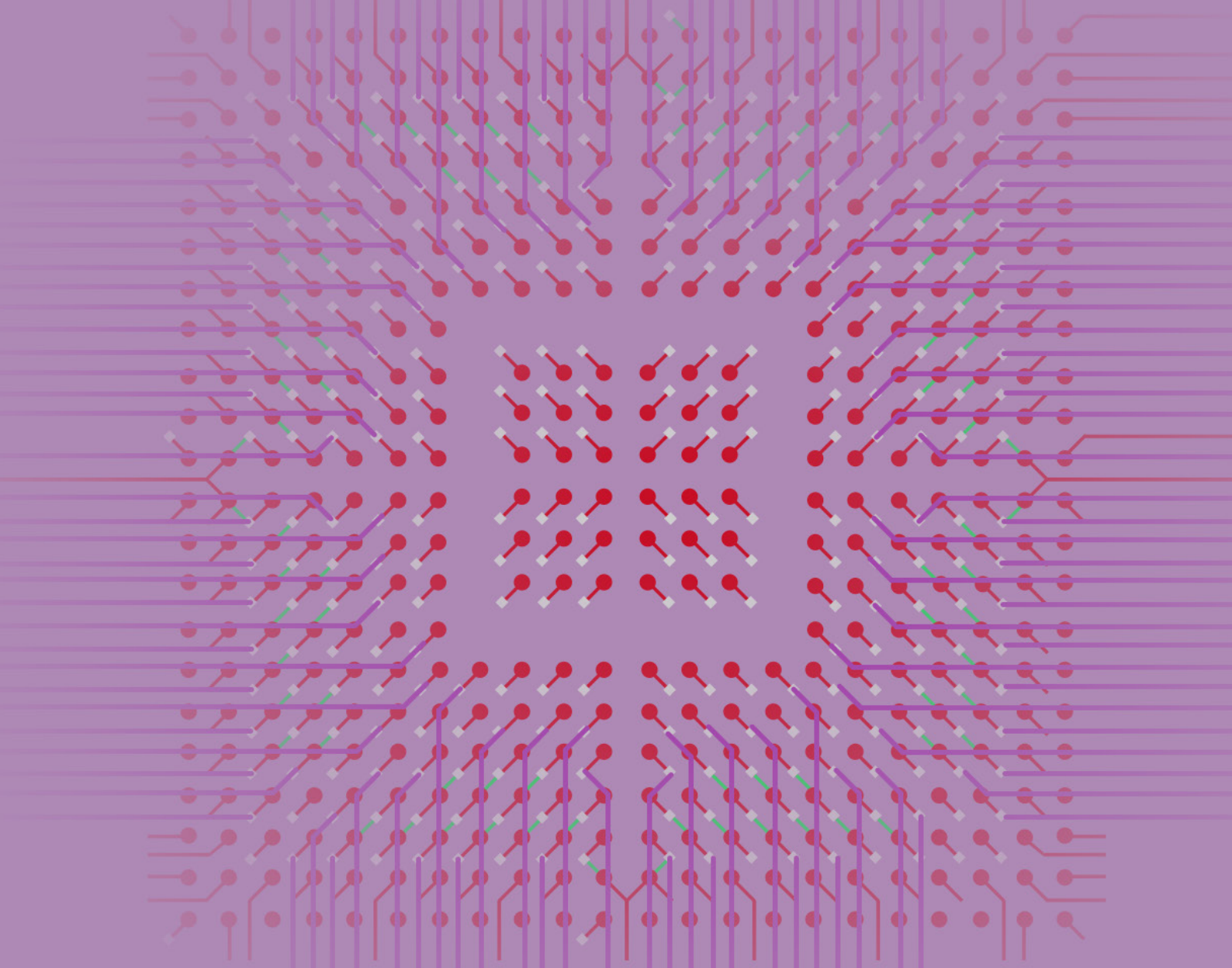
Altium

Area Sensitive Part of the Board



Charley Yap
Field Applications Engineer

AREA SENSITIVE PART OF THE BOARD

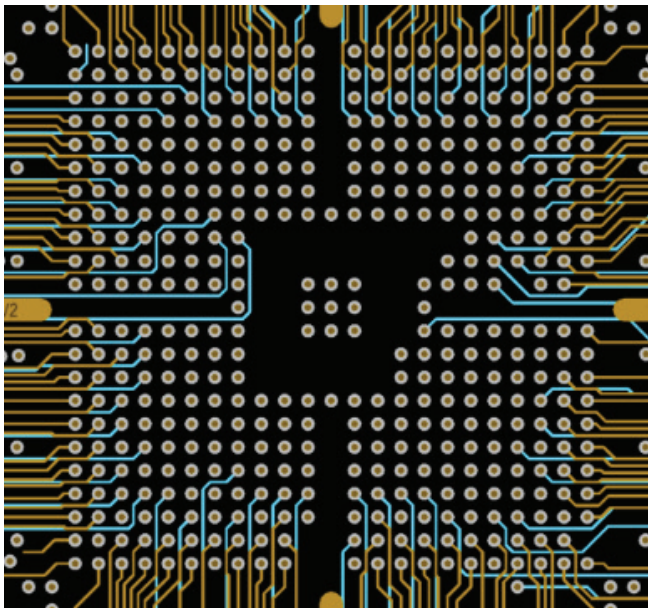
INTRODUCTION

Currently, the standard for housing a variety of advanced and multifaceted semiconductor devices like FPGAs and microprocessors is done through Ball Grid Array (BGA) device packaging. BGA packaging for embedded design has evolved significantly over the years, to keep up with the technological progression of chip manufacturers. This particular type of packaging can be broken down into standard BGAs and micro BGAs. With today's technology, the demand for I/O availability poses a number of challenges, even for experienced PCB designers, due to multiple exit routes.

The primary challenge to designers is developing suitable exit routes that won't cause fabrication failures or other issues. There are several applications that you need to ensure proper fan-out routing strategy, including pad and via size, number of I/O pins, number of layers required to fan out the BGA, and trace width spacing. There's also the question of how many layers a board should have, which is never an easy decision for a designer. More layers mean higher the overall cost of the product. On the other hand, sometimes you need more layers to suppress the amount of noise that the design could encounter.

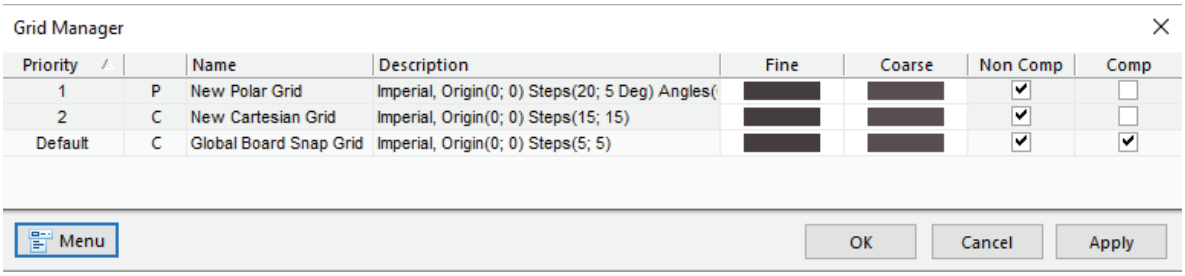
THE WORK AREA

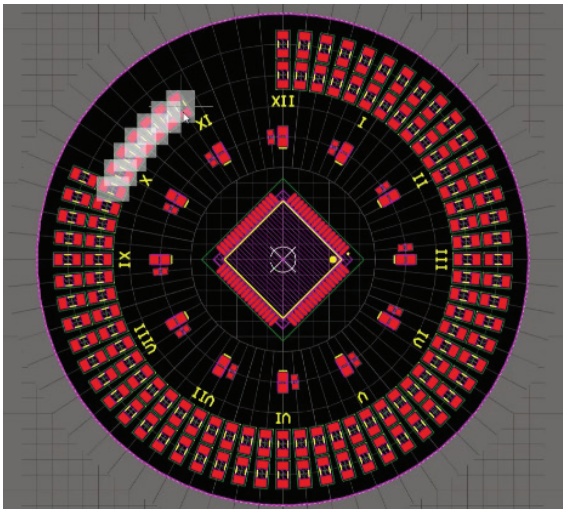
Once a designer has determined the trace and space width of the design, the size of the via, and the amount of traces in a single channel, they can then ascertain the number of layers that they need. The best practice is to minimize the usage of I/O pins, to have fewer layers. Commonly, the first two outer sides of the device do not require vias, while the inner section requires vias to be routed underneath it. Many designers call this the dog-bone. It is a short trace from the pad of the BGA, with a via at the other end. The dog-bone fans out, partitioning the device into four sections. This allows the remaining inner pads to be accessed by another layer and provides an escape route out just beyond the edge of the device. The process will continue until all pads completely fanned out.



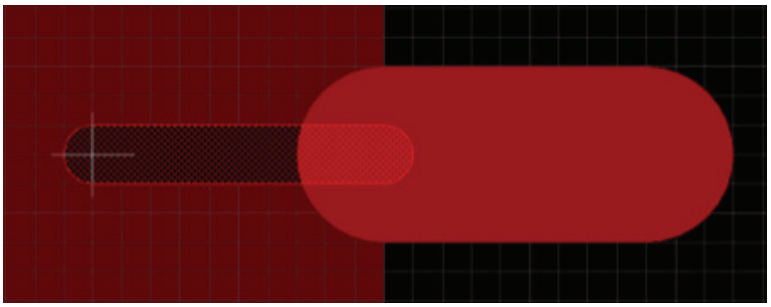
Routing is not always consistent when it comes to snap grids. A good example of this is when a user has to perform a neck-down, from a wide width trace to a narrow width trace. When this happens, it can be tedious for the user to keep changing the settings back and forth to accommodate the proper snap grid. But if we introduce a grid within a grid, where a user can automatically modify the sensitivity of their snap grids to a more comfortable setting, that can ease the burden of repetitive and monotonous processes. Another example is dealing with circular component placement. Polar grids are useful features in an advance snap management system. They can be extremely helpful when designing circular boards, especially during component placement.

AREA SENSITIVE PART OF THE BOARD





Today, routing BGAs is much easier than in previous years, since most software tools offer an auto-fan-out feature. However, there are still some instances where users need to route sections of the device manually. Intelligent track-segment routing was born out of these manually routing cases. Altium Designer provides a smart solution, which allows users to define a section of a board and apply special rules, to accommodate sensitive areas. By implementing certain design rules to these regions, traces can now expand and contract automatically within the area, as the routing enters and exits.



CONCLUSION

Designing a BGA is no easy task. It requires a number of design rule checks (DRC), to ensure proper width spacing for all traces, as well as careful study, to determine how many layers are needed, in order for the design to be successful. As technology continues to grow at a rapid rate, so does the challenge each designer faces, to route their design into very tight spaces. Depending on the size of the BGA, users need all the extra help they can get from the tools at their disposal, to stay on top of things. The Altium Designer room base routing gives its users full control over which constraints will be enforced within the specified area, and provides a variety of automated solutions, such as automatic neck-down, for a smooth routing experience.