

BGA Placement and Routing consideration

Introduction

The purpose of the hints is to help speed up and improve the quality of your PCB design.

General design guidelines

There is a variety of different BGA pitches around today. From the popular 1.27mm through 1.0mm, 0.8mm and down to 0.5mm pitch. The most important task to do before starting the design is to consult with your PCB manufacturer to finalize the best traces/clearances of the traces in order to define if one or two traces will go in between the BGA pads. Deciding upon via sizes and drills is next in line. Most PCB manufacturers can handle 4mil traces with 4mil space.

Since BGA dimensions are in mm it is preferred to design the PCB on mm grid and mm parameters.

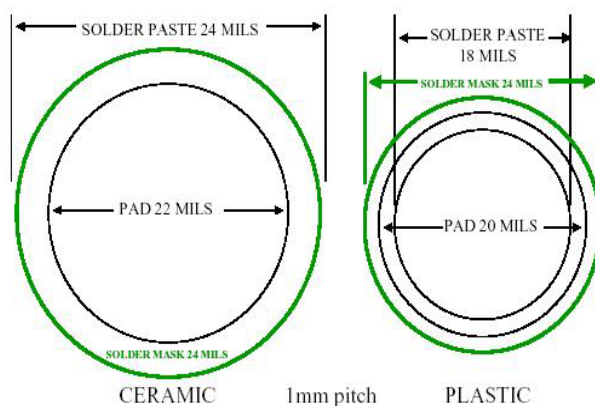
BGA materials

There are two basic BGA materials: **Ceramics** and **Organic** (FR-4).

There is a different set of rules for the design with each substrate. Most common is the FR-4 BGA. It is used in high volume products and in low layer count.

The Ceramic substrate differs from the FR-4 substrate in the TCE, therefore it is used when high temperatures are expected, and is the material of choice for processors. The difference in the PCB layout design will reflect in bigger pads size and in larger solder paste pads in this substrate than in the FR-4 substrate.

Ceramic VS Plastic



PAD sizes in FR-4

Pitch	Pads Size	Trace/Space	Via
1.27mm	25mil	2X 5/5	25mil
1.0mm	19mil	2 X4/4	19mil
0.8mm	16mil	1X5/5	16mil

PAD sizes in Ceramics

Pitch	Pads Size	Trace/Space	Via
1.27mm	28mil	1top-2bottomX 5/5	25mil

Breakout Escape

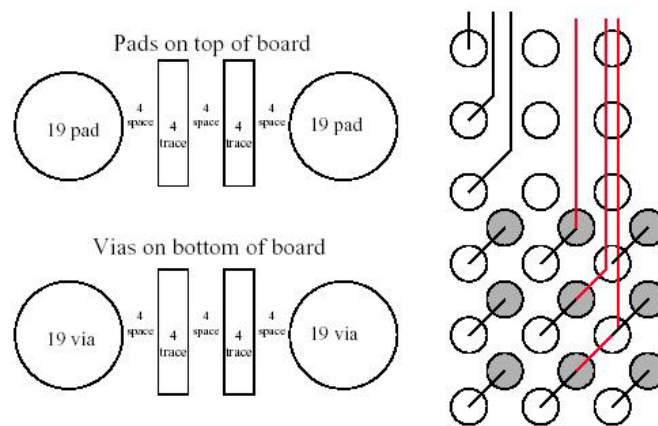
Number of Layers: (Regarding the breakout from the BGA Only)

Pitch	Breakout Escape rows	No. of Layers
1.27mm	T-6; B-6:	4
1.0mm	T-6; B-6:	4
0.8mm	T-4; Any other-6:	4

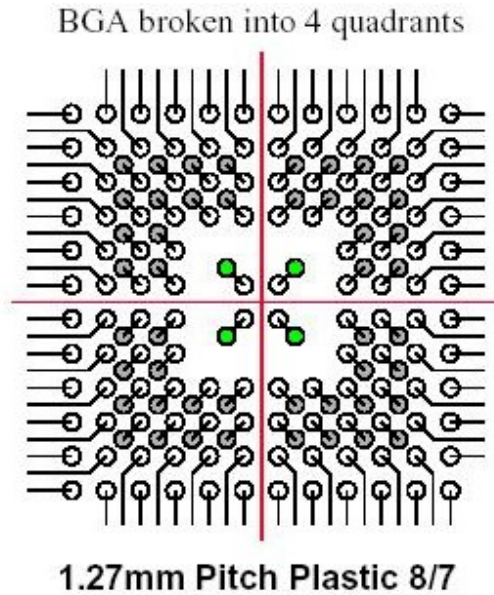
Any additional layer pair enables the outbreak of 6 more rows.

Bellow is an example for breakout escape with 1mm BGA and 4mil trace by 4mil space.

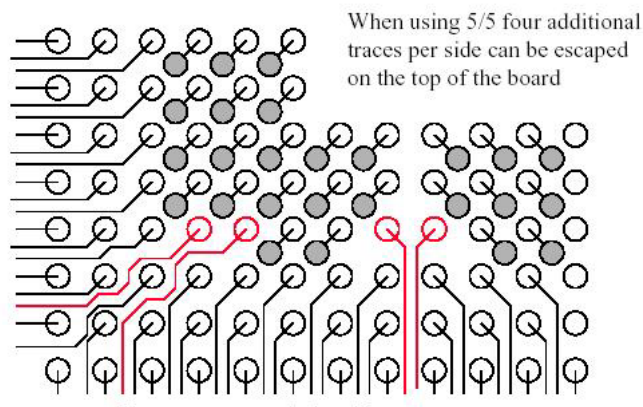
1mm Pitch Plastic 4/4



It is possible (and recommended) to divide the BGA area into four quarters. It makes the breakout escape more symmetrical and flexible.



BGA Quadrants



Placement

- Place BGA far away from board edge
- Leave at least 200mils space for rework around the BGA
- Do not place components under the BGA
- Place the fan out via on grid

Routing

- Use a mm grid and dimension for all BGA routes
- The BGA is a block for route
- Avoid routing signals and additional vias underneath the BGA
- Plug vias from the BGA side
- Do not pour copper under the BGA top side.
- Keep the 75% rule for power trace
- Connect the power first

Summary

In order to route efficiently a board with one or more BGA components and maintain a low layer count, the above rules have to be kept and a close dialog with the PCB manufacturer has to take place.