



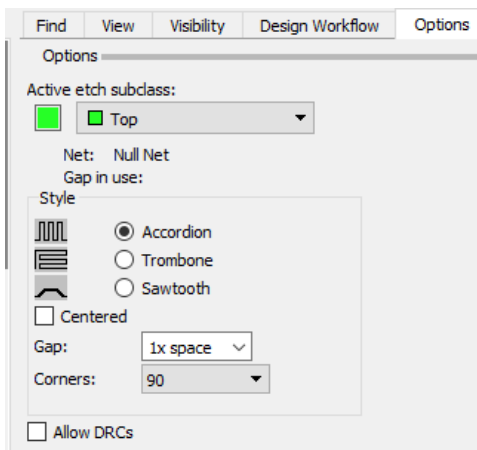
Introduction

Many high speed designs today require PCB traces to match in length, so that when the signal leaves the driver it will arrive at the receiver at exactly the same time within some small tolerance. To “length match” nets in a PCB used to be extremely time consuming. You would route all the nets, then measure the longest one, then on a net by net basis, add extra twists and turns into the remaining tracks so that eventually the group of tracks that you need to match are the same length. Allegro PCB Designer and OrCAD PCB Designer Professional both use the total etch length command as well as the option to use Relative Propagation Delay (or Matched Groups) to define this capability. Both tools offer the Delay and Phase Tuning functions available from the Route menu. This Application note describes how to use these two features. To define Matched Group properties or Total Etch Length properties please refer to either the “defining diff pairs” app note or the “net scheduling” app note, both available from Parallel Systems support.

Delay Tuning

The delay tune command interactively elongates nets not meeting minimum timing or length constraints. It can be used on single nets or on differential pairs. Three styles of elongation are available: trombone, accordion, and sawtooth. Amplitude is user controlled by moving the cursor in a positive or negative direction. Please refer to the screenshot on the following page to see the “Amplitude” definition.

To add “delay” use the Route - Delay Tune command, then look to the Options fold out menu (on the right hand side of the screen) for settings:-



Active etch subclass - Indicates the etch subclass currently showing in the design.

Net - Specifies the name of the net being tuned. (not selected in example screenshot).

Gap in use - Indicates the gap being used for current delay tuning. For additional information, see the description of Gap below.

Style - Based on your design needs, choose one of these elongation styles: Accordion, Trombone, or Sawtooth.

Centered - Check this box to centre the elongation around the chosen cline segment. The mouse position defines one-half the amplitude. If you do not centre the elongation, then all the elongation is on one side of the specified cline segment.

Gap - Represents the edge-to-edge distance between adjacent parallel cline segments of the elongation etch. Up to seven previous settings are remembered and are available on the drop-down list. You can enter:

Special values, such as [N] x width, where [N] is an integer that means N times the line width (default). Special values such as [N] x space, where [N] is an integer that means N times the line-to-line spacing. For example, enter 5x space for 5 times the line-to-line spacing. A number such as 25 to represent the gap of 25 user units. The initial value is set to 1x width. The value of the gap, currently used, appears at the top of the Options tab.

Corners - Establishes how the elongation etch corners. Values are: 90, 45, and FullArc. Use FullArc to create a single semi-circle connecting the ends of the parallel cline segments of the elongation etch.

Miter size - Appears only if you choose 45 in the Corners field. The miter size controls the size of 45-degree corners. The miter size is the amount that is cut away from the pre-cornered segments. When cornering orthogonal segments, this is the x or y offset between the endpoints of the new segment. Previous settings are available through a drop-down list. You can enter special values, such as [N] x width, where N is an integer that means N times the line width. For example, if you type 4x, the value is 4x width. The initial value is set to 1x width. *Note:* If you type a value of 5 in this field, it results in a corner segment length of 7.1. In general, the segment length is approximately the square root of two times the miter-size value.

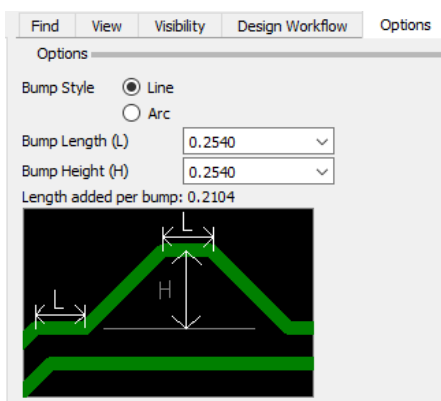
Allow DRCs - Check this box to ignore design rules, yet still report DRC errors, when creating the elongation rectangle. In this mode, elongation is created regardless of any elements that cause a DRC. A DRC color can appear on the trace, and when you have completed the elongation rectangle, DRC markers can appear. If you do not check this box, the editor does not create physical DRCs. If filling the full elongation rectangle results in a DRC, the editor does not let you make the rectangle. If the editor cannot determine an elongation rectangle that does not create a violation, then it does not create elongation etch.

Once set LMB (Left Mouse Button) the start point of the track you wish to add delay to and move the mouse in the direction of the added delay. The length will be modified accordingly. Use the real-time feedback bars that operate whilst using this command to confirm the actual lengths added, green meets the constraints, red does not.

Phase Tuning

Phase Tuning is an alternative to using the mouse guided delay tune command and offers the precision of finite length adjustment to signals that are length/phase constrained. It is especially effective on static or dynamic phase-constrained Differential Pairs where iterative etch compensation may be required at various points along the path of either member of the pair. Simply make a mouse click at any point on the cline path to add in a single-parameterized phase bump.

The command is located in the Route Menu of the PCB Editor. When invoked, parameters can be set in the Options panel. Select a style of Line or Arc then define its respective length/size parameters. The form will compute the added compensation for each bump before applying you it.



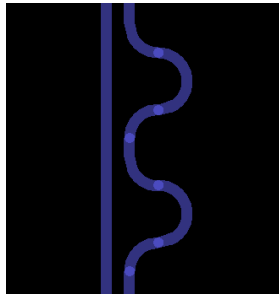
Bump Style - Defines bump as line or arc.

Bump Length - Defines the length of each bump. This length determines the minimum segment length between the new bump and the vertices of the original selected segment. The default value is two-times the minimum-line-width from the design CSet.

Bump Height - Defines the width of each bump. The default value is two-times the minimum-line-width from the design CSet.

Length added per bump - Shows the length of each bump that will be added to the cline. This value varies depending upon the value of bump length and height.

Some examples of the Phase Tune and Delay Tune are shown below.



Bump Style Arc



Bump Style Line

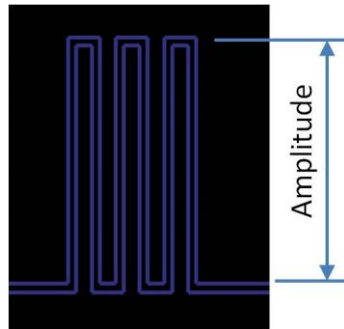
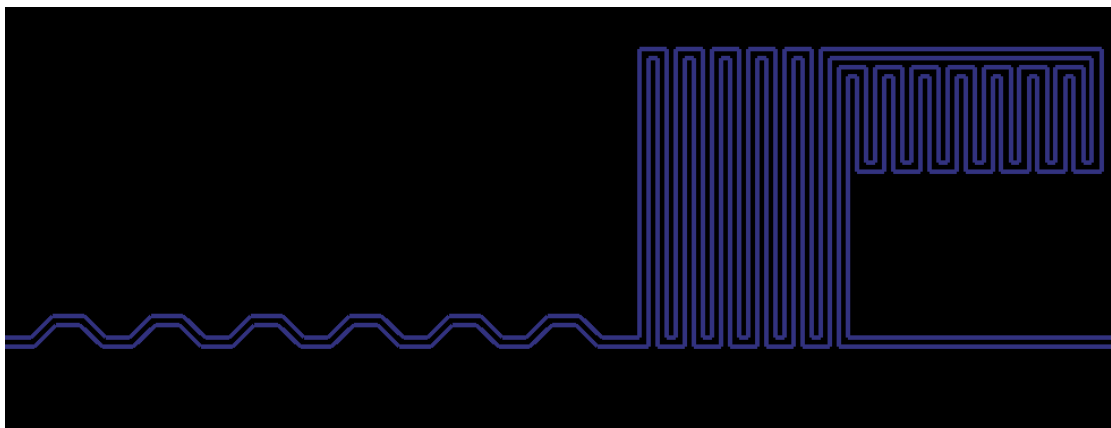


Figure showing the added "amplitude" by selecting the delay tune type then select the track and move the mouse upwards.



Delay Tune sawtooth, accordion and trombone types.

Users of Allegro PCB Designer + High Speed option also have access to Timing Vision, AiDT (Auto Interactive Delay Tuning) and AiPT (Auto Interactive Phase Tuning) which will automatically add the required length to the matched group or differential pair. These commands are available under the Route menu. Timing Vision physically changes the colour of the clines to display whether a matched group or differential pair is meeting the timing constraints or not.

The following are trademarks or registered trademarks of Cadence Design Systems, Inc. 555 River Oaks Parkway, San Jose, CA 95134
Allegro®, Cadence®, Cadence logo™, Concept®, NC-Verilog®, OrCAD®, PSpice®, SPECCTRA®, Verilog®

Other Trademarks

All other trademarks are the exclusive property of their prospective owners.

NOTICE OF DISCLAIMER: Parallel Systems is providing this design, code, or information "as is." By providing the design, code, or information as one possible implementation of this feature, application, or standard, Parallel Systems makes no representation that this implementation is free from any claims of infringement. You are responsible for obtaining any rights you may require for your implementation. Parallel Systems expressly disclaims any warranty whatsoever with respect to the adequacy of the implementation, including but not limited to any warranties or representations that this implementation is free from claims of infringement and any implied warranties of merchantability or fitness for a particular purpose.