PCB Tools Breadth and Depth
Part 1 – Interactive Routing
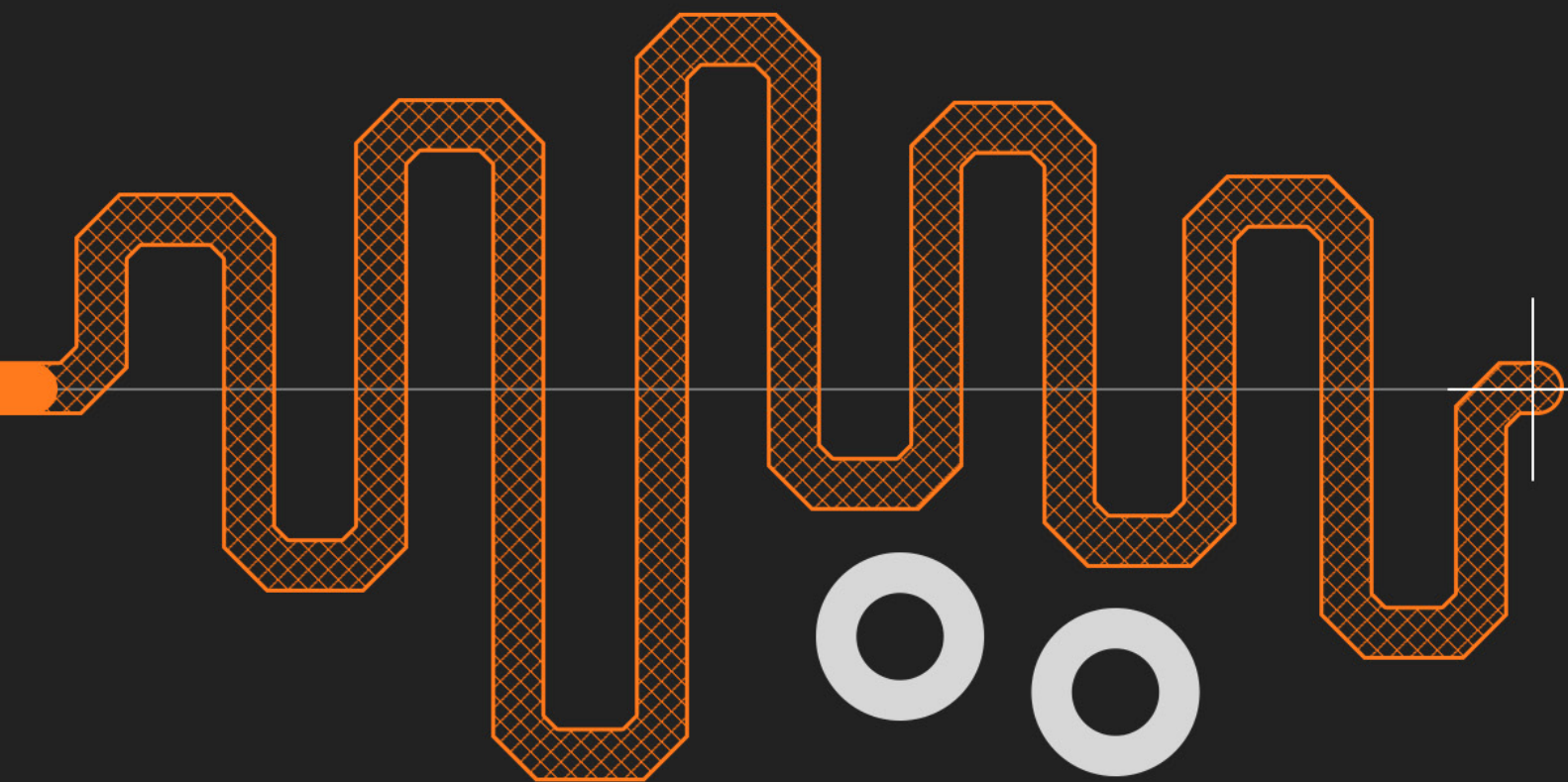
Charles Pfeil

Product Manager

INTRODUCTION

The breadth of our PCB design tools is about having the capability to address a broad set of technology challenges. The depth of functionality is a matter of providing the engineer with options to resolve distinct variations on the original task. An efficient layout product has not only the broad support of component, signal and fabrication technology, but also has deep functionality to fulfill the many variations in design requirements productively.

INTERACTIVE ROUTING

For most designs, the routing process remains the most time-consuming layout task. As the technology for all aspects of PCB design has evolved over many years, the routing software has to advance to support those requirements.

The Altium Designer interactive routing environment provides the ability to route boards consisting of laminate, HDI, flex and rigid-flex constructions. Let's now explore a few capabilities that illustrate the depth of functionality available.

DYNAMIC CLEARANCE BOUNDARIES

When routing through congested areas, it is often difficult to find out if there is sufficient space for the routing. The clearance boundary feature not only shows the available channels, but it also puts a web in the spaces where the trace will not fit. Because of this web, you do not need to attempt routing through each gap to try and figure out if the trace will fit. Rather, you can look ahead and quickly determine the channel to use for routing.
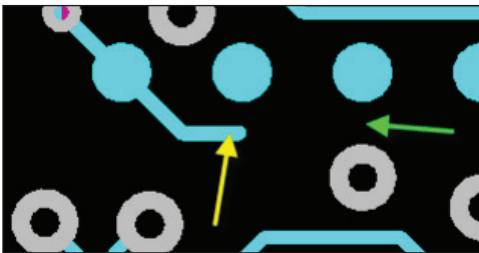
Starting Point - In the image below, you can see when routing the trace (yellow arrow) it seems that it would fit between the pad and via (green arrow).

Start Routing - When the routing starts, the dynamic clearance boundary is shown, and you can see the web that is created (green arrow) which displays the trace will not fit. Whenever there is a web (large or small), it simply means the current trace will not fit through the gap.

Smaller Width - There are options to use a width of any pre-defined proper width or any width you would like. Here the trace has been changed to a smaller width, and you can see the clearance web is gone, which means there is room to route through that gap.

Of course, you may not want to change the width, this is just to illustrate that the clearance boundaries dynamically change based on the width being used. As a side note, the purple arrow (above) shows a via with the hole having two colors. This is a buried via, and the colors indicate the start and end layers of the via span.

Altium

PIN SWAPPING WHILE ROUTING

While routing, one of the most time-consuming tasks is to route from one BGA to another and to deal with the connection crossovers that the pin ordering dictates. With Altium Designer, the pins can be swapped as part of the routing task with the Subnet Swapping function.

The technique is to route the traces out from the BGAs in an orderly fashion from both directions, then once you see how the connections are tangled, use the swapping function to put them in order and route them together. This can be more effective than swapping pins before routing because the optimal order isn't necessarily the one with no connection crossovers.
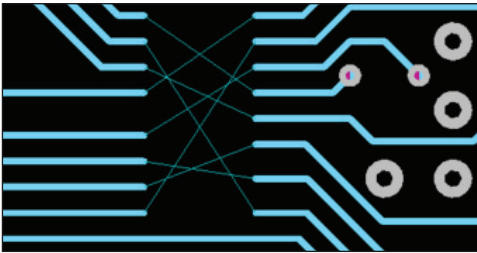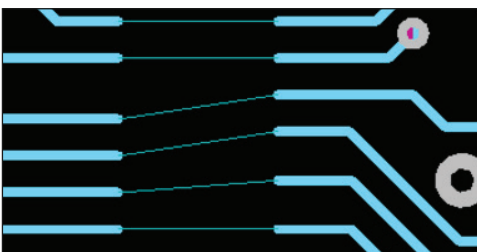
Initial Routing - Route out from both ends of the connection.
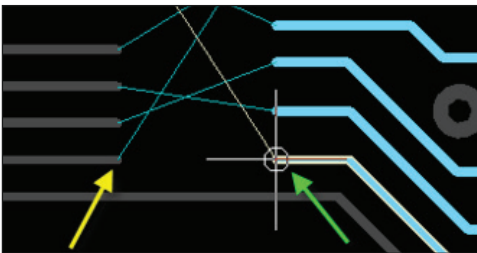
Start Swapping - Click on the trace end (green arrow) and the other trace end that you want to use for that net (yellow arrow).
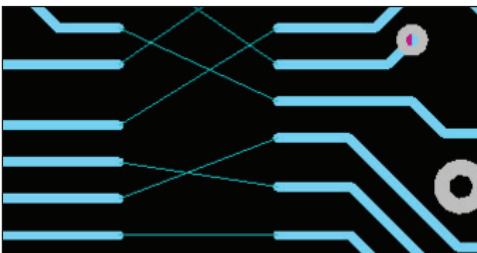
Result after the Second Click - When you click on the other trace end, the connections automatically swap.

Final Result - Here is the outcome of the subnet swapping. Just route the traces together and you have optimized the exits from the BGA, making the overall routing task much easier
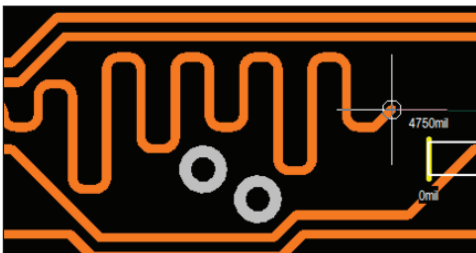
TUNE ACCORDIONS

You can add tune accordions while routing. One of the important things is that the tuning fills the available area automatically as you move your cursor through that area.
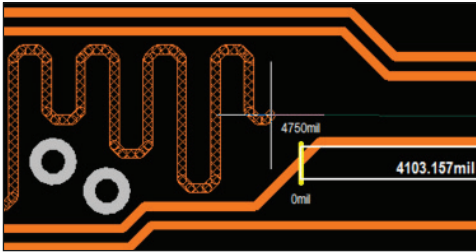
Adding Tuning - This tuning is being added with 45-degree miter corners. The Length Gauge shows that the target length is 4750 mil, and the current length is 4103.157 mil.

Tune with Arcs - You can control the radius of the arc or even have a semi-circle pattern. This image shows an arc replacing the 45-degree miter corner.

Help	F1
End Accordion	Shift+A
Remove Last Segment(s)	BkSp
Terminate Current Trace	Esc
Toggle Routing Mode	Shift+R
Choose Favorite Width	Shift+W
Toggle Length Gauge	Shift+G
Toggle Push Mode	Shift+P
Switch Layer For Current Trace	Shift+S
Place Segment	Enter
Toggle Elbow Side	Space
Cycle Placement Mode	Shift+Space
Edit Tuning Pattern Settings	Tab
Decrease Amplitude by Increment	,
Increase Amplitude by Increment	.
Decrease Miter Radius	1
Increase Miter Radius	2
Decrease Gap by Increment	3
Increase Gap by Increment	4
Toggle Auto Complete	5
Next Tuning pattern	P
Toggle Amplitude Direction	Y

Additional Options - This pop-menu illustrates the depth of options when accordions are being added. Everything can be adjusted either by choosing it in the pop-menu or using a shortcut key.

CONCLUSION

With the routing process being one of the most time-consuming board layout tasks, designers need as many tools as possible for today's complex design requirements. This paper focused on three of those tools found in Altium Designer - Dynamic Clearance Boundaries, Pin Swapping while Routing, and Tune Accordions. These tools will provide additional efficiency for any PCB designer needing to design highly complex board layouts within strict deadlines and standards.

Altium