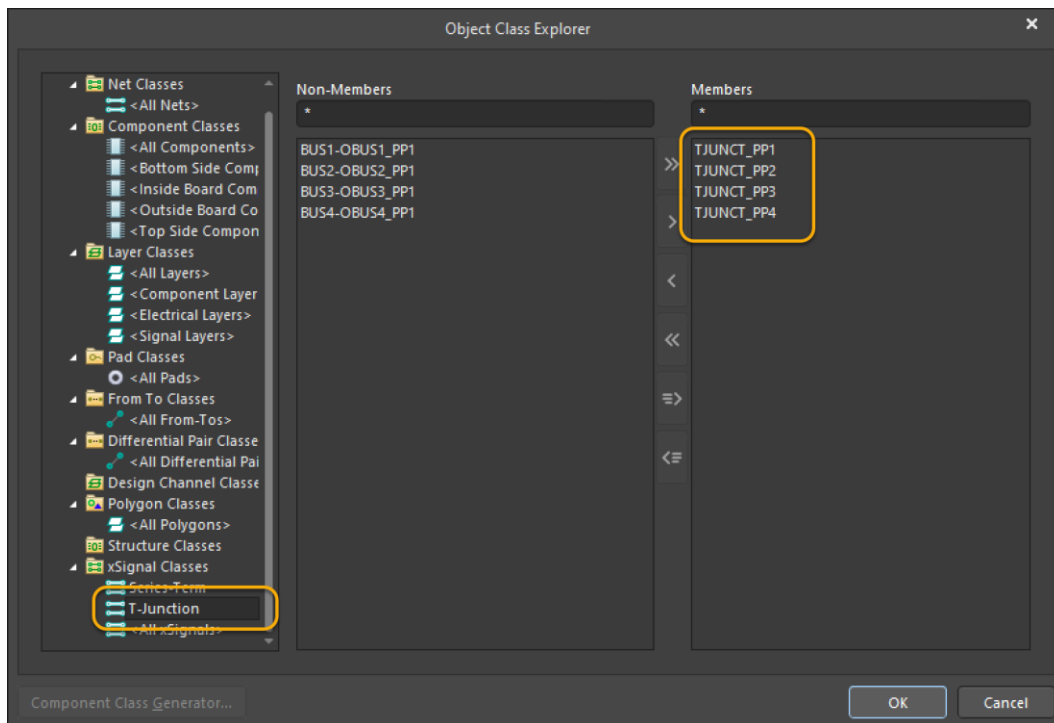


Figure 17. New xSignal class for the T-Junction net

40. Change your focus back to 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_PP1 to TJUNCT_PP4 to see the signal path as shown in Figure 18.

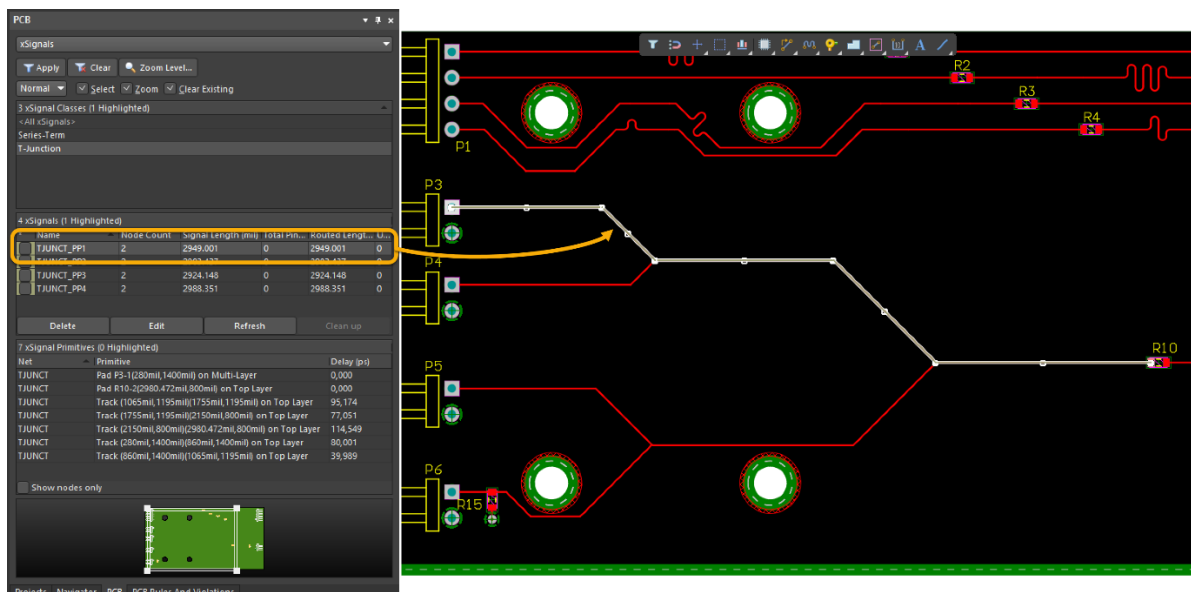


Figure 18. One signal path from the T-Junction xSignal class

1.6.2 Rules for xSignals from net with T-Junction

41. Open the *PCB Rules and Constraint Editor* from the **Design** menu. We will be creating a new *Length Rule* for the *T-Junction* xSignals.
 - a) In the *High Speed* section, open the *Length* rule category.
 - b) Right-click on the *Length* category and add a **New Rule**.
 - c) Double-click on the new rule so we can configure its settings.
 - d) Configure the rule with the following parameters using Figure 19 as a reference.
 - i) Change the Name to *Length_TJunction*.
 - ii) In the *Query* section, *Where the Object matches*, select **xSignal Class** from the drop-down menu.
 - iii) From the second drop-down menu, select the **T-Junction** class.
 - iv) Change the Values in the *Constraints* area to
 - Enable the **Length Units** option
 - *Minimum:* 3900mil
 - *Maximum:* 4100mil
 - v) Click on **Apply** to save the modifications.

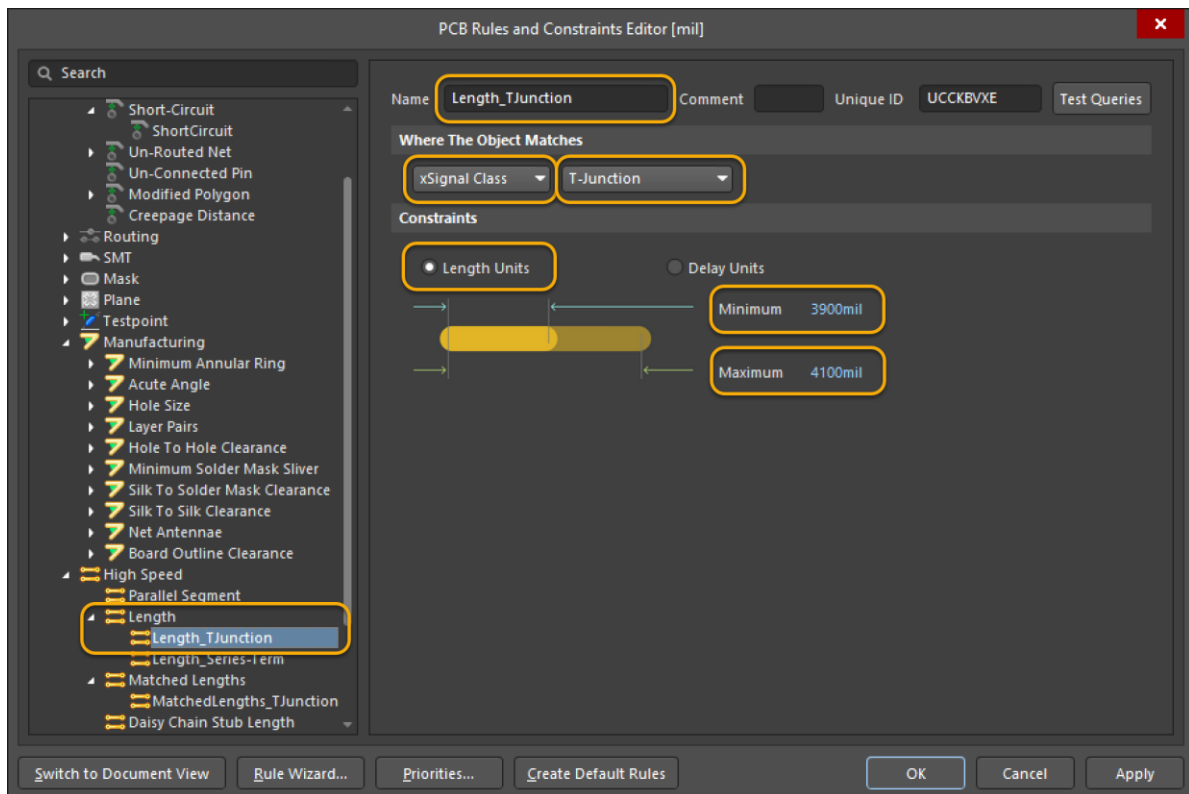


Figure 19. *Length_TJunction* rule

42. With the *PCB Rules Constraints Editor* still open, we will also create a *Match Length* rule. This rule specifies the allowable difference in net lengths. This rule is essential in a high-speed design, where the challenge is not just about how long it takes the signals to arrive, but how important it is that the specified signals arrive at the same time.
43. From the *High Speed* rule category, select the *Matched Lengths* rule.
44. Right-click on *Matched Lengths* and select **New Rule**.

45. Double-click on the new rule to configure its settings.
46. Using Figure 20 as a reference, set the following rule parameters as shown below.
 - a) Change the **Name** to `MatchedLengths_TJunction`.
 - b) In the Query section, *Where the Object matches*, select **xSignal Class** from the drop-down menu.
 - c) Select from the second drop-down, select the **T-Junction** xSignal class.
 - d) Change the Values in the *Constraints* area to:
 - i) Enable the Length Units option
 - ii) *Tolerance*: 10 mil
 - iii) Enable the **Group Matched Lengths** option
 - e) Click **OK** to save the modifications and to close the dialog.

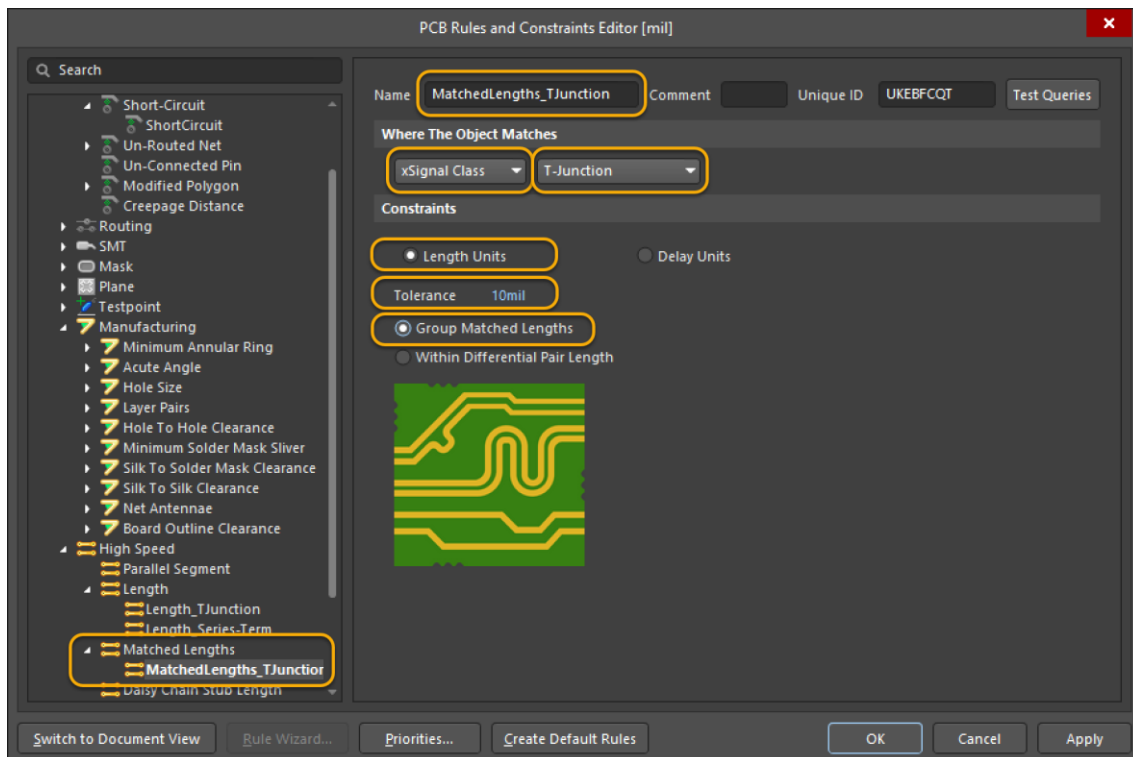



Figure 20. Matched Lengths rule for TJunction

47. Change the focus to 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 break the Length rule as they're highlighted in orange.
 - c) Click the **Clear** button at the top of the *PCB* panel to clear any selections.
48. We will need to Length tune all four TJUNCT xSignals, as shown in Figure 21.
49. From either the **Route** menu or from the *ActiveBar*, start the **Interactive Length Tuning** command.
50. 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) *Max Amplitude*: 120mil with *Step*: 10mil
 - b) *Radius*: 30mil with *Step*: 10mil
 - c) *Style*: Rounded

51. Click on 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 21.
52. Right-click to exit the command once completed.

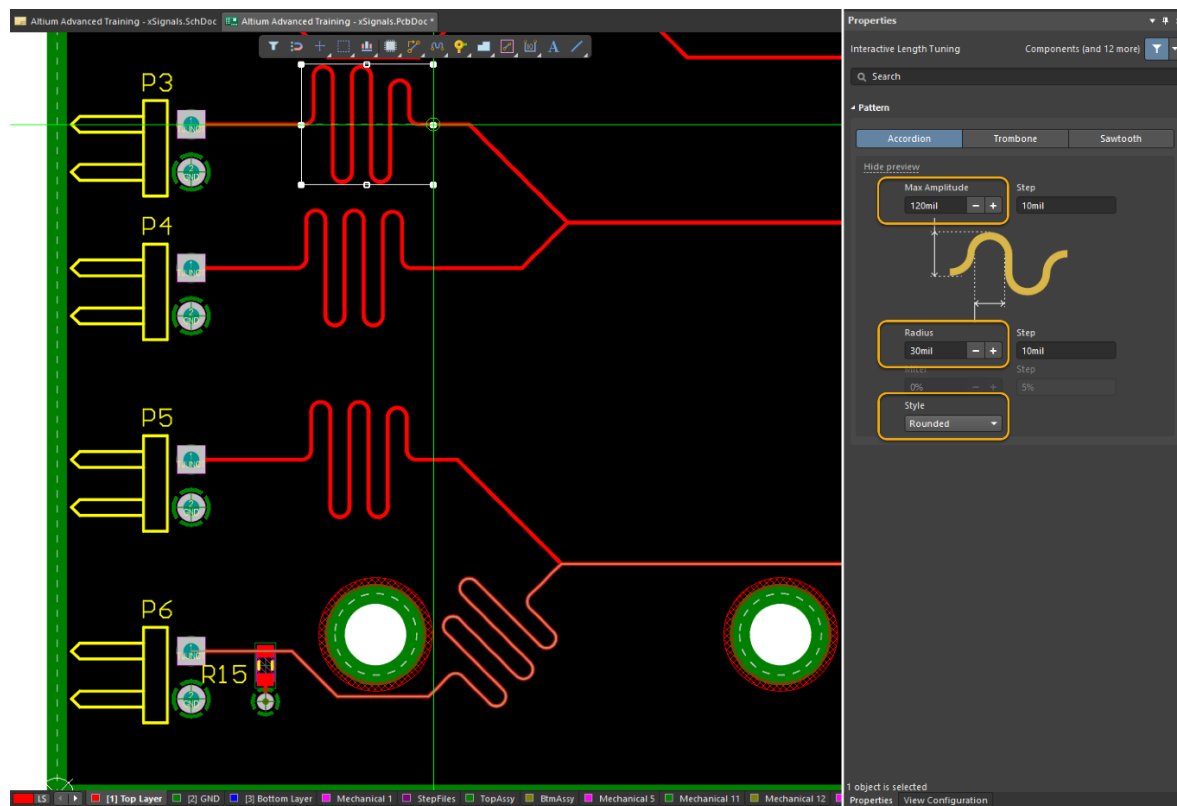


Figure 21 Tuned T-Junction xSignal Class.

1.7 Creating xSignals using the xSignals Wizard

Next we will create the xSignals for the T-Junction Signal paths with the xSignal wizard. As a second modification, we will add some new tuning patterns for the length tuning.

1.7.1 Preparation

53. Add the xSignals Wizard.PcbDoc document to the Projects (*Project* panel, right click command - **Add existing to project...**) and open the PCB file.
The PCB is included in project folder.

1.7.2 Running the xSignal Wizard

54. From the **Design** menu, select **xSignals**, then **Run xSignals Wizard**.
55. On the pages that appear in the Wizard, do the following:
 - a) *Initial Welcome* page: Click **Next**.
 - b) *Select the Circuit* page: Select **Custom Multi-Component Interconnect** and click **Next**.
 - c) *Select the Source Component* page: Change the Filter to R, select **R10** and click **Next**.
 - d) *Select The Source Nets* page: Select **TJUNCT** and click **Next**.
 - e) *Select Destination Components* page: change the filter to P, select **P3, P4, P5, P6** and click **Next**.
 - f) *xSignals Routes* page: No changes needed, click **Next**.

- g) *xSignals Length Tuning* page:
 - i) Enable the option **Yes, I want checked xSignal to have the same routed length**
 - ii) Check that all four xSignals are selected.
 - iii) Keep the default names, `xSignals_R10_P3, P4, P5, P6_MatchLengthsClass` and `MatchLengths_xSignals_R10_P3, P4, P5, P6`
 - iv) Set the **Length Tolerance** to `20mil` and click **Next**.
- h) *Report and continue* page: ensure that the **Finish Wizard** is selected and click **Finish**.


1.7.3 Creating Rules after using the xSignal Wizard

56. Open the *PCB* panel and ensure that xSignals is selected from the drop-down menu.
 - a) Note how 2 xSignal classes were created from the wizard.
 - b) Select one of the new xSignal classes from the PCB panel and you'll see that three xSignals are breaking the *Matched Length* rule we just created with the wizard.
57. From the **Design** menu, select **Rules...** You can also use the **D»R** shortcut keys.
58. Open the section *High Speed* category and expand the *Matched Length* rules. Note that the Wizard created a *Matched Lengths* rule – not a *Length* rule. For our training example, we also need a *Length* rule.
59. Under the *Length* rule category, right-click and add a **New Rule**.
60. Double-click on the new rule so we can configure its settings:
 - i) *Name*: `Length_TJunction`.
 - ii) In the Query section, *Where the Object matches*, select **xSignal Class** from the drop-down menu.
 - iii) From the second drop-down, select one of the xSignal classes created by the wizard.
 - iv) Change the Values in the *Constraints* area to *Minimum*: `3900mil` and *Maximum*: `4100mil`.
61. Click **OK** to close the *PCB Rules and Constraint Editor*.

1.7.4 Length Tuning

1.7.4.1 Length tuning with Trombone

Next we will length tune the T-Junctions signal paths. Instead of using the *Accordion* style, we will now use the *Trombone* pattern.

62. Open the *PCB* panel and select the `xSignals_R10_P3, P4, P5, P6` xSignal class. Note how all xSignals break the Signal Length/Routed Length rule.
63. Click **Clear** in the *PCB* panel.
64. Start the **Interactive Length Tuning** command from the **Route** menu, or from the *ActiveBar* .
 - a) Your cursor will turn into a crosshair. Do **not** start tuning yet.
 - b) Press the **TAB** key to open the *Properties* panel. Using Figure 22 as a reference to change the following settings:
 - i) *Pattern*: `Trombone`
 - ii) *Style*: `Rounded`
 - iii) *Radius*: `50mil`
 - iv) *Step*: `10mil`

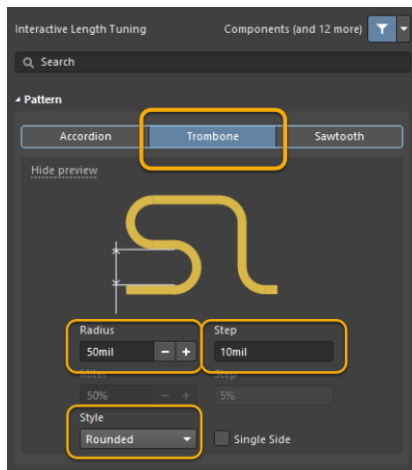



Figure 22. Trombone pattern

65. Click on the Pause icon  to continue the length tuning.
66. Start length tuning each route for components P3, P4 and P5 as shown in Figure 23. Once the Length Guide Indicator turns greens, this means the routes will adhere to our rule.



The Trombone does not need to be perfect. After it has been placed, you can optimize it after you placed it by selecting it, and moving it as you wish.

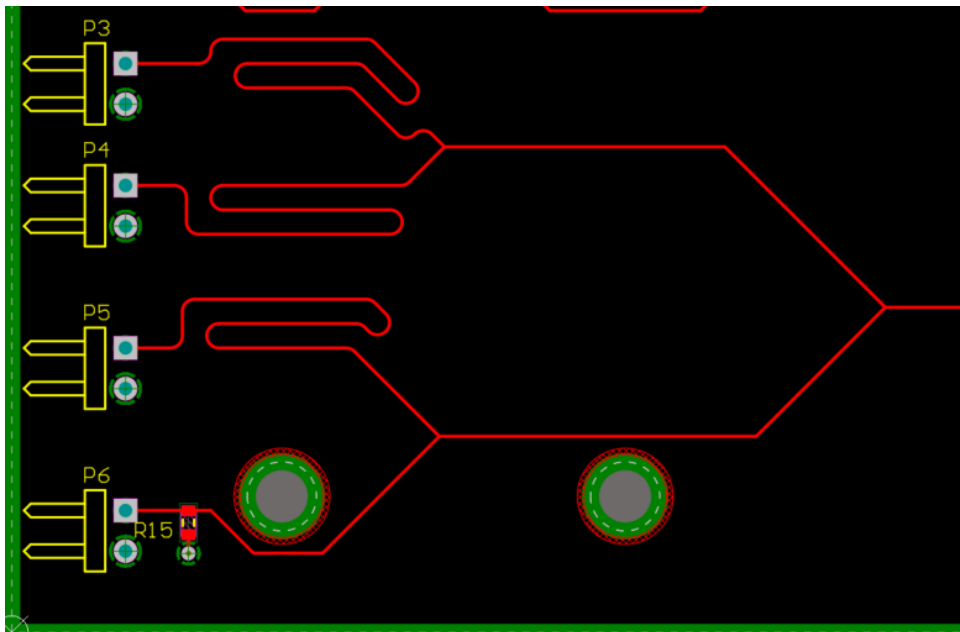


Figure 23. First Trombone length tuning

1.7.4.2 Modify Trombone

The last signal path for P6 is too close to a board cutout, this means we will not have enough room to length tune the path in this area. We will need a *Trombone* pattern with more loops.

67. Use the Length Tuning command again to tune the signal path for P6, to the right of the board cutout as shown in Figure 24. The Signal Path will be still too short, but we will change that with the next steps.
68. Left-click to place the *Trombone* and right-click to exit the command.

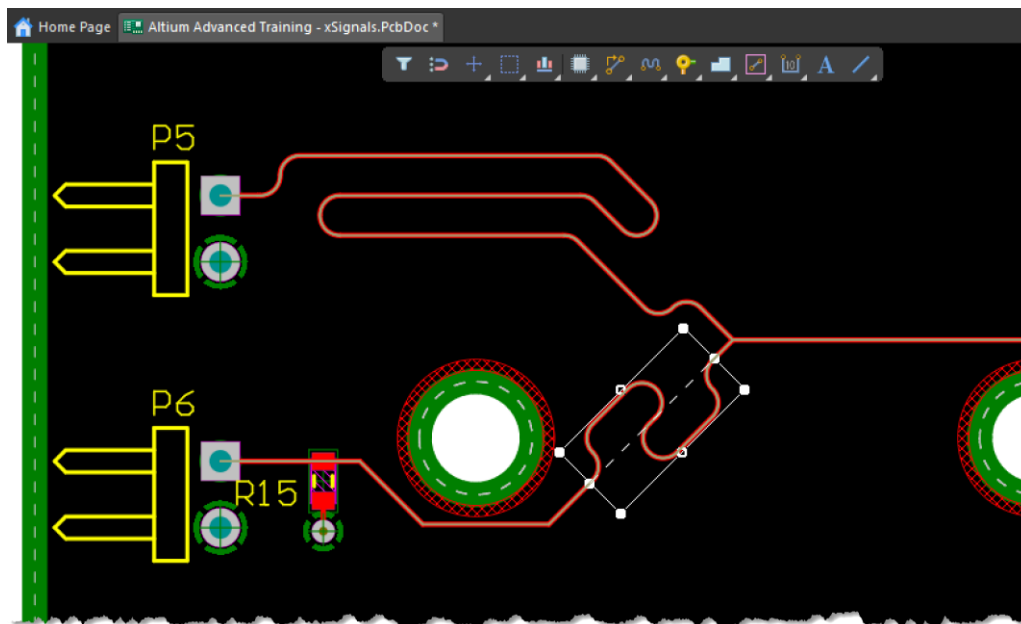


Figure 24. First Trombone for Signal Path P6

69. Select the Trombone you just placed. You'll notice that there's a bounding box around it. This is called the Sleeve.
70. Once selected, you'll see the vertex points as shown in Figure 25.
71. Click and Drag the lower vertex point away from the board cutout. By changing the size of the bounding box, Altium will either add loops, remove loops, or change the length of the existing loops to update the signal length.

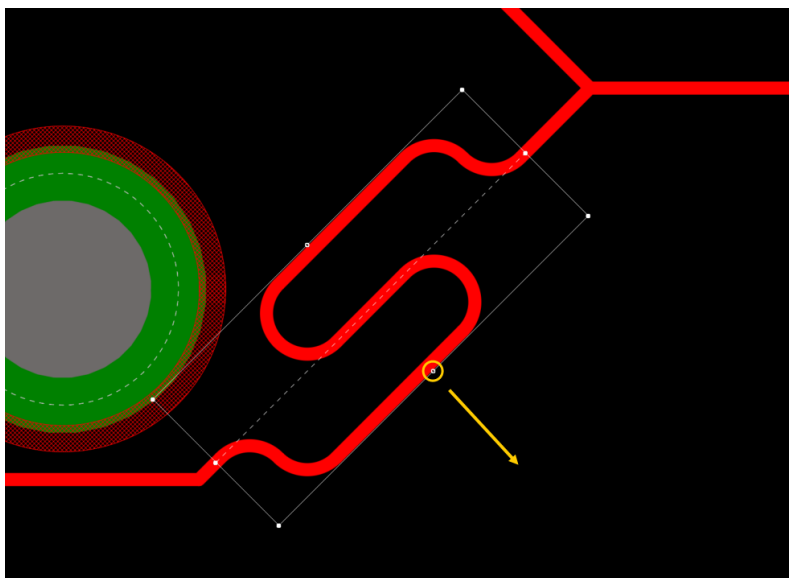


Figure 25. Adjusting the bounding box of the Length Tune

72. Continue to drag the vertex until enough loops are added so that it adheres to the length rule as indicated by the *Length Guide Indicator* as shown in Figure 26.

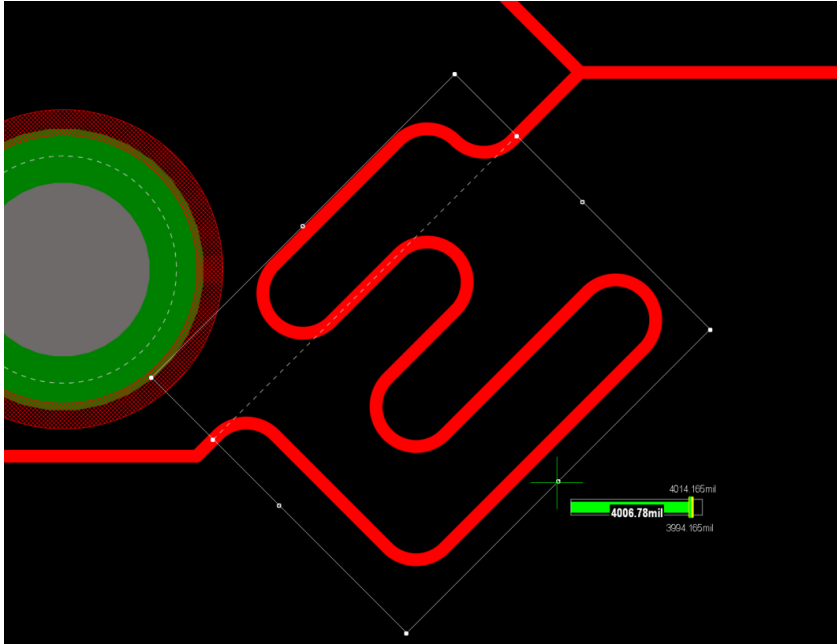


Figure 26. Adjusting the Length Tune to adhere to the Length rule

73. Open the *PCB* panel to ensure that neither of the xSignal classes have any errors, and that all signal routes adhere to our rules.

1.8 Unions and Length Tuning

When Length Tuning is added to your design, every route that has been tuned will be listed in the *PCB* panel for you to easily review the tunes in your design.

74. Still in the *PCB* panel, select **Unions** from the drop-down menu as shown in Figure 27.
75. In the pane *Union Type* pane, select **Interactive Length Tunings**.
76. In the next pane, you'll find all of the length tuned objects that you've placed during this exercise.



In case you placed a Length Tuning object without a pattern or unrequired tuning, with incorrect in-correct value e.g., the radius or amplitude, you can easily find this “empty” Length Tuning object in the *PCB* panel by selecting **Unions - Interactive Length Tunings**.

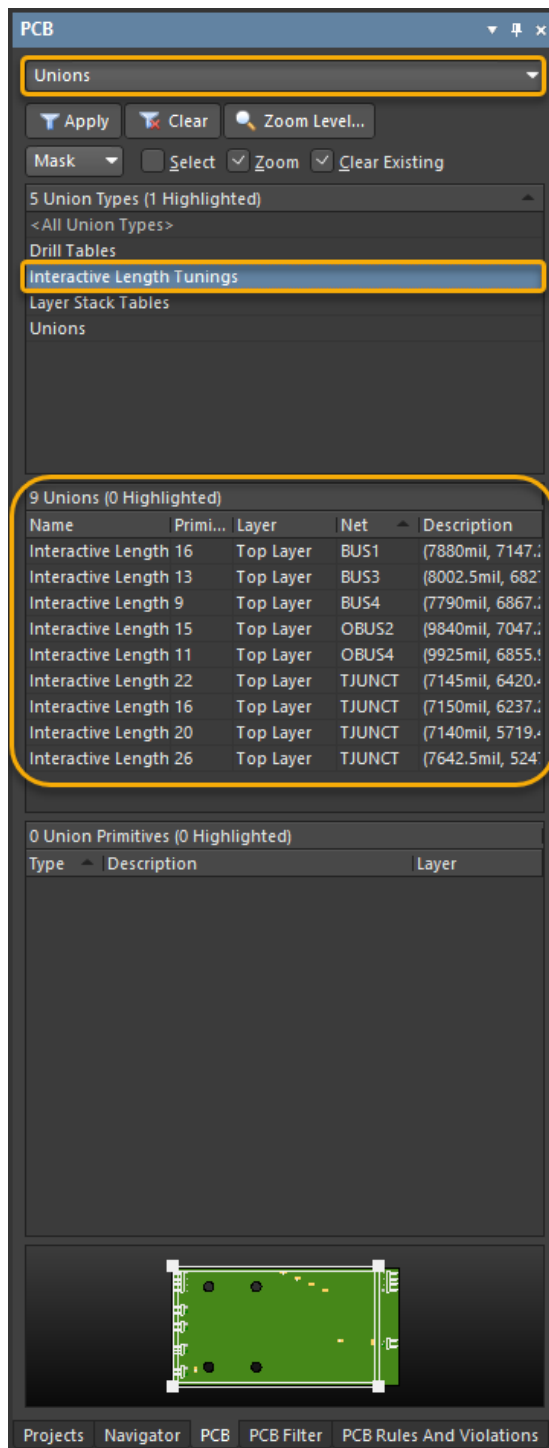


Figure 27. Union PCB Panel

77. Feel free to save your modifications.
78. **Close the project and any open documents.**

Congratulations on completing module

Using xSignals in a High-Speed Design

from the

Altium Designer Advanced Course

Thank you for choosing Altium Designer