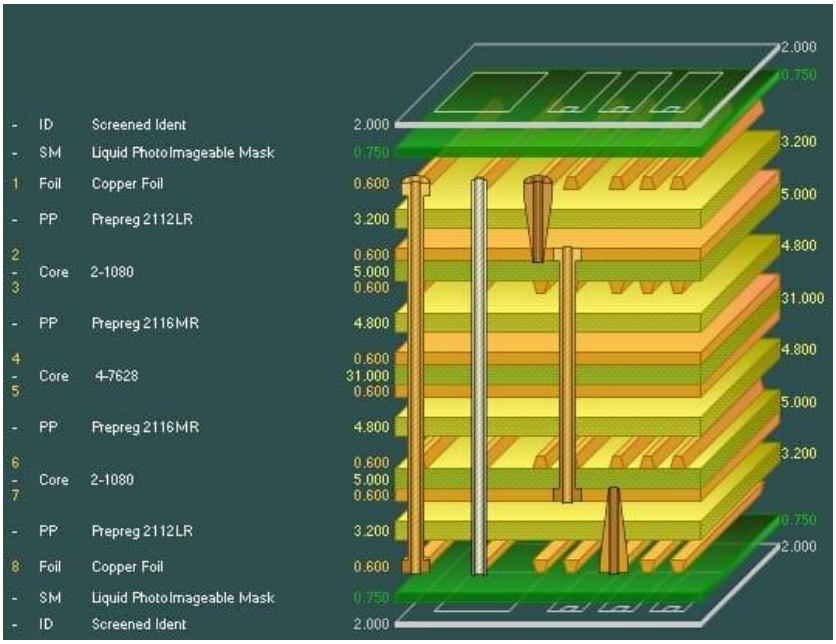
STACKUP DESIGN FOR PDN



Source- Rayming PCB & Assembly

RAGHAVENDRA ANJANAPPA
System Design-SI, PI & EMI-EMC Engineer
Semiconductor – Package Design/High Speed PCB Design/DFx
Applications Engineering – EDA

Requirements and Guidelines for Good PCB Stack-up Design

The PCB stack-up (layer assignment) is an important factor in determining the optimal performance of the power distribution system. An optimized PCB stack-up for higher power integrity performance can be achieved by following these requirements:

Power and ground plane pairs/islands should be closely coupled together. The capacitance formed between the planes can be used to decouple the power supply at high frequencies. Whenever possible, the power and ground planes should be solid to provide a continuous return path for return current.

Use a thin dielectric thickness between the power and ground plane pair. Capacitance is inversely proportional to the separation of the plane pair. Minimizing the separation distance (i.e., the dielectric thickness) maximizes the capacitance.

Keep the power and ground plane pair as close to the TOP and BOTTOM surfaces as possible (see Figure 1). This will help in minimizing the decoupling capacitors mounting, via, and the power/ground plane pair spreading loop inductance.

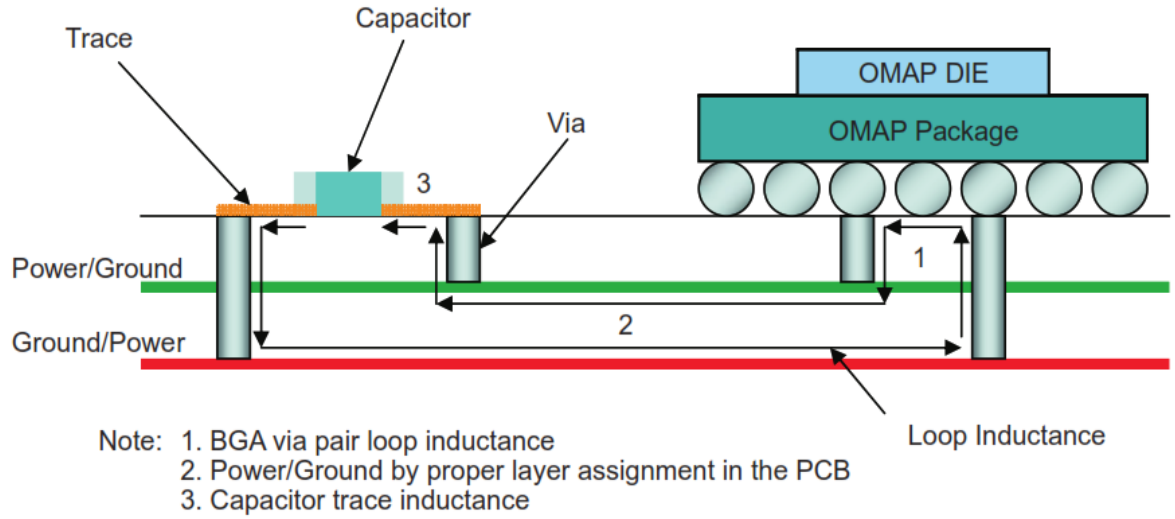


Figure 1 Minimize loop inductance by proper layer assignment in the PCB

The placement of power and ground planes in the PCB stack-up (determined by layer assignment) has a significant impact on the parasitic inductances of power current path as shown in Figure 1. For this reason, it is recommended to consider layer order in the early stages of the PCB PDN design cycle, putting high priority supplies in the top half of the stack-up and low-priority supplies in the bottom half of the stack-up as shown in the following figures. Figure 2 and Figure 3 show examples of typical PCB stack-up designed with power integrity in mind.

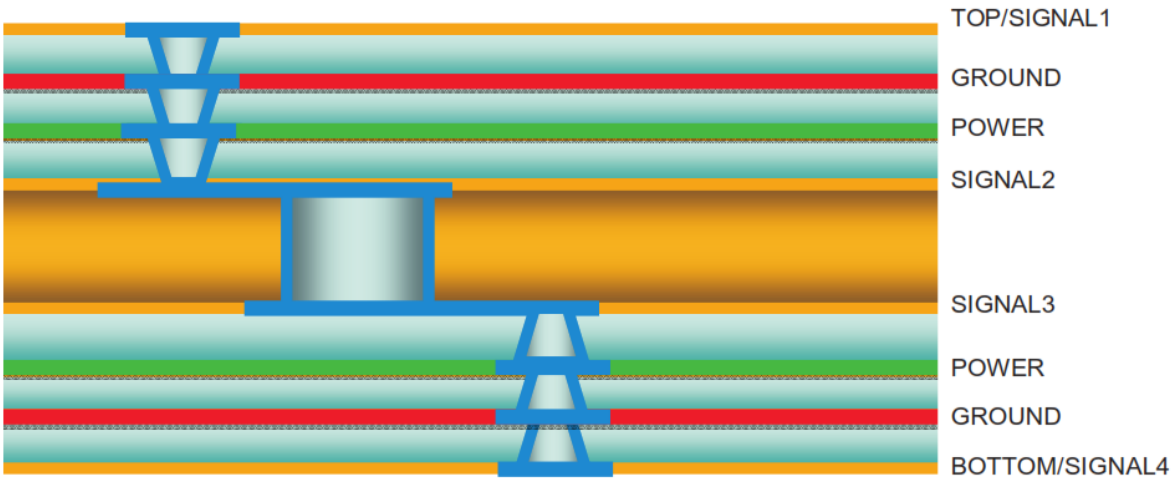


Figure 2 Example of layer assignment that uses the HDI Vias

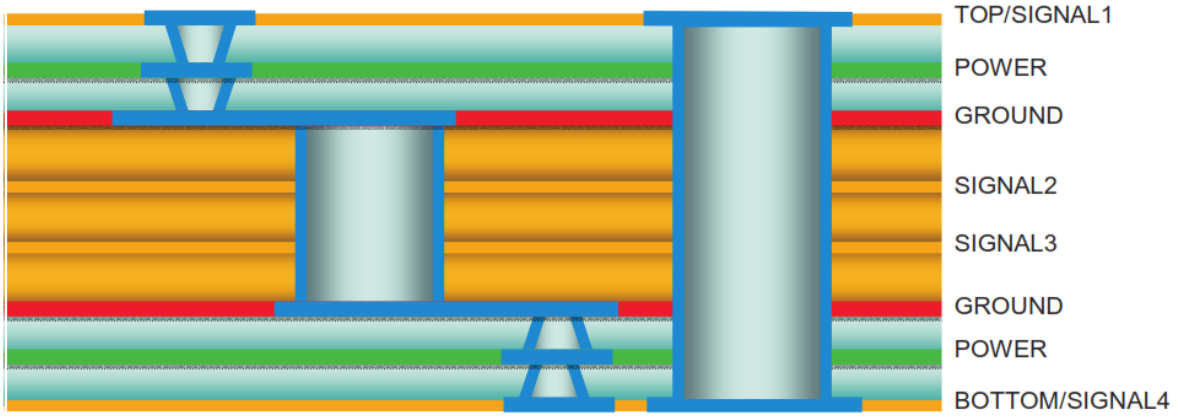


Figure 3 8 -layer stack-up that uses plated through hole Vias

Reference Layer Stack-up

Below are some reference layers stackup rated based on the below factors in consideration.

1. Signal Integrity
2. Power Integrity
3. HDI Cost
4. Route Density

STACKUP A

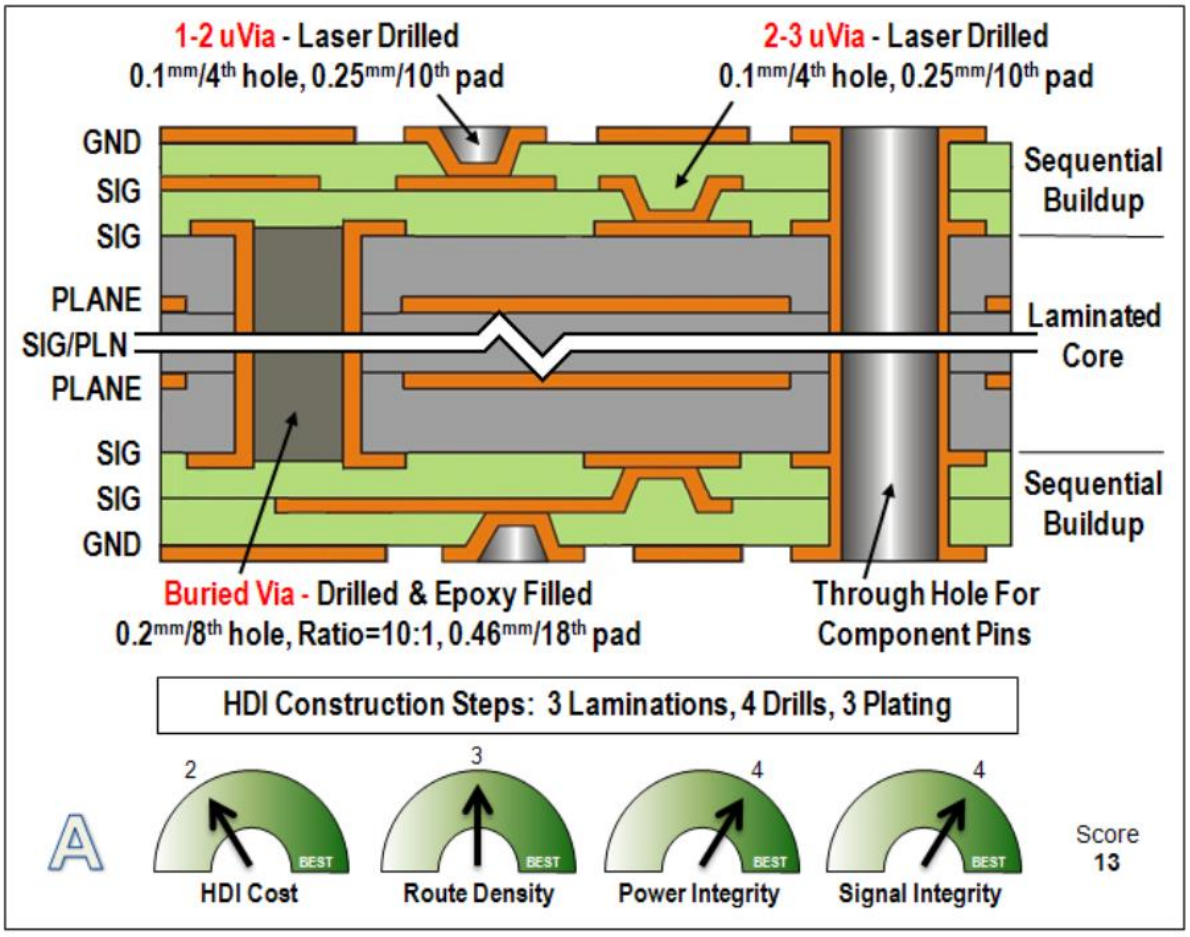


Image Source - BGA Breakouts and Routing by Charles Pfeil Second Edition

Stackup A Comments:

Total score = 13

- This is a great average of the variables and a good stackup if you are starting out with HDI.
- The via models are simple and it won't be difficult to find vendors who can fabricate them.
- The ground plane on the outer layers provides a high rating for power and signal integrity.

STACKUP - B

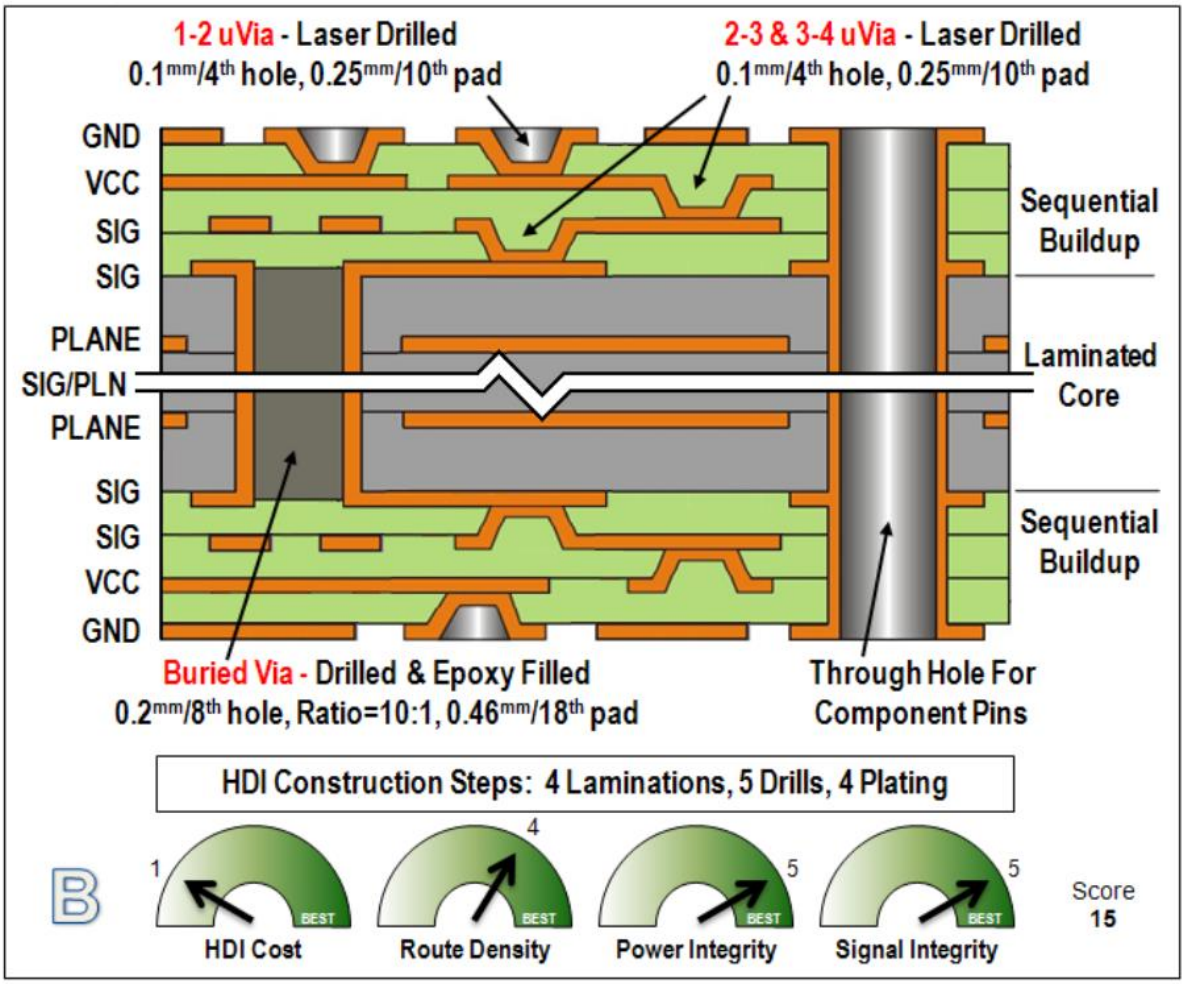


Image Source - BGA Breakouts and Routing by Charles Pfeil Second Edition

Stackup B Comments:

Total score = 15

- The GND and VCC on the outer layers provide the best power and signal integrity.
- The additional buildup layer increases the cost (more laminations, drills, and plating steps) but also improves the route density as opposed to losing an HDI routing layer due to the VCC plane.
- The via models are simple and it won't be difficult to find vendors who can fabricate them.

STACKUP – C

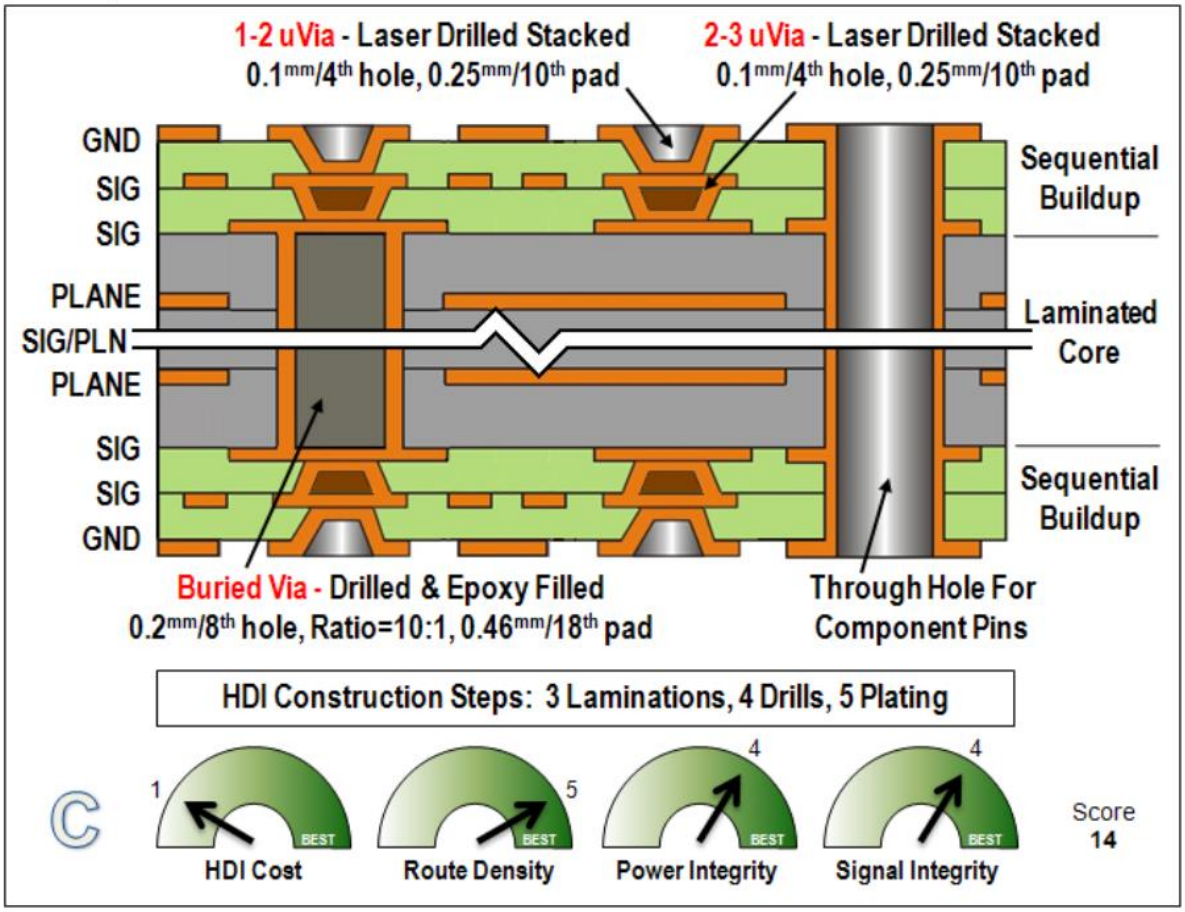


Image Source - BGA Breakouts and Routing by Charles Pfeil Second Edition

Stackup C Comments:

Total score = 14

- The stacked vias enable the best route density but also increases the cost and may limit the number of vendors who can fabricate this stackup.
- The ground plane on the outer layers provides the high rating for power and signal integrity.

STACKUP – D

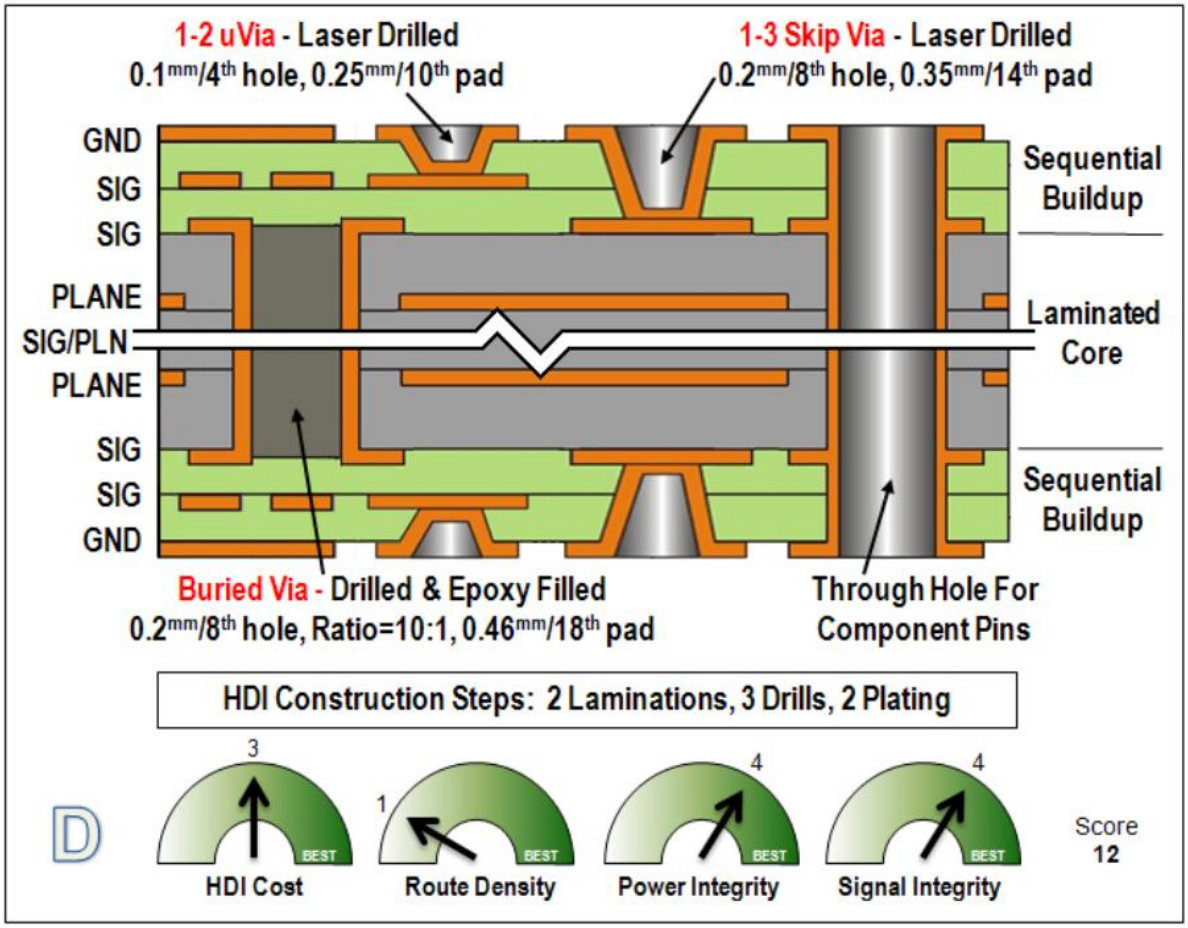


Image Source - BGA Breakouts and Routing by Charles Pfeil Second Edition

Stackup D Comments:

Total score = 12

- The ground plane on the outer layers provides the high rating for power and signal integrity.
- The skip via reduces laminations and plating steps, which lowers cost; however, contributes to a relatively low route density.

STACKUP – E

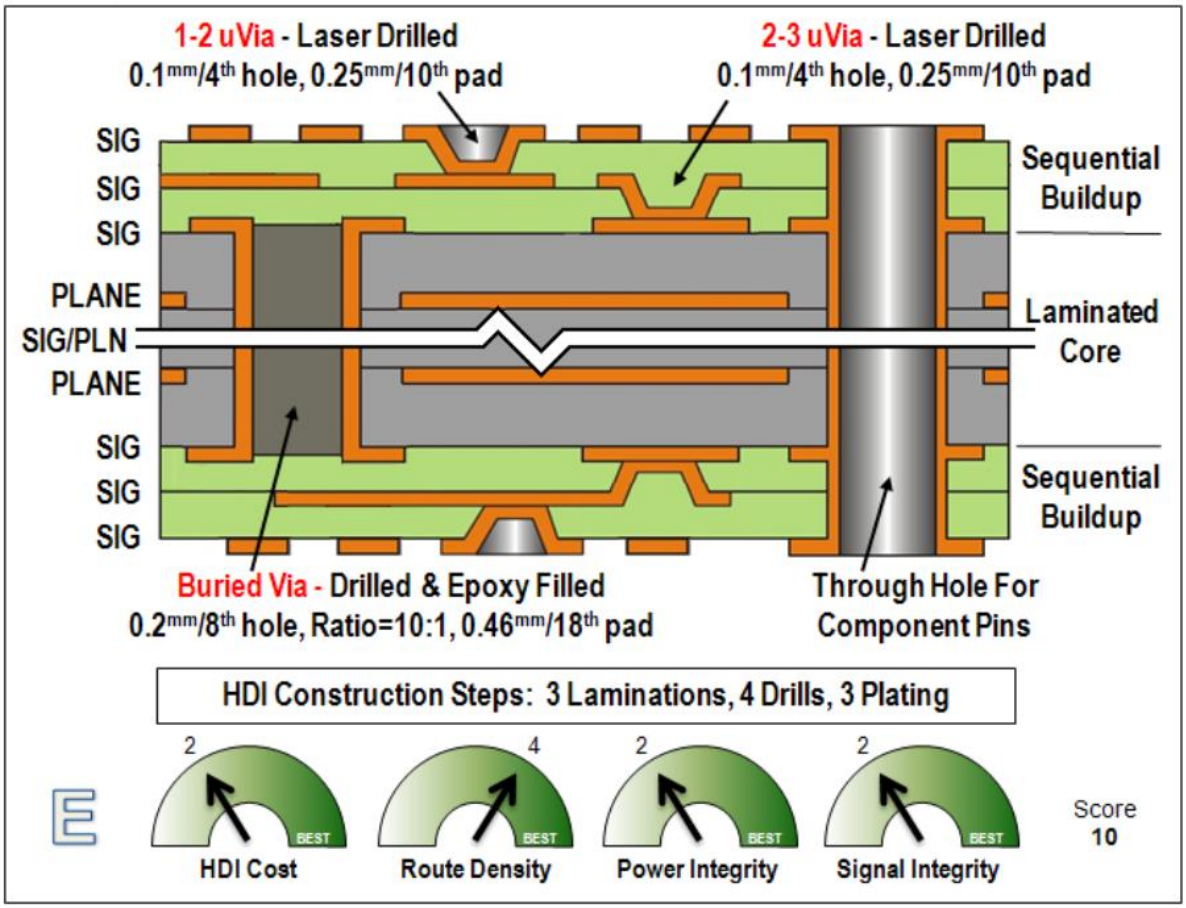


Image Source - BGA Breakouts and Routing by Charles Pfeil Second Edition

Stackup E Comments:

Total score = 10

- The lack of a ground plane on the outer layers reduces power and signal integrity; however, it does provide for improved route density assuming routing would be done on the outer layers.
- The via models are simple and it won't be difficult to find vendors who can fabricate them.

STACKUP - F

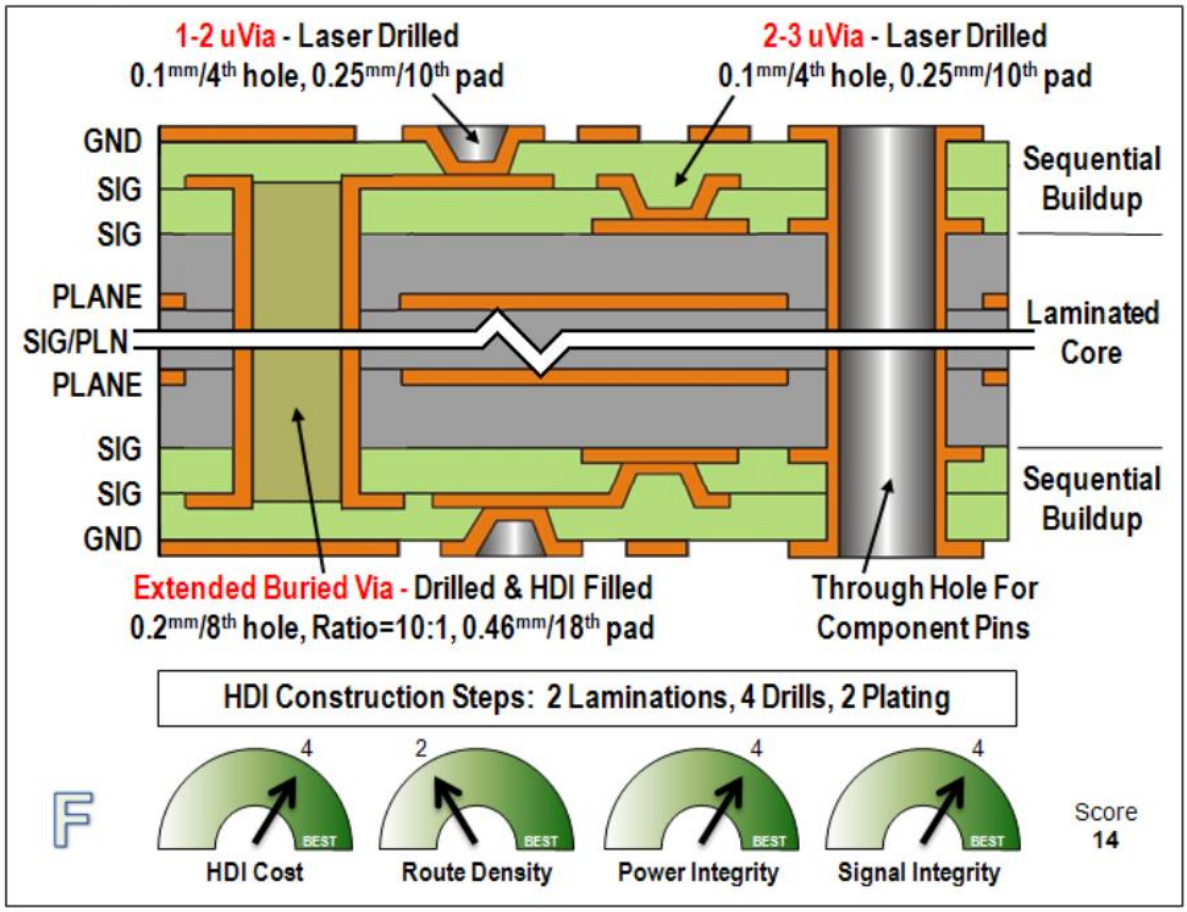


Image Source - BGA Breakouts and Routing by Charles Pfeil Second Edition

Stackup F Comments:

Total score = 14

- The ground plane on the outer layers provides the high rating for power and signal integrity.
- The extended buried via reduces the lamination and plating steps which lowers cost; however, it reduces route density

STACKUP – G

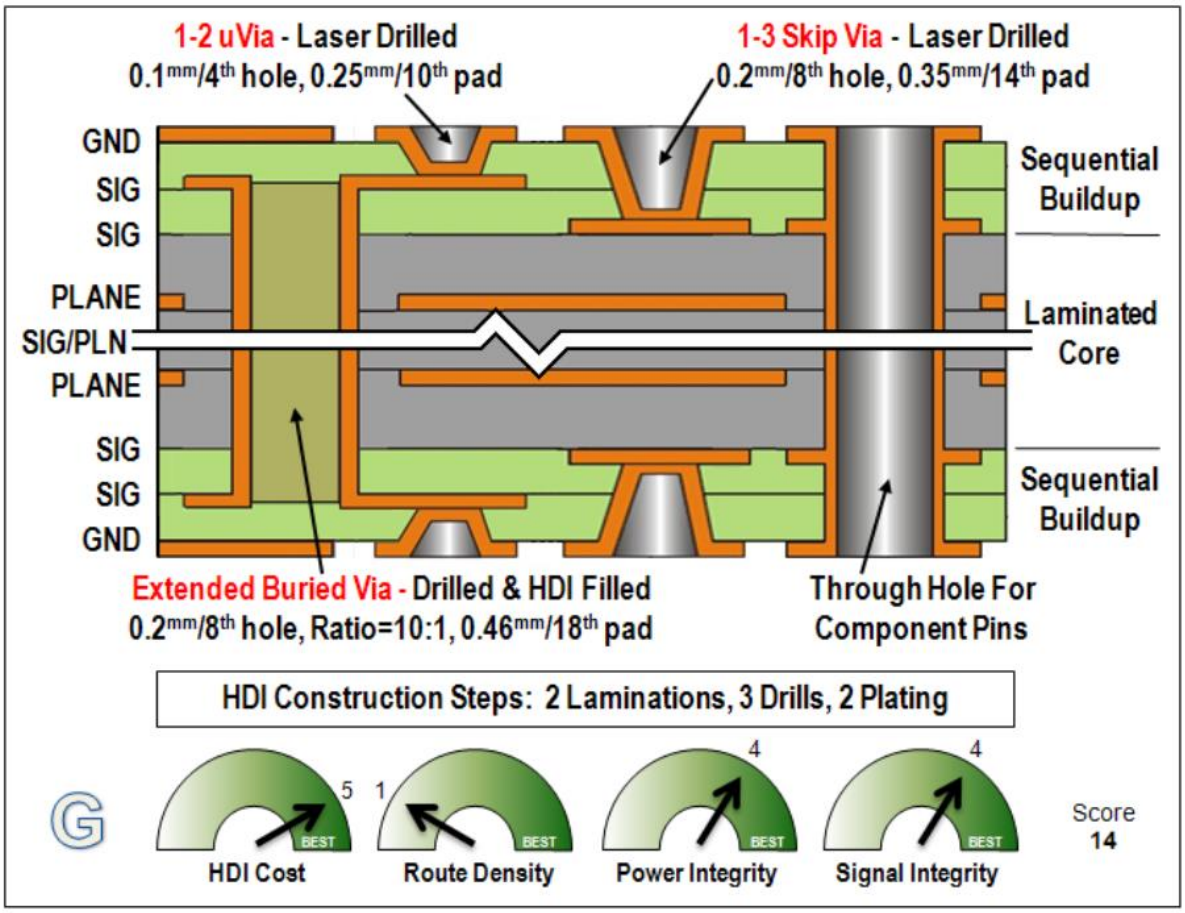


Image Source - BGA Breakouts and Routing by Charles Pfeil Second Edition

Stackup G Comments:

Total score =14

- The ground plane on the outer layers provides the high rating for power and signal integrity.
- The extended buried via and the skip via reduces the lamination and plating steps which lowers cost; however, it also reduces route density.

References

Book – BGA Breakouts and Routing by Charles Pfeil Second Edition.

www.ti.com