# Altium Designer

## Advanced Course

Module: Length Tuning Routing

Software, documentation and related materials:

# Table of Contents

# Length Tuning Routing

## 1.1 Purpose

Length matching in Altium Designer provides an easy method for matching lengths for critical high speed signals. Length matching can be performed between signal nets, between differential pairs, and between signals of a differential pair. Lengths can be matched to a length rule, a routed length, or a manually entered target value.

In this exercise, we will be tuning matched lengths to a targeted length and a tolerance to nets within the same differential route class. The Differential Pair Class, REC_DIFF needs to have a routed length between 2100 and 2200 mil total length. The length matching needs to be 10mil between legs of a differential pair and 100 mils between pairs.

## 1.2 Shortcuts

Shortcuts when working with Length Tuning Routing
**F1:** **Help**
**D-R:** **PCB Rules and Constraints Editor**
**T-D:** **open the DRC**
**U-P:** **Interactive Diff Pair Length Tuning**
**Shift+G** **Toggle Tuning Gauge On / Off**
**U-R_** **Interactive Length Tuning**
**CTRL+S:** **Save Document**

## 1.3 Preparation

1. **Close all existing projects and documents.**
2. Open the `Length Tuning Routing.PrjPCB` project found in its respective folder of the Advanced Training.
3. Open the `Net_Length_Tuning.PcbDoc` document from the *Projects* panel.

## 1.4 Interactive Length Tuning

### 1.4.1 Design Rules

4. From the **Design** menu, select **Rules**. This can also be done using the **D » R** shortcut keys.
5. Scroll down and expand the *High Speed* rule category as shown in Figure 1.
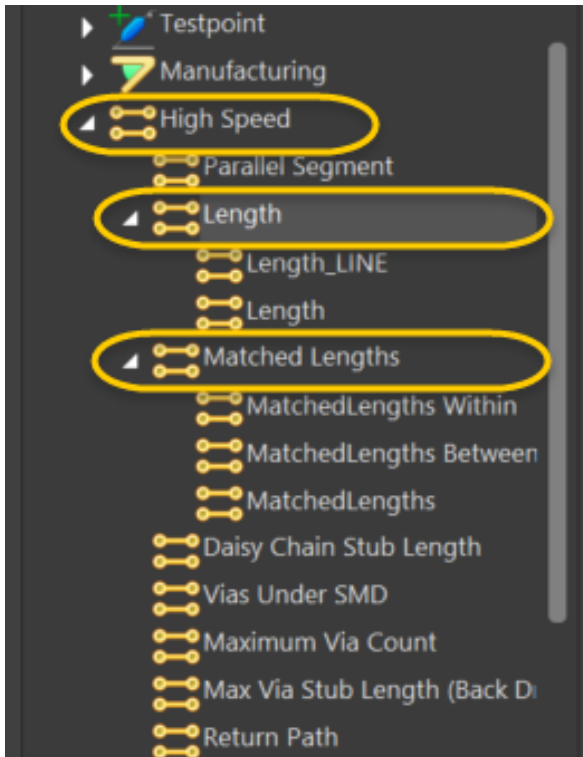
*Figure 1. High-Speed Design Rules*

6. Expand and select the various rules listed under the *Length* and *Matched* Length rule categories to see what rules will be applied during the Interactive Length Tuning.

7. Select **Cancel** to close the *PCB Design Rules and Constraints Editor*.

## 1.4.2 **Design Rules Check (DRC)**

We will now run the Design Rule Check to see if we have any design errors in regards to the length of our signals.

8. From the **Tools** menu, select **Design Rule Check…**

9. Click the **Run Design Rule Check…** button in the lower left corner of the dialog.

10. The *Messages* panel will appear, along with a report showing all of the Rule Violations in our design. Note that there are several length and matched length violations.

11. Close the *Design Rule Verification Report* as well as the *Messages* panel and return to the PCB document.

## 1.4.3 **Length Tuning for Differential Pairs**

12. Open the *PCB* panel if not already opened.

13. Select the **Differential Pairs Editor** from the drop-down menu.

14. Ensure the **Zoom** checkbox is checked. The zoom level can be adjusted by clicking the **Zoom Level…** button and adjusting the slider.

15. Click on the REC_DIFF differential pair class to zoom to the two pairs on the PCB as shown in Figure 2. Note that the routed lengths are shown for each pair.
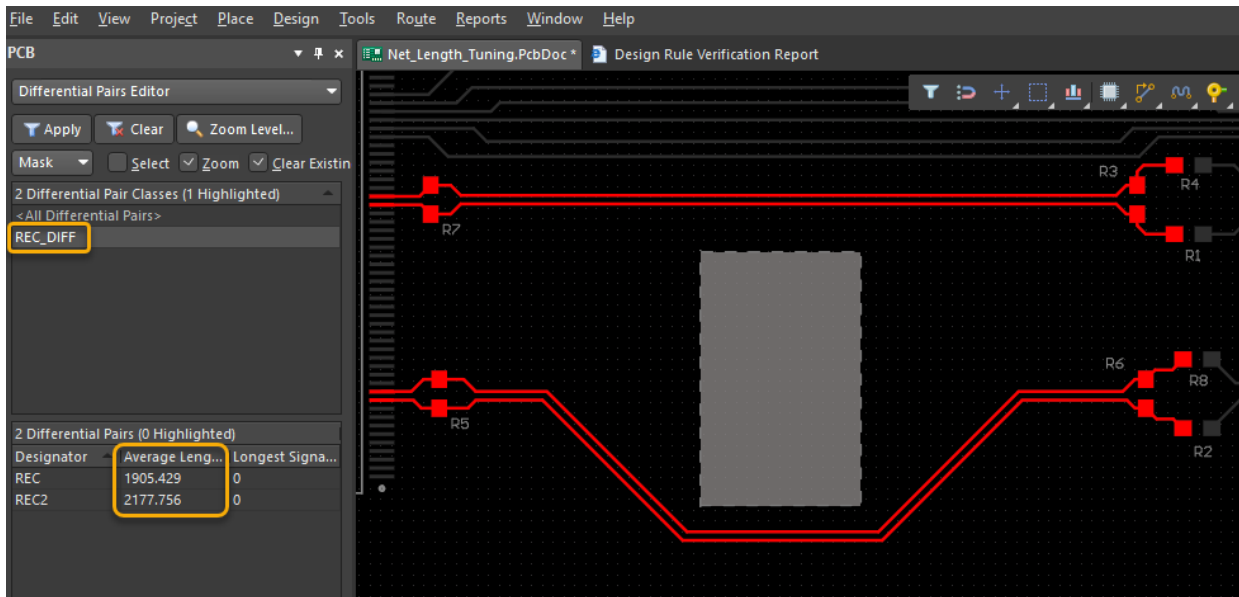
*Figure 2. Differential Pairs Editor – PCB panel*

16. From the **Route menu,** select **Interactive Diff Pair Length Tuning**. Alternatively, you can access this command from the ActiveBar 🔀, or by using the **U » P** shortcut keys.

> There are 3 different accordion styles to choose from. Accordion, Trombone and Sawtooth. To change the accordion style, you must hit the **TAB** key before left-clicking on the route to start the tuning process. We will use Accordion in the exercise.

17. With the crosshair on your cursor, click on one of the tracks in the REC differential pair. This is the differential pair near component R7.
18. Press the **TAB** key to pause the length tuning process so that changes can be made in the *Properties* panel.
19. In the *Properties* panel, in the *Target* section, set the *Source* to **From Diff. Pairs** as shown in Figure 3.
20. Click on the REC2 differential pair. This will ensure the target length will be pulled from that differential pair.
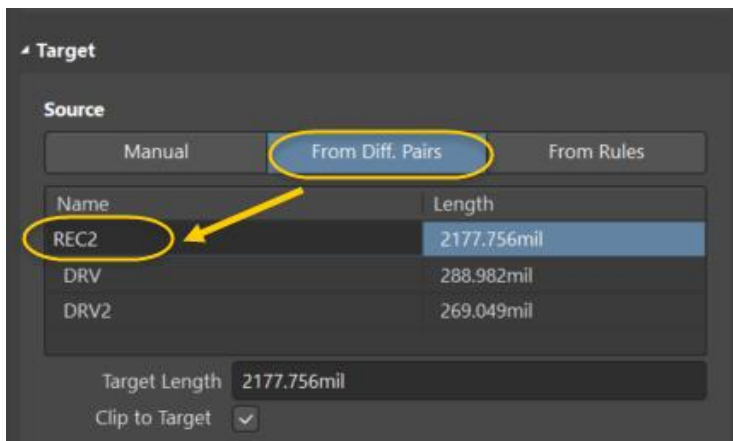


*Figure 3. Target length for the differential pair length tuning*

21. In the *Pattern* section, set the values for the accordions as shown in Figure 4 below.

    a) *Max Amplitude* = `30`

    b) *Amplitude Step increment* = `10`

    c) *Space* = `20`

    d) *Space Step Increment* = `10`

    e) *Style:* Mitered Lines

> The tuning style must be determined before placing the length tune. If you do not have the Accordion style as the default style, repeat the length tune command and hit the TAB key before placing the tune. This will allow you to change the style.
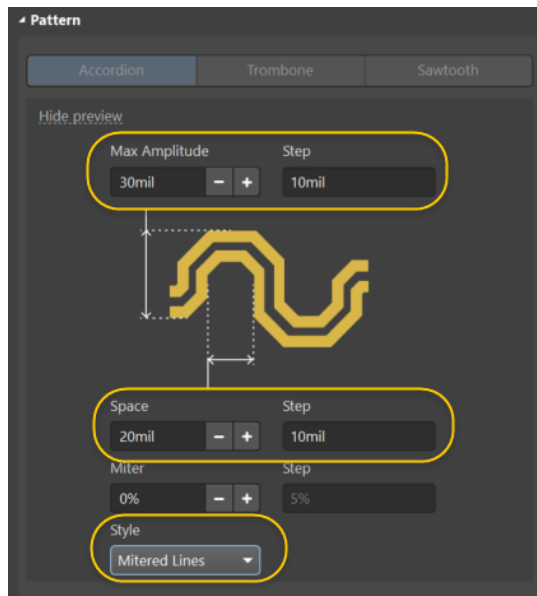


*Figure 4. Differential Pair Tuning properties*

22. Click the **Pause** icon to return to the length tuning command.

23. Accordions will be added by moving the mouse along the differential pair route until the length tuning gauge turns green. The gauge can be enabled or disabled using **Shift+G** during the tuning process as shown in Figure 5.
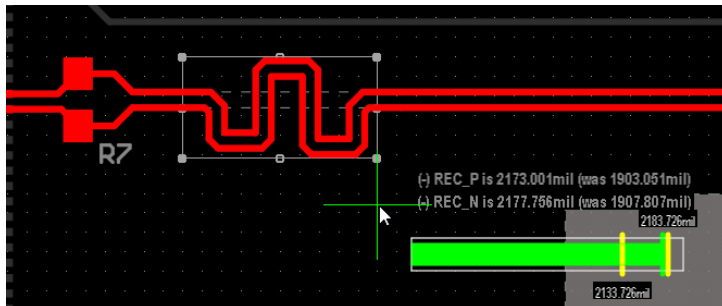


*Figure 5. Length Gauge turns green when within tolerance*

24. Left click to commit the added accordions, then right-click to terminate the command.

## 1.4.4 **Length Tuning for LINE Signals**

25. Open the *PCB* panel.

26. From the drop-down menu at the top of the *PCB* panel, select **Nets**.

27. In the *Net Classes* section, select LINE.

28. In the *Nets* section, you will see all 6 LINE signals, along with their Signal Length as shown in Figure 6. The Signal Lengths are highlighted in orange because they do not meet the desired length that is determined in the design rule.
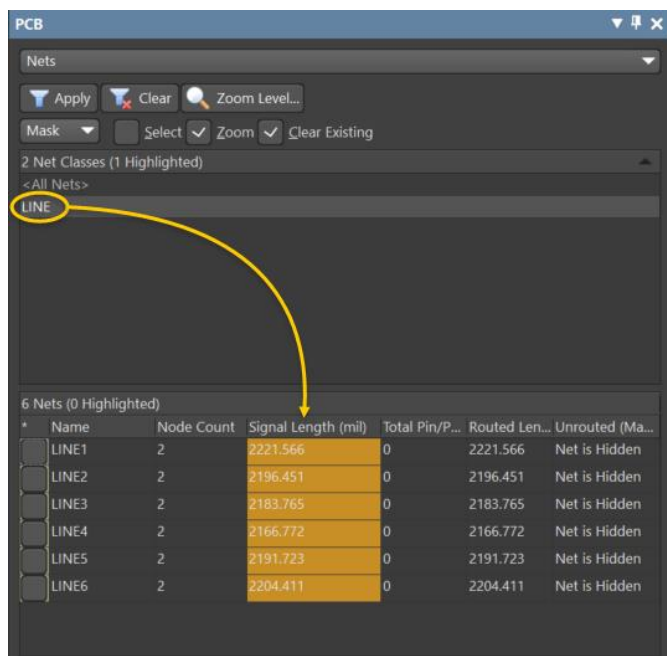


*Figure 6. LINE signals and lengths*

29. From the **Route** menu, select **Interactive Length Tuning**, or use the **U » R** shortcut keys. Alternatively, you can access this command from the ActiveBar .

30. Click on one of the traces in the LINE class to start adding accordions.

31. During the tuning process, hit the **TAB** key to bring up the *Properties* panel.

32. For the *Source*, under the *Target* section*, select* **From Rules** as shown in Figure 7.
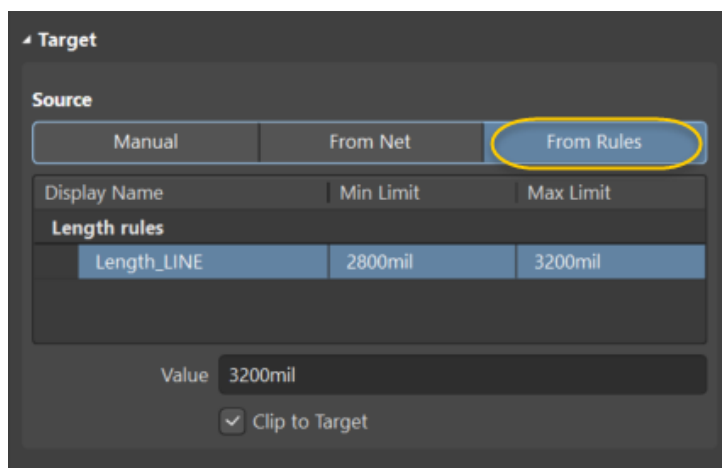


*Figure 7. Source length for signal length tuning*

33. Click on the **Pause** icon 〚‖〛 or hit **Enter** to continue placing the accordions.

34. Move your cursor until you see the length gauge turn green. Then, left-click to complete the tuning.

35. With the tuning command still on your cursor, continue to length tune the remaining 5 nets.

36. Your end result should look similar to Figure 8. The *Signal Lengths* in the *PCB* panel should no longer have an orange highlight, showing that we're adhering to the design rule.
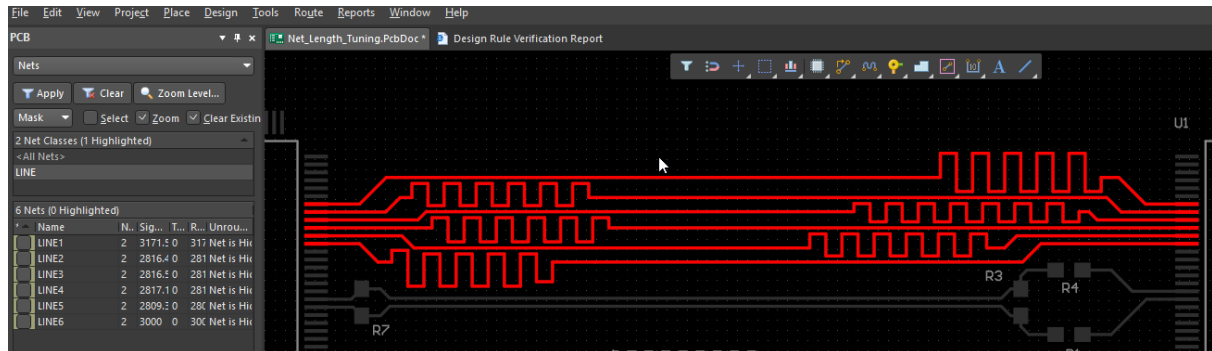


*Figure 8. Staggered tuning on Line tracks*

> ⚠ The target length should not be shorter than original length, since the routed track can be made longer by adding deviations, but not made shorter.

37. Lastly, run a **Design Rule Check…** from the **Tools** menu. There should no longer be any violations.

38. Feel free to save the changes.

39. **Close the project and any open documents.**

# Congratulations on completing module

Length Tuning Routing

## from the
Altium Designer Advanced Course


Thank you for choosing Altium Designer

**Altium**®