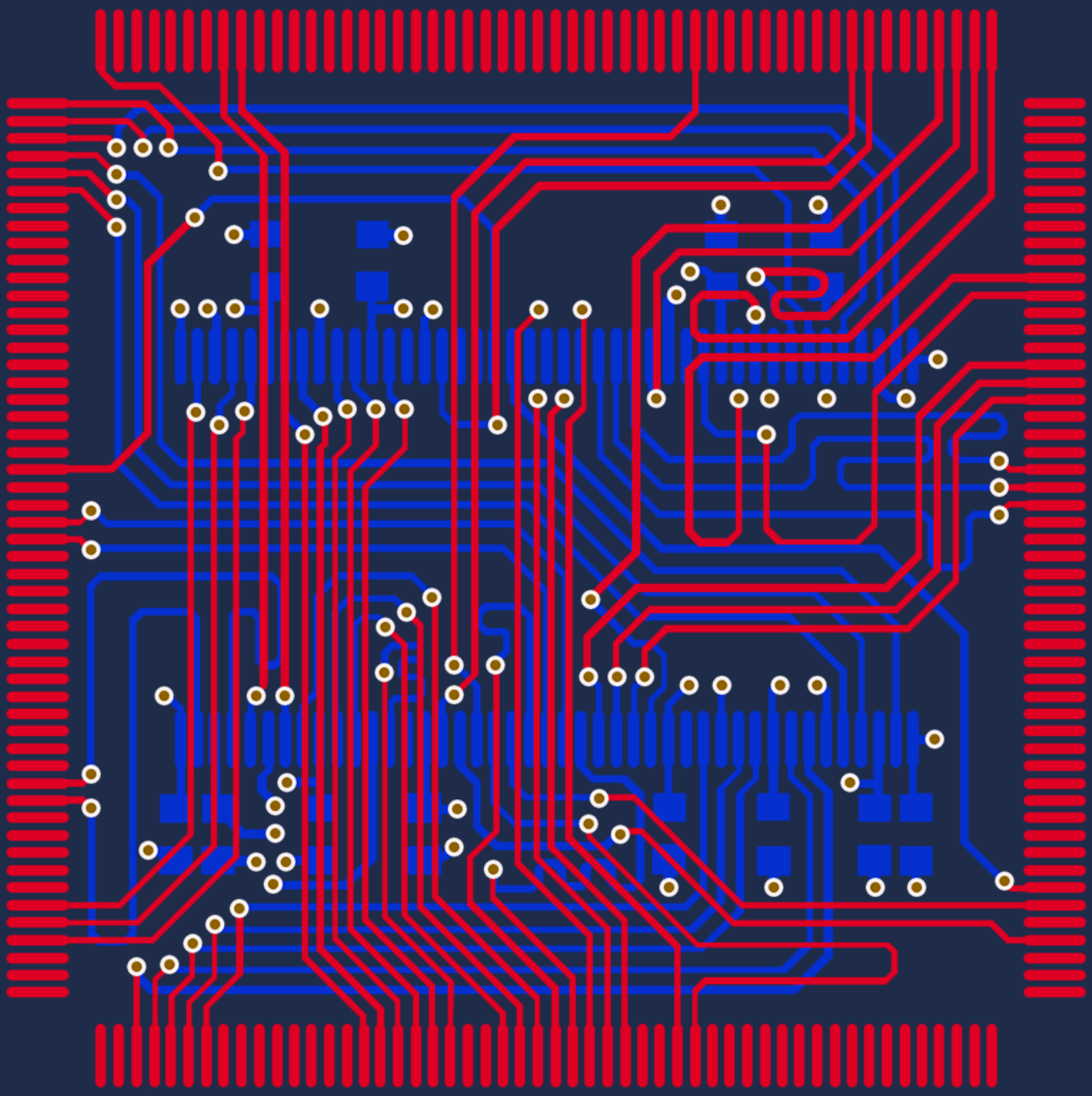
Altium®

BGA Fanout Routing



Christian Keller

Field Applications Engineer

BGA FANOUT ROUTING

OVERVIEW

PCB developers are deluged with new challenges caused by increasing density and smaller components. Ball Grid Arrays (BGAs) create particular challenges during layout with hundreds of connections in just a few square centimeters.

FANOUT- AND ESCAPE-ROUTING

Because of the density and distance from the connection points, only the two outermost rows of a BGA can be connected directly to surface circuit traces. All other terminals of the BGA cannot be connected in a direct path on the surface. Fanout and escape routing is integrated in many PCB design systems to enable further connections. In fanout and escape routing, the two outermost rows, and all other rows of a BGA, are automatically connected to the center of the terminals via a short circuit trace that is executed at a 45 degree angle. This provides a blind-via that forms a direct connection to the next signal layer. Routing can be executed on the next signal layer.

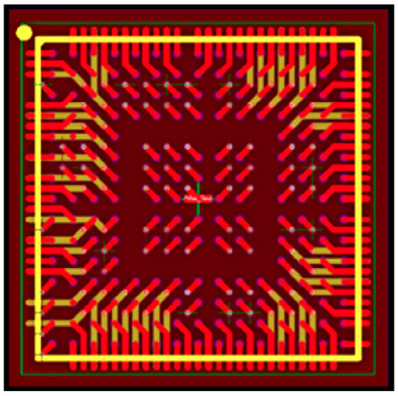


Image 1. BGA with classic fanout routing of all the electrical connections.

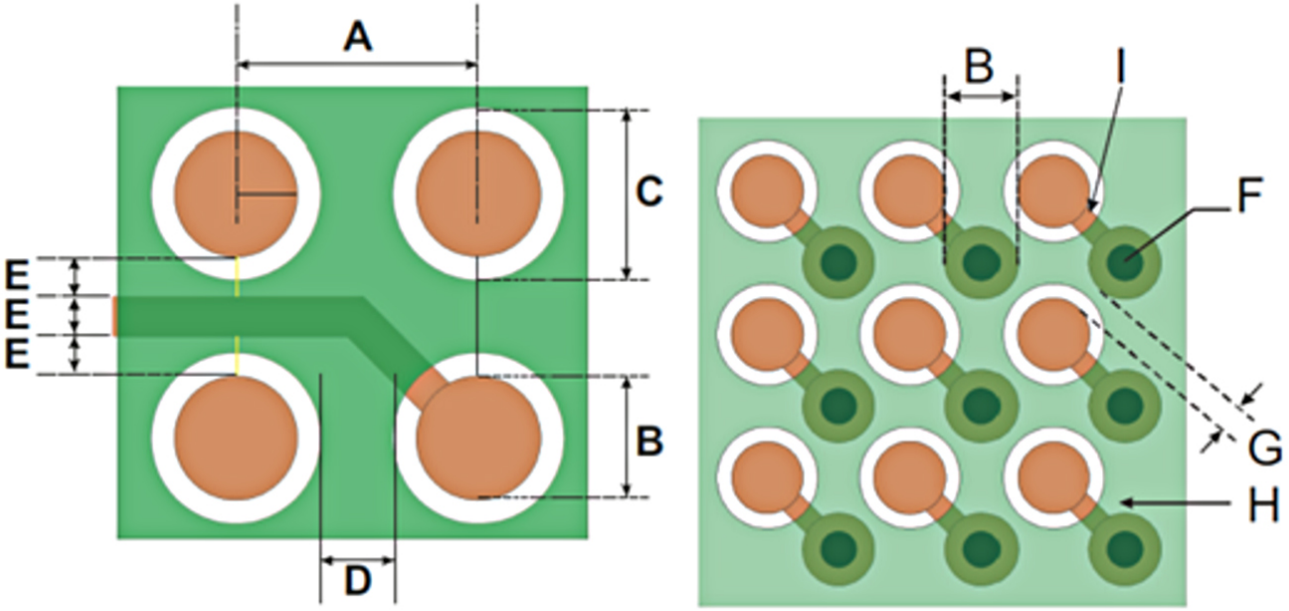


Image 2: Visual representation of terminals of a BGA with their connection possibilities.

Using via-in-pad eliminates the need for the additional trace to the center of the connections, thereby creating additional space for circuit traces. Therefore, with via-in-pad, the through contact can be placed directly at the terminal of the BGA.

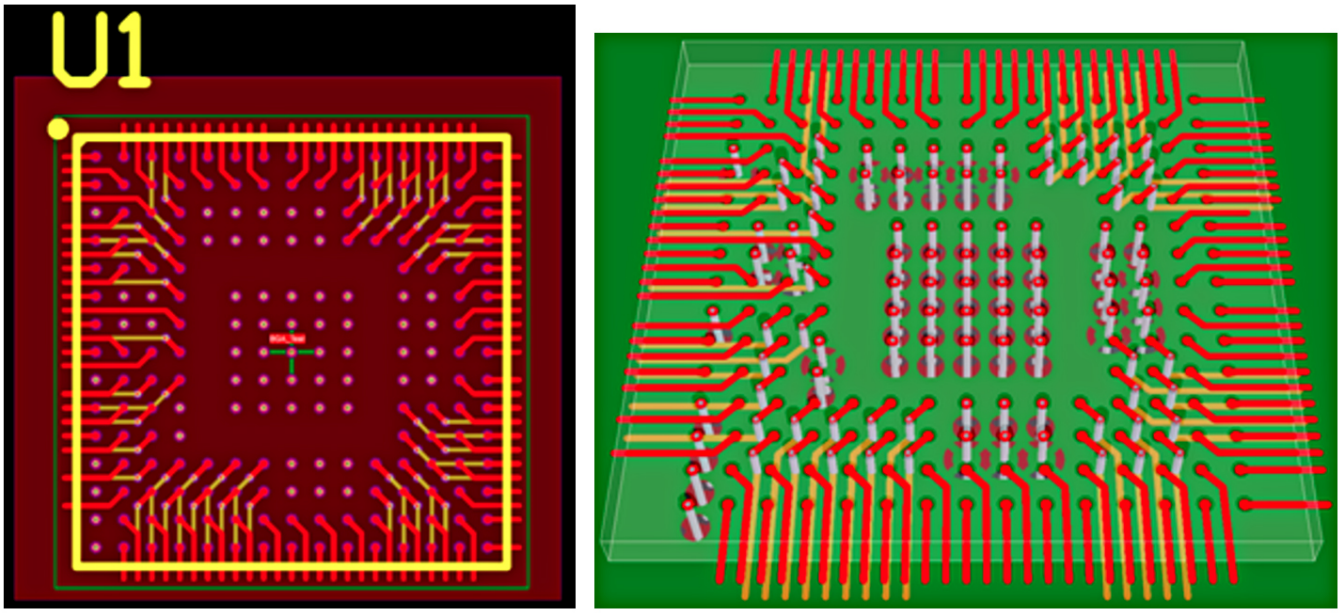


Image 3. Signal lead with fanout and escape routing, and a voltage supply connection as with via in pad.

During circuit board manufacturing, these through contacts will be filled with a non-conducting medium and cured. Later, the ends are metallized, planarized, and also over-contacted. This makes the surface of the via flat and can be used the contacts of the BGA. This solution can be used for both stacked and staggered microvias and/or blind vias. The IPC 4761 to connect to describes how via-in-pads, for example filled and capped Vias (IPC 4761 Type VII) are prepared. Despite the higher manufacturing costs, via in pads will always be preferred, because of the higher integration density of BGAs and their lower inductance at high frequencies (signal quality).

Fanout routing in Altium Designer lets you decide between the classical fanout (auto or BGA) and via in pad. For classic fanout routing, Altium Designer offers all the necessary settings for the direction of the fanout (direction from pad) whether the Via should be placed in the center between the terminals of the BGA or not (via placement mode). The via is placed between the terminals of a BGA in most cases, because of costs and ease of manufacturing.

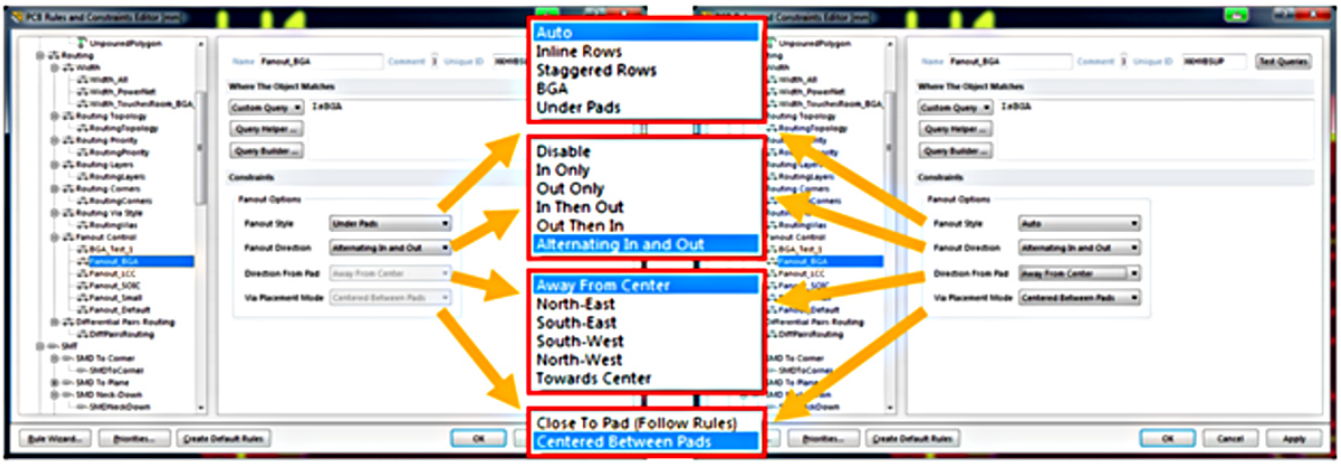


Image 4. Design Rule Editor (Fanout control) in Altium Designer.

In many cases, the via is shifted toward the terminals. Escape routing is optimized with this "off-grid" connection. However, a new strategy for fanout and escape routing must be defined for each BGA. The classical autorouter available in most ECAD systems quickly runs into its limitations.

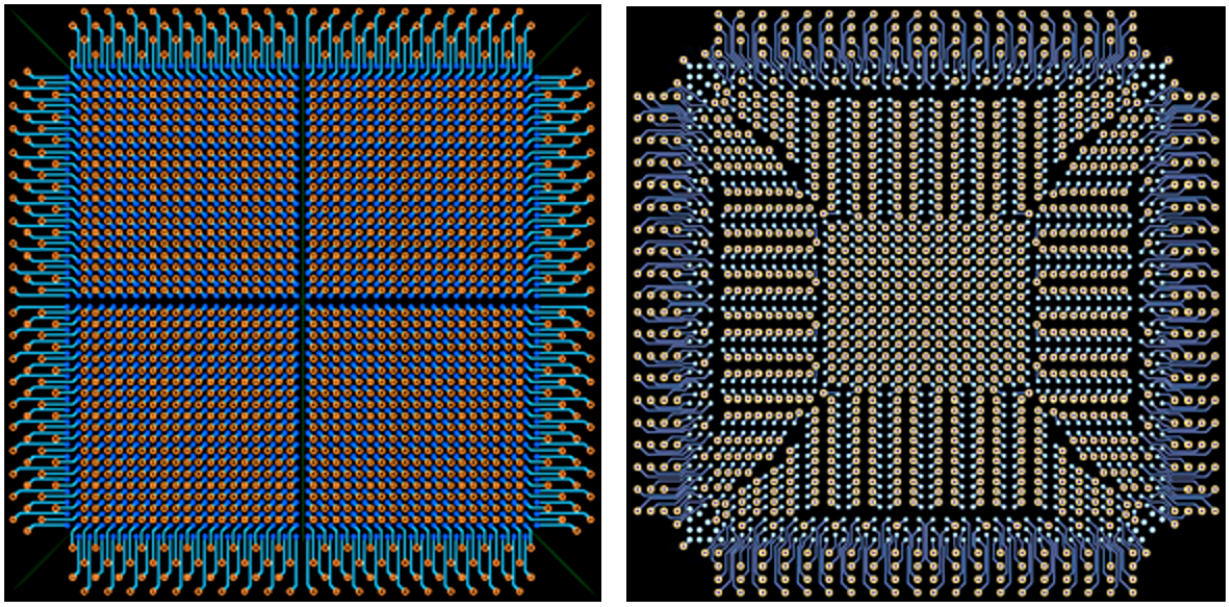


Image 5. Different pin assignments for the fanout and escape routing of BGAs.

PCB manufacturing has improved to accommodate the finer features needed to break out BGAs. Dimensions of trace width and minimum via size have all shrunk. High Density Interconnect (HDI) has also become more and more common for high-speed boards with very dense parts placement.

A pad size of 18 mils (0.45 mm) and a drill diameter of 6 mils (0.15 mm) are considered as a standard for through-hole connections for 1 oz. copper. Contact your PCB fabricator to determine their exact minimums for these and other manufacturing-critical dimensions.

When working with BGA breakouts, narrower traces can be used near the device to facilitate getting the signals out from the package. However, it's virtually impossible to maintain target impedance with traces less than 4 mils on common substrates. Because of that, these traces should only be used near the device for breakout and not generally across the entire PCB.

SUMMARY

PCB designers have become used to the ever shrinking boards and components, and will continue for the foreseeable future. New technologies, such as via-in-pad, smaller traces near BGAs, and design rules incorporated into Altium Designer give designers the tools to layout these signal-dense devices. In addition, ActiveRoute® provides the flexibility to let the tool route large breakouts, completely under the control of the designer. Using these tools can make breaking out and routing BGAs as painless as is possible.