Two of the core challenges with routing a high speed design are controlling the impedance of the routes, and matching the lengths of critical nets. Impedance controlled routing ensures that the signal that leaves an output pin is correctly received by the target input pins. Matching the route lengths ensures that timing-critical signals arrive at their target pins at the same time.

Complimenting the controlled impedance routing capabilities, the 6.7 release of Altium Designer saw the introduction of an interactive net length tuning feature. See reference article, http://www.altium.com/files/AltiumDesigner6/LearningGuides/AP0107%20Impedance-Controlled%20Routing.pdf

Interactive length tuning provides a dynamic method of optimizing and controlling the route length of a net by inserting variable amplitude patterns, or ‘accordion’ segments, in accordance with the available space, rules, and obstacles in your design.

**Tuning the length of a net**

The real elegance of Altium Designer's length tuning feature is that it cleverly combines sophisticated software algorithms with intuitive user control. Length tuning segments are added by simply wiping the cursor along the route path, with the dimensions and positions of the various tracks and arcs that make up the tuning segments automatically calculated and inserted by Altium Designer. Keyboard shortcuts give complete control over the style and properties of the tuning segments as they are being added.

Launched from the Tools menu, the Interactive Length Tuning command prompts you to select a route. After clicking on a net (or any free line or track for that matter), you simply slide, or wipe the cursor along the path of the route. No need to worry if you drift off course either, as soon as you bring the cursor back over the route Altium Designer will add tuning segments up to that point.

![Interactive Length Tuning](image-url)

Tuning segments are automatically added as the cursor moves along the route path.
**Controlling the tuning style and size**

There are 2 key ingredients to mastering the length tuning tool: knowing the shortcuts, and understanding the Net Length Indicator display. Let’s start with the shortcuts.

So you’ve launched the command and clicked on a net, and are proudly showing your length tuning skills to your coworkers. Try these shortcuts as you slide the cursor along the route, and you’ll be sure to impress:

<table>
<thead>
<tr>
<th>Shortcut</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Spacebar</td>
<td>Cycle through the 3 tuning patterns: rounded, mitered with arcs, and mitered with lines,</td>
</tr>
<tr>
<td>, (comma)</td>
<td>Decrease tuning amplitude</td>
</tr>
<tr>
<td>. (full stop)</td>
<td>Increase tuning amplitude</td>
</tr>
<tr>
<td>3</td>
<td>Decrease tuning pitch</td>
</tr>
<tr>
<td>4</td>
<td>Increase tuning pitch</td>
</tr>
<tr>
<td>1</td>
<td>Decrease corner miter</td>
</tr>
<tr>
<td>2</td>
<td>Increase corner miter</td>
</tr>
<tr>
<td>Y</td>
<td>Toggle starting direction</td>
</tr>
<tr>
<td>Tab</td>
<td>Open the Interactive Length Tuning dialog.</td>
</tr>
</tbody>
</table>

And if you have trouble remembering shortcuts, don’t worry. You only actually need to remember 1 shortcut, Shift+F1. This is the shortcut for shortcuts – use it during any Altium Designer interactive command to see a list of shortcuts for that command. That’s right, Shift+F1 during component placement, during interactive routing or during track dragging, to get a list of shortcuts for that command.

3 tuning patterns are available, press Shift+F1 for a list of shortcuts that control the tuning pattern properties.
Controlling the target length

The length tuning offers 3 approaches for specifying the target length. The length can be:

1. Manually defined
2. Based on a net that has already been routed
3. Defined by design rules

To choose which of these methods is being used, press Tab during length tuning to open the Interactive Tuning Length dialog. The bottom of the dialog includes options that define the shape and dimensions of the tuning pattern, you already know how to control most of these with shortcuts. Options for selecting from the three Target Length modes are at the top of the dialog.

**Manual:** The manual mode is exactly that, you enter the length in the Target Length field. The list on the right keeps track of the values you have typed in, in case you want to use one again.

**From Net:** After selecting From Net you can choose a net from the list on the right. The length of this net will become the target, but it will be overridden if there are more restrictive design rules defined — we'll look more closely at the rules in a minute.

**From Rules:** You need to have one or both of the Length and Matched Length design rules defined to use this mode. Altium Designer will then obey the most stringent combination of these rules.

Once you've selected the mode, check out the description region of the dialog, the description is invaluable for understanding how the length tuning tool is currently configured.

### Target Length

<table>
<thead>
<tr>
<th>Name</th>
<th>Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length rules</td>
<td>40mm - 56mm</td>
</tr>
<tr>
<td>Matched Net Length rules</td>
<td>52.106mm - 53.106mm</td>
</tr>
</tbody>
</table>

**Target Length** 54.053mm

Target length set to the middle of the valid range. Valid range constrained by current rules is between 52.106mm and 53.106mm.

The target length being defined by the rules. Note that the most stringent combination of the rules is used, so the range is 52.106 to 56, and the target length is set to half way between these values.

**Configuring the design rules**

Ok, time to look at the design rules. I'll start with the Matched Length design rule.

The Matched Net Length design rule specifies that the target nets must be routed to the same length, within the specified tolerance. Older versions of Altium Designer included other Constraint settings for this rule, these are not used by the length tuning tool, so ignore them.

And what are the target nets, well that depends on the rule scope, or query. As you know, the object that each rule targets is determined by the scope, or query. A typical query for this type of rule would be `InNetClass('MyEqualLengthNets')`, which means that the rule applies to all the nets in the net class `MyEqualLengthNets`. Select Design » Classes to set up a net class.

The length tuning tool will then find the longest net in the set of target nets and give you a valid range of that length, plus or minus the Tolerance specified in the rule.

Complementing the Matched Length rule, the Length rule specifies the overall routed length of a net, or set of nets. Either or both of these rules may be important in your design, it all depends if your potential issues are related to skew (signals arriving at different times – think Matched Length rule), or the overall signal delay (think Length rule).
Both the Matched Length rule and the Length rule are specified as distances, whereas your design-mind will be thinking in terms of time – how much later can this signal arrive after that one? The time it takes for a signal to propagate across a printed circuit board depends on a number of factors, including the board materials and the layer (surface or internal route). For traditional FR-4, the delay will be in the order of 4 to 6 picoseconds per millimeter. Now that might seem like a pretty small delay, but the problem is more complex than simply considering the timing conditions of an individual routing connection. You’ll be factoring these delays into your overall timing scheme so you can correctly budget for them in the layout.

As mentioned earlier, the length tuning tool considers both of these rules and works out the tightest set of constraints. So if the maximum length specified by the Length rule is shorter than the longest length targeted by the Match Length rule (plus the tolerance), then the Length rule wins and its length is used during tuning. If this is starting to sound like gibberish then just keep an eye on the description region of the Net Length Tuning dialog, it will let you know if there are no valid rules, or which numbers it is using from the rules.

**Using the Net Length Indicator**

Whenever you are adding tuning segments with the length tuning tool, the Net Length Indicator displays. The Net Length Indicator provides a visual gauge of how closely the current route length matches the required length.

The indicator consists of a white box, 2 vertical yellow bars, and a vertical green bar. Within the white box a colored slider will appear, indicating the current length of the net, with the exact length displayed in text on top of the slider.

The white box indicates the total range of possible lengths, the meaning of its upper and lower limits depends on the target length mode you have chosen.

If the target length mode is manual, the lower limit of the slider box will be the length of the current net, and the upper limit will be the length of the current net plus twice the difference between the current net length and the target length.

If the target length is based on design rules, the lower and upper limits of the slider box are taken from the Length rule. If there is no Length rule then the slider box is scaled from the length of the current net, the same as it is in manual length mode.

The green bar in the Net Length Indicator marks the target length, and the yellow bars indicate the minimum and maximum lengths allowed. The minimum and maximum are determined from the tightest set of constraints defined by the design rules.

![Net Length Indicator example](image)

**Net Length Indicator example**

- Indicator minimum is 40 (from Length rule)
- Indicator maximum is 56 (from Matched Net Length rule)
- Target minimum is 55.106 - 3 = 52.106 (length of longest net in set of target nets, minus the tolerance in Matched Net Length rule).
- Target maximum is 56 (from Length rule, since this is less than length of longest net plus the tolerance in Matched Net Length rule)
- Target length is mid way between target minimum and target maximum (56 – 52.106) / 2 + 52.106 = 54.053
What else do I need to know?

Board design is an iterative process, and generally involves plenty of pushing and shoving to arrive at the final placement and routing. So what do you do when you decide that you need to relocate tuning segments?

The easiest way to remove tuning segments is to simply route over the top of them. As with any existing routing, select Interactive Routing from the Place menu, then click and place a straight routing segment straight across the top of the tuning segments. As soon as you have completed the route and right-clicked to escape, all the tuning segments will be removed.

And are there downsides to using accordion-style tuning segments? Of course, it is just another technique used in the overall process of good design. If the adjacent accordion sections are too close together for too long, then crosstalk coupling can distort the signal. For more information read this interesting article on Serpentine (accordion) Delays by an industry expert, Dr Howard Johnson [http://www.signalintegrity.com/Pubs/edn/serpentine.htm].

Other useful links

TRAININGcenter video on using the Length Tuning feature (search for length tuning)
http://www.altium.com/Community/TRAININGcenter/TrainingVideos/

Website for Eric Bogatin, signal integrity lecturer and industry expert
http://www.bethesignal.com/

Website for Dr Howard Johnson, high speed design lecturer and industry expert
http://www.signalintegrity.com/

Website for Lee Ritchey, lecturer and high speed PCB design expert
http://www.speedingedge.com/

Article on the value of signal integrity simulation, by Fred Saal, Quad Design Technology

Complimenting the controlled impedance routing capabilities, reference to article,