Nortel Networks PCB *Channel Routing*

PCB *Channel Routing*

PCB (Printed Circuit Board) layout has become very challenging as Ball Grid Array (BGA) package densities and pin counts have increased the need to reduce line widths and spacing in order to "fan out" or escape BGAs, thereby impacting PCB layer count and manufacturability. The increasing number of layers in PCB designs drives up cost and complexity. Nortel Networks PCB *Channel Routing* technology provides a means to help reduce BGA layout time, PCB layer count, and PCB product cost. Nortel Networks has *Channel Routing* software available for license that works with a CAD tool to enable PCB designers to efficiently escape BGAs using *Channel Routing*. Software right to-use and PCB right-to-manufacture licenses for *Channel Routing* are available from Nortel Networks. U.S. patents have been granted on certain aspects of *Channel Routing*. Applications for patents on additional aspects of *Channel Routing* are pending in the U.S. and other countries.

| The Problem	The *Channel Routing* Solution
Silicon packaging pin counts and densities have increased to the point that in most designs the number of layers to break out the largest BGA devices dominates the layer count required to track a board.	1) Opens up more local routing channels for signal escape with fewer routing layers within a package.
When grounds, power, and cross-connect layers are included, layer counts of twenty or more are often required, resulting in high cost and complexity.	2) Opens up more global routing channels for interconnecting devices with fewer routing layers within a PCB.
Increased functionality on a smaller footprint forces designers to a densely packed, double sided surface mount PCB with high-density array packages. This compounds the challenge for layout designers to escape all signals and reduce the number of PCB layers.	3) Enables significant reduction in layer count (in conjunction with component integration and vias). 4) Improves electrical performance due to close placement of discrete devices to driver circuit, resulting in less via parasitic inductance and capacitance.
The area array devices (most commonly BGAs) have not been designed for optimum breakout, but have been driven to maximize footprint density.	5) Reduces restrictions for double sided placement of components.

Channel Routing: An Application Example

With high pin count BGAs, it takes more and more layers to connect the fanned out pins to the rest of the circuit board. In theory assuming the worst-case scenario where 100% of pins need to be routed out of the package, a 1600 pin (40x40) BGA has 156 possible routes per layer (one trace between vias). This means that nine (9) layers would be required simply to connect the BGA to the rest of the circuit. High layer counts, via counts, and high via aspect ratios make PCB fabrication more difficult and more expensive.

If more routing channels are created, a PCB designer can connect more traces per layer and reduce the total layer count. *Channel Routing* uses micro or blind vias and pin sharing to form additional channels in the BGA fan out pattern. The new channels enable 40 extra connections per layer. In addition, if via sharing on power pins is used and there is no fan out of unconnected pins, the BGA becomes more open and easier to route.

Up to four (4) routing layers can be eliminated (eight total layers), in this example, helping to significantly lower the cost of the PCB and improve manufacturability.
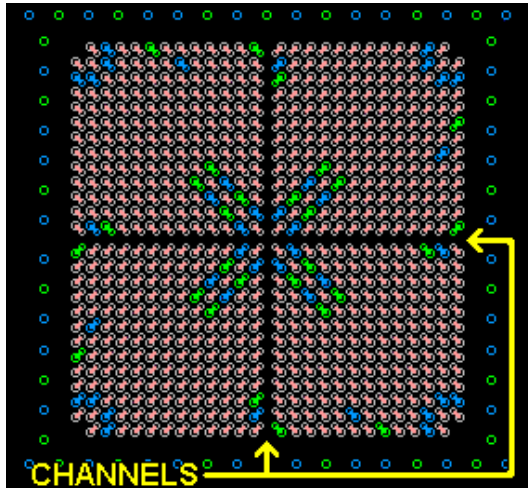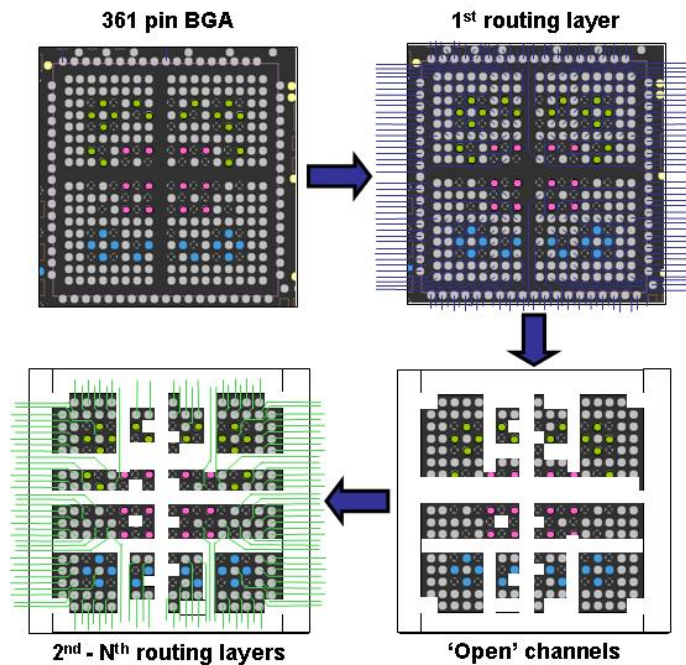

Top Layer ---Without Channel Routing
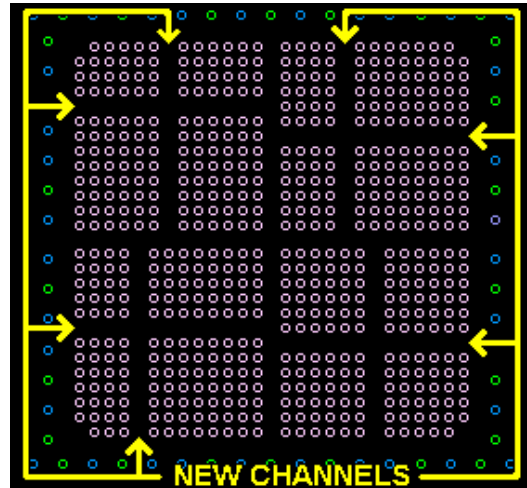
Inner Layer --- With Channel Routing

Channel Routing: Conceptual Flowchart

"Using Nortel Networks patented Channel Routing technology, complex PCBs can be designed with fewer layers, thereby enabling even higher density products with improved manufacturing yields and overall, a more cost effective PCB," said Ryan Stark, Vice President, Advanced Technology.

"We believe Channel Routing can benefit those companies that are pushing the envelope in complex PCB design."

Channel Routing: Benefits

Cost reduction	* Fewer PCB layers.
	* Creates space for component placement on secondary side of PCB.
Efficiency	* Reduces PCB design time. Automates BGA fan out with Channel Routing software
	* Improved Time-to-Market.
	* Faster order fulfillment.
Easy to use	* PCB designers use the same software interface and design rules that they use today. Channel Routing software "plugs into" an existing CAD tool.
Manufacturability	* Simplifies manufacturing. Improves yields.
	* Enables complex PCBs to be designed and manufactured – may mean the difference between success and failure for complex PCBs.
Performance	* Improves electrical performance.
	* Reduces via parasitic inductance and capacitance.
	* Potentially smaller foot print and shorter tracks.
Leading Edge Technology	* No capital investment.
	* Enables combination of multiple components into a single PCB.
	* Enables product miniaturization.
	* Facilitates trends to higher BGA pin counts and more complex PCBs.
Benefits Summary	* Reduces costs, improves yields, increases productivity, and enables product design breakthroughs.

When To Use *Channel Routing*

1. When the layer count required to fan out a board is dominated by the number of layers to break out the high pin count (e.g. BGA) devices, *Channel Routing* opens up local routing channels for signal escape resulting in fewer routing layers within a package.

2. When routing difficulty arises from a high concentration area, *Channel Routing* opens up more global routing channels for interconnecting devices with fewer routing layers.

3. When thickness and real estate area requirements are impossible to meet with conventional technology, *Channel Routing*, combined with component integration and micro or blind vias, can enable significant reduction in layer count and board size.

4. When high-speed design requires close coupling of pins, *Channel Routing* can improve electrical performance due to close placement of discretes to driver circuit and less via parasitic inductance and capacitance.

5. When high-density requires double-sided surface mounting of component, *Channel Routing* can help reduce restrictions for double-sided placement of components.

6. When high-speed edge coupled differential pair routing is required for fine pitch device (only one line per channel with conventional techniques), *Channel Routing* enables edge coupled routing plus larger separation between signal pairs and pin field.

7. When optimum pin assignment of devices such as FPGAs is required, *Channel Routing* enables pre-planning pin location.

8. When ECAD auto routing difficulty arises from high-density small pitch array devices, *Channel Routing* helps to route device pins to the peripheral.

Channel Routing Software

The *Channel Routing* software is used with an existing CAD tool to define the BGA pins. The *Channel Routing* software then selects sequential sets of pads to be routed or escaped on the first routing layer closest to the device with micro or blind via technology. The software then removes the non-functional pads from all layers below the first and creates "channels" or pathways for the designer to now "fan out" a greater number of traces from the BGA, either automatically or manually, in each subsequent layer. This enables the designer to reduce the total number of layers in the printed circuit board. The *Channel Routing* software is available under license from Nortel Networks.

The **Channel Routing** software provides the PCB designer with multiple design options for use on large BGAs. The extent to which it is used will depend on design requirements. Designers may choose to use all, or just some of the features, to facilitate the best routing result for the PCB design.

The four steps to using the *Channel Routing* software are:

1. Add the "Dogbone" fan outs and form the channels
2. Choose the logic and power pin sharing parameters
3. Add internal fan outs to the micro or blind-via layer
4. Route all remaining signals on selected layers

Note: The software may be stopped at any point, enabling the designer to choose the extent to which *Channel Routing* is used to achieve the best overall PCB design.
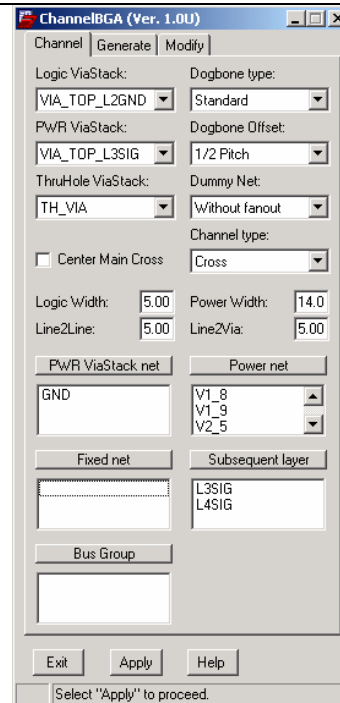
For further information on *Channel Routing* technology and/or licensing information for PCB designers or PCB manufacturers, please contact:

CopperCAD®
CADWare
www.CopperCAD.com

For Sales and Support contact:
CopperCAD Sales
Tel 905-488-8958
Fax 905-488-8955
Email: CR@coppercad.com
        www.CopperCAD.com

Nortel
IPR & Technology Licensing
Tel   905-863-1319
Fax   905-863-8431
Email: techlicensing@nortel.com
www.nortel.com/corporate/technology/tlp/