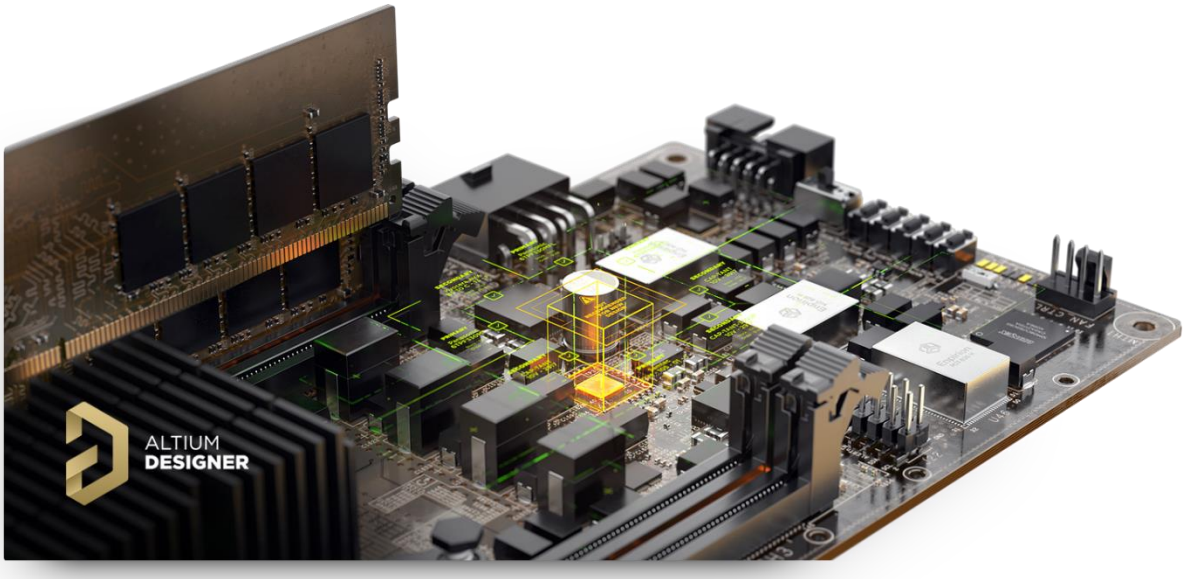

Altium Designer

Advanced Course

Module: Using xSignals in a High-Speed Design with Differential Pair

Table of Contents

Using xSignals in a High-Speed Design with Differential Pair

1.1 Purpose

In this Module you find some information about the combination of xSignals and Differential Pairs with 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**
Interactive Diff Pair Length Tuning **U » P**

1.3 Preparation

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

1.4 xSignal and Differential Pairs

1.4.1 Schematic Information

Next we will load a predefined project source files that show the configuration for xSignals with Differential Pairs.

3. Open the `Differential Pair Setup.SchDoc` document.
4. Notice the two separate USB paths, U1 to U3 / USB1A (Signal D1 – J_D1) and U2 to U3 / USB1B (Signal D2 - J_D2).
5. To define the differential pair, in combination with PCB Rules and PCB Class information, a Blanket was placed. The Differential Pair directive and a Parameter directive was attached to the Blanket, as seen in Figure 1.

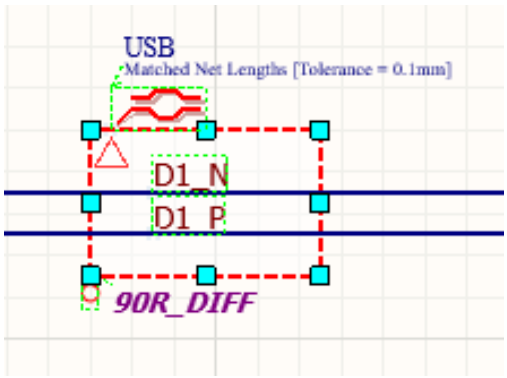

Figure 1. Differential Pair definition

6. After placing the Differential Pair directive, it was updated with a Differential Pair Class Name and a Matched Net Lengths PCB Rule, as seen in Figure 2.

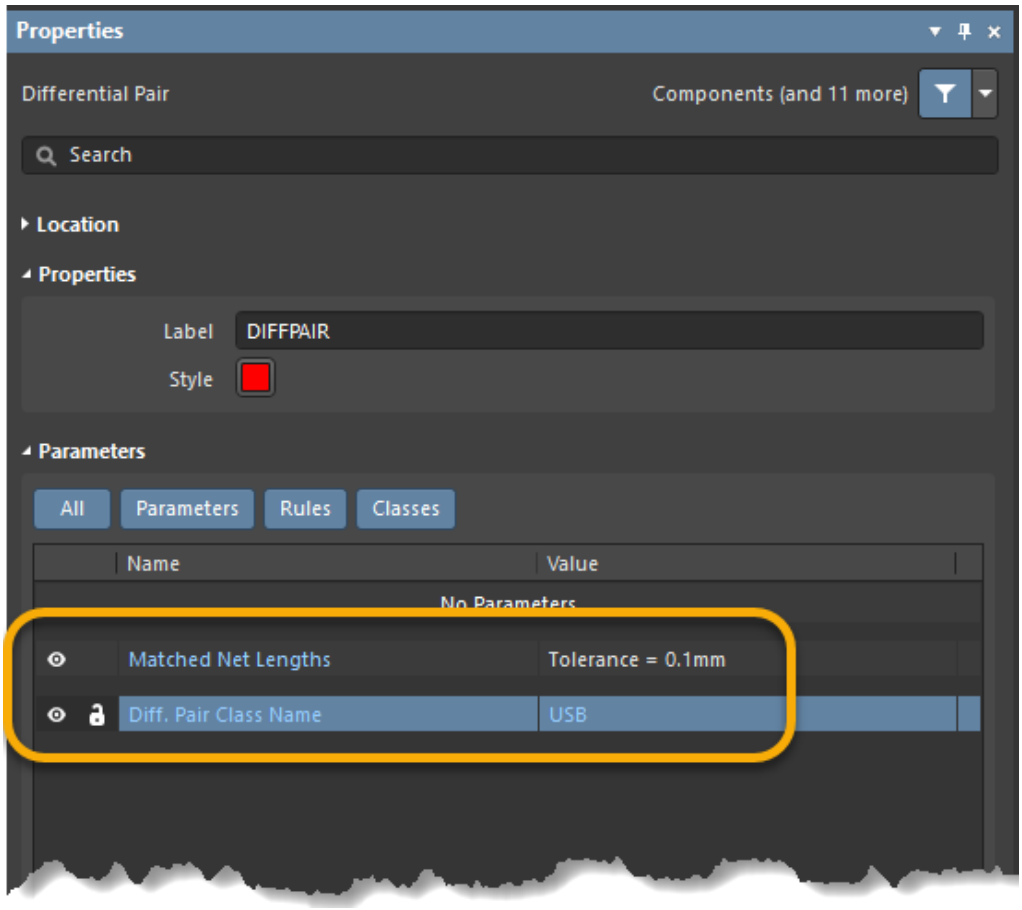

Figure 2. Diff. Pair directive with PCB Rule and Diff. Pair class information.

1.4.2 **PCB Rules**

7. Open the PCB `Differential Pair xSignals Setup.PcbDoc`.
8. Run the DRC, **Tools » Design Rule Check…,** the DRC will report two Matched Net Lengths Violations.
9. Open the *PCB Rules and Constraint Editor* (**D - R**)
   a) At the *Routing - Differential Pair Routing* section you see the rule `DiffPairsRouting_90R` for the Signals **Diff Pair Class** - **USB** that uses the *Impedance Profile* **D90,** defined in the *Layer Stack Manager*.
   b) The section *Highspeed – Matched Lengths* has three rules.
      i. `MatchedLengths_GROUP` for the length tuning of the xSignals class `USB_All`.
      ii. `MatchedLengths _All _Diff_Pairs` for matching the length within a differential pair.
      iii. `Schematic Matched Net Lengths` defined for the Net Class 90R_Diff.
   c) Close the *PCB Rules and Constraint Editor* without saving modification.
10. Open the *PCB* Panel.
    a) Select from the drop-down list the view **Differential Pair Editor**,
    b) Select the Class **USB** to see the four Differential Pair definitions D1, D2, J_D1, J_D2.
    c) Select from the drop-down list the view **xSignal** to see the three xSignal Classes **USB_All**, **USB_TOP** and **USB_Bottom** and the individual xSignals. The signals from USB_Bottom have a routed length of 39.9mm, USB_TOP signals a routed length of 30.1mm.

The xSignals in this Project were already created for you. The xSignals could be created with one of the methods you learned in the previous sections. Additional xSignal information are defined,but are not used for the USB Signal path in this training example.

1.4.3 Interactive Diff Pair Length Tuning

11. Start the **Interactive Diff Pair Length Tuning** command from the **Route** menu, or from the *ActiveBar* . Tune the `USB_TOP` to the target Length of 39.9mm as seen in Figure 3.
    a) Style: **Mitered Lines**
    b) *Max Amplitude*: `1mm` with *Step*: `0.1mm`
    c) *Space*: `0.3mm` with *Step*: `0.1mm`
    d) Miter:15%

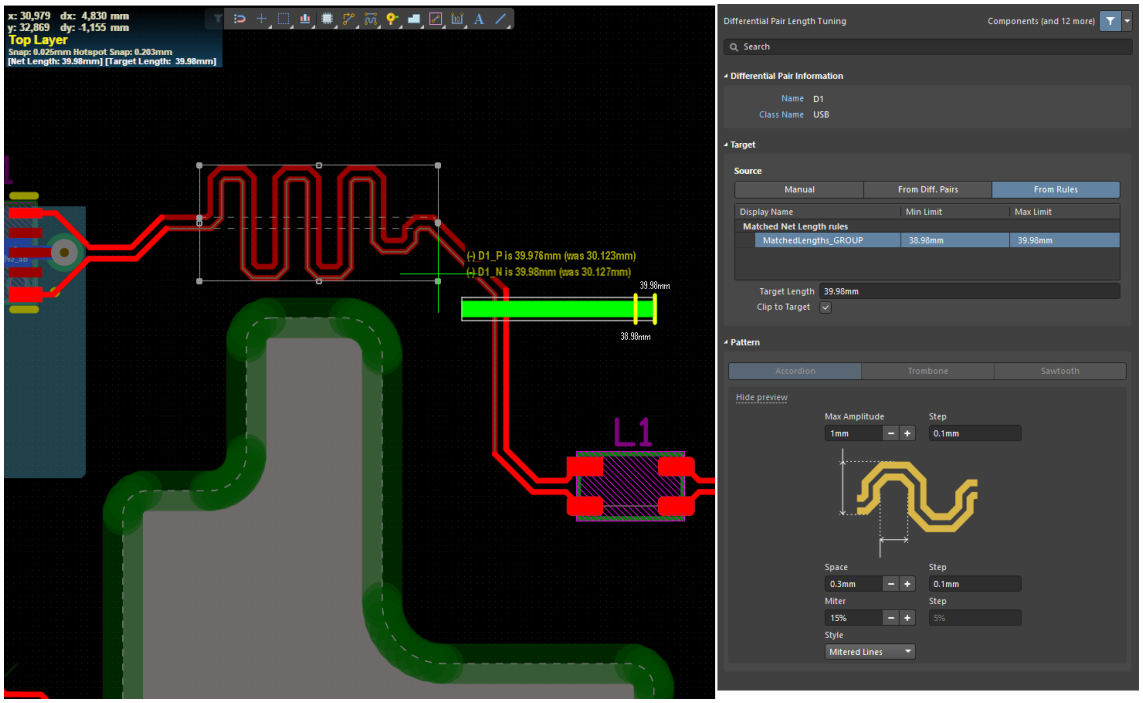

Figure 3. Differential Pair length tuning for USB_TOP

12. Run the DRC, **Tools » Design Rule Check…,** to verify that the design is error free.
13. Close the Project and PCB document, feel free to save your modifications.
14. **Close the project and any open documents.**

Congratulations on completing module

Using xSignals in a High-Speed Design with Differential Pair

from the

Altium Designer Advanced Course

Thank you for choosing Altium Designer