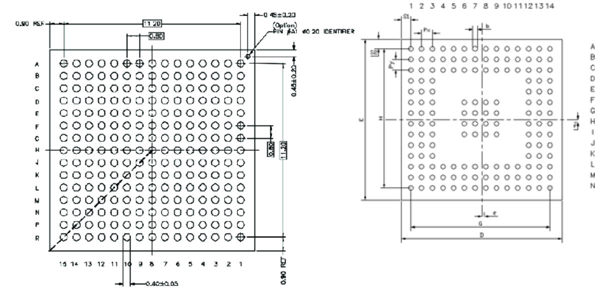
FINELINE

The Global PCB Experts

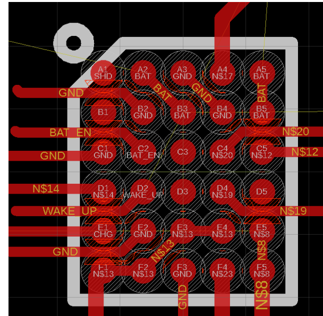
Design recommendations for 0.4 mm pitch BGA

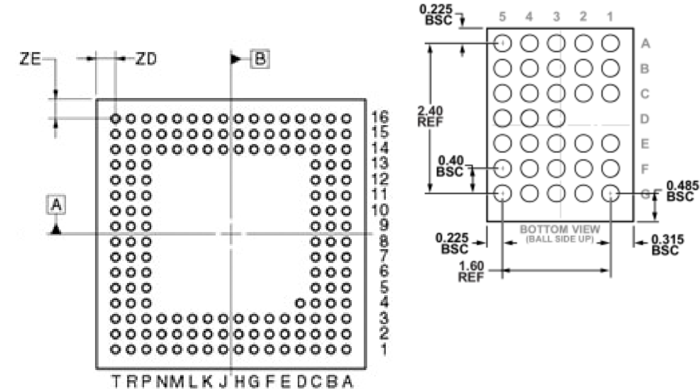
0.4 mm pitch BGA component

- Different 0.4 mm pitch footprints
- Design rules are the same for all footprints
- Stacked via is needed to fan-out each BGA row
- Number of BGA rows determin the level of stacked via
- “ELIC”(Every Layer Inter Connect) necesarry >10 rows



Stacked via with ELIC technology





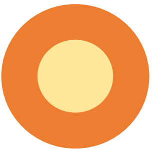
Via constructions

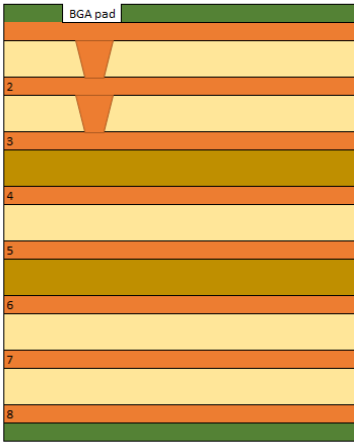
FINELINE

- Stacked via in pad
- Stacking vias and via in pad gives more design freedom
  - Vias will be filled and capped with copper(IPC 4761 Type VII)
- Fan-out each BGA row on individual layer

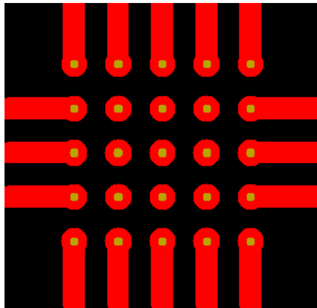
Stacked via

Via in pad

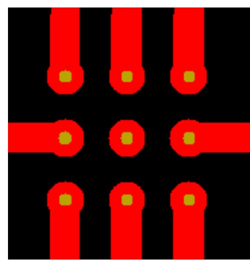




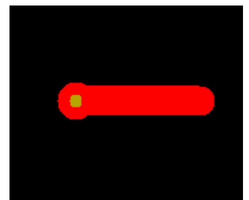
Row 1 fan-out on layer 1



Row 2 fan-out on layer 2



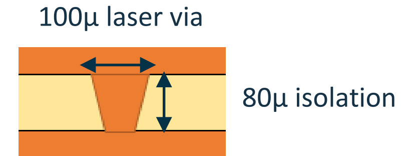
Row 3 fan-out on layer 3

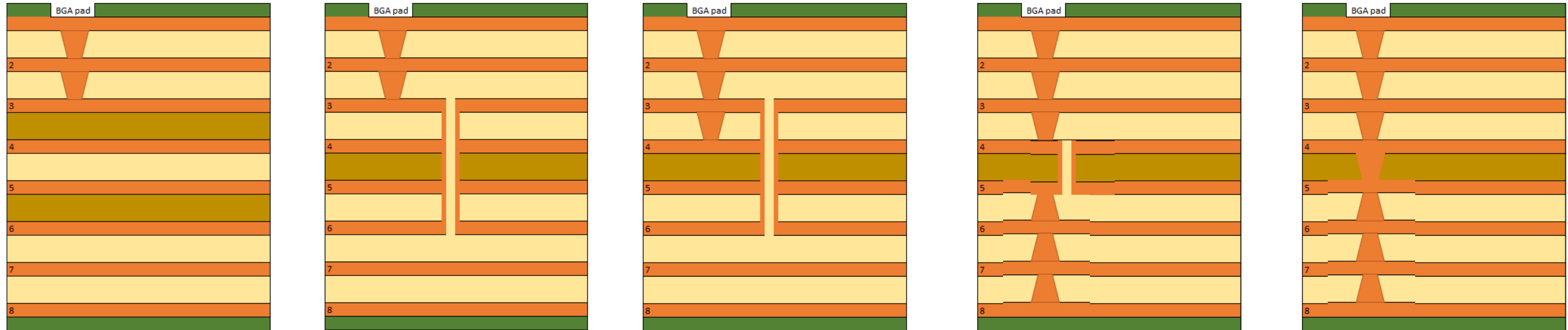


Stack-up configurations

FINELINE

- Important to keep construction symmetrical
- Aspect ratio of blind via max. 1:1 (preferred 1:0,8)
  - e.g. 100µ hole, 100µ(80µ) isolation between layer



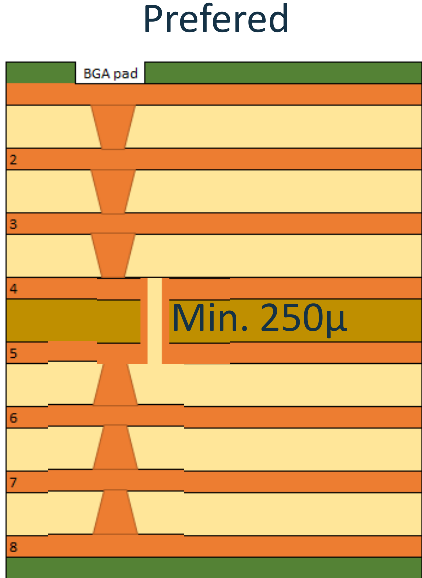


ELIC (Any layer technology)

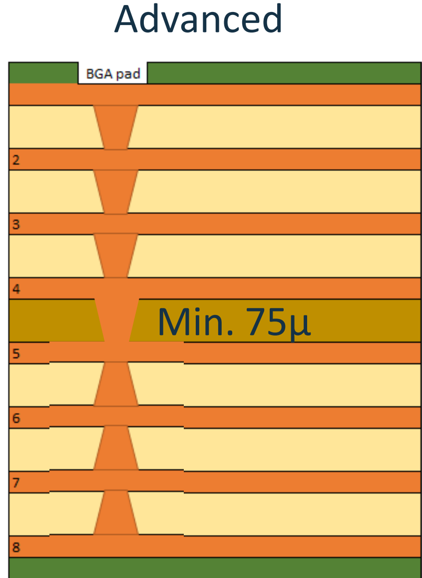
FINELINE

- ELIC (Every layer Inter Connect)
  - Connection with any layer pair possible
    - E.g. 1-3, 2-7, 1-6, 5-8, etc. etc.
- Two options for middle core
  - With buried via resin filled (prefered)
    - Minimum core thickness 250µ
  - With buried via Cu filled (advanced)
    - Core 75µ

Prefered



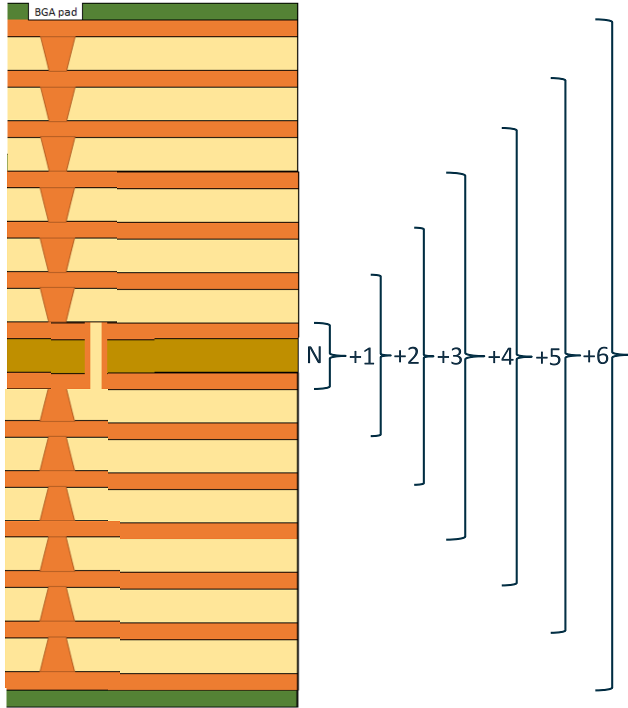
Advanced

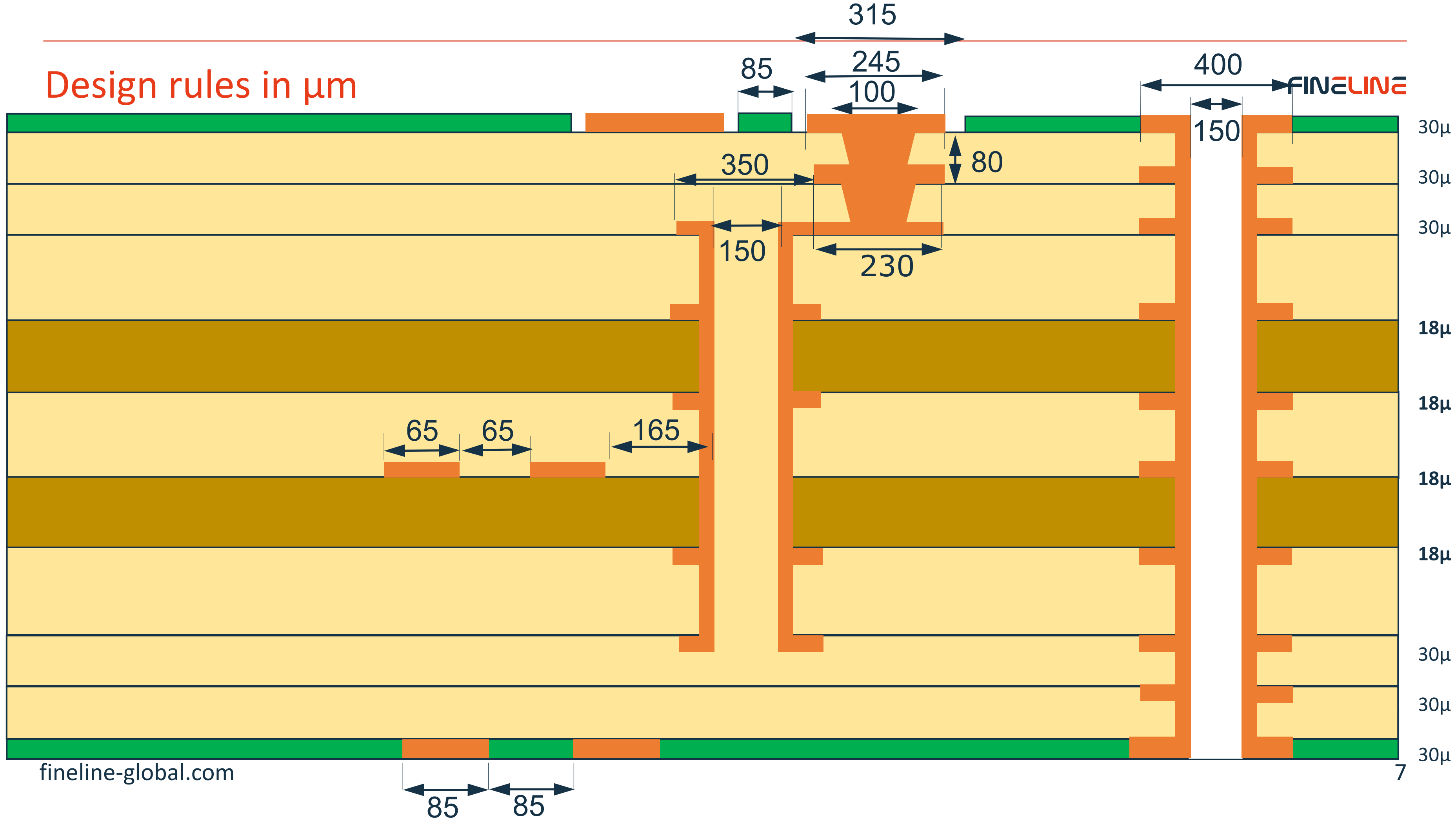


ELIC (Any layer technology)

FINELINE

- Maximum number of stacked via is 6 per side
- 6+N+6 (7 times pressing)
- High performance material required
  - High Tg, high Td, high TMA
  - Low CTE
  - Halogen free
- Material must be suitable for 7 times pressing and 5 times reflow

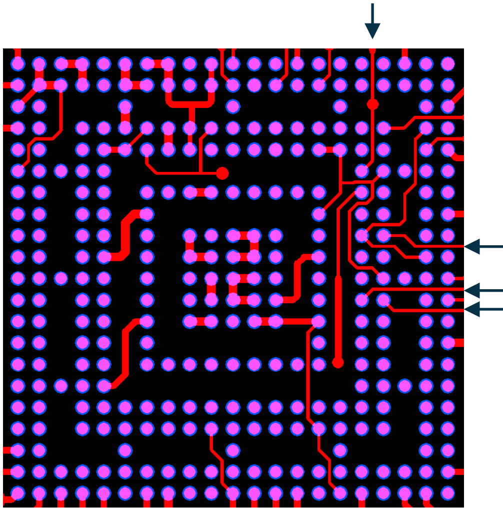


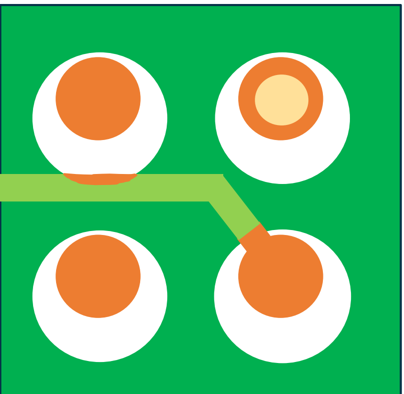

Design rules in µm
FINELINE
315
85
245
100
400
150
80
350
150
230
65
65
165
85
85
30µ
30µ
30µ
18µ
18µ
18µ
18µ
30µ
30µ
30µ
fineline-global.com

Non preferred

FINELINE

- Do not route traces between BGA balls
  - difficult to compensate the 50µ trace so trace will end up at ~35µ(low yield)
  - Solder mask mis-alignment can cause exposed trace(low yield)

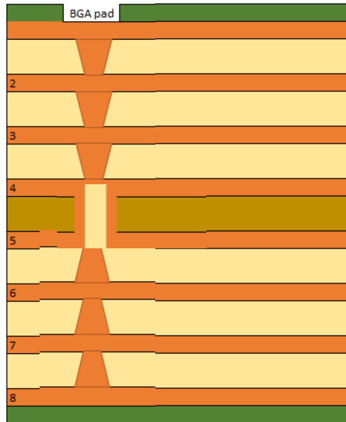


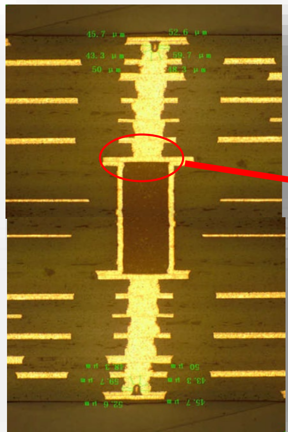


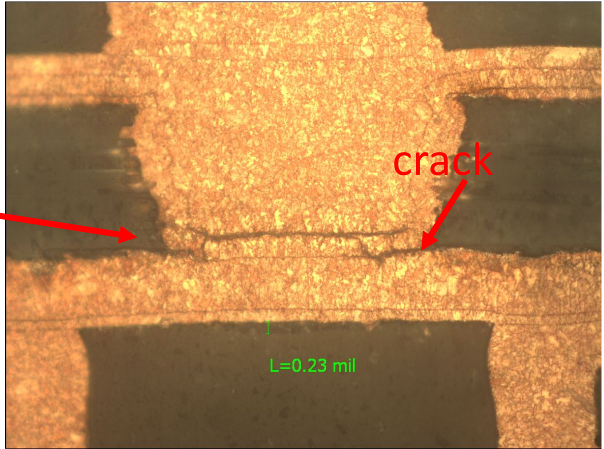
Non prefered

FINELINE

- Not recommended to do blind on top of buried via
- Possible relaibility issues when exposed to high temperatures
  - Z-axis expansion is the main problem so material with ultra high Tg is key
  - > 3 times reflow can be a problem







Contact

FINELINE

- If you are not sure what which design rule to choose for your design please contact one of our technical members of the Fineline team for support.

FINELINE

The Global PCB Experts

Boy van Veghel | boy.vanveghel@fineline-global.com