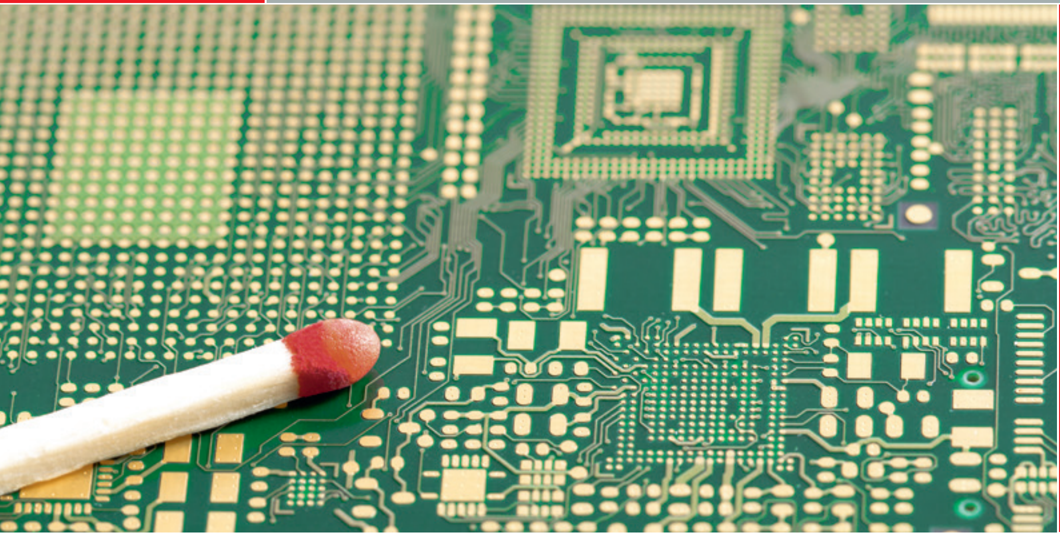
DESIGN GUIDE HDI
Version 1.2 / September 2018

WÜRTH ELEKTRONIK

HDI Design Guide



Through hole vias or microvias?
It is also a question of reliability!

According to the IPC-2221A/IPC-2222 design guidelines a maximum aspect ratio of 6:1 to 8:1 is recommended for through hole vias (aspect ratio = relation hole depth to drill diameter). Likewise, a minimum drill diameter of 0.25 mm is recommended for a 1.60 mm standard PCB thickness.

These parameters are absolutely suitable for production and they are also recommended by Würth Elektronik. For high IPC Class 3 reliability requirements parameters like this are essential. Due to reliability reasons it is not possible to arbitrarily decrease the via pad size and the hole diameter. The IPC 2221A design recommendations and tolerances result in a minimum pad size of 0.55 – 0.60 mm.

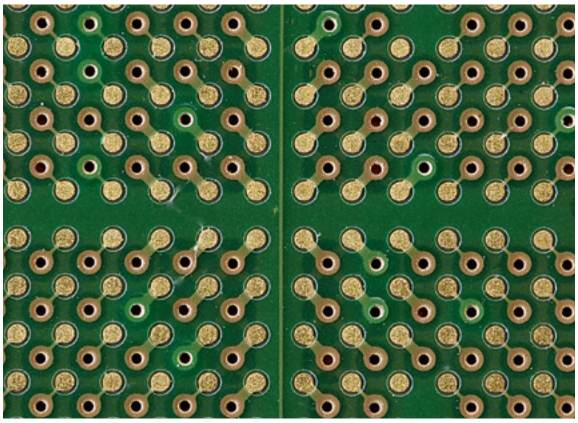


Figure 1: Pitch 1.00 mm

BGA pitch 1.00 mm

If the overall complexity allows it, a design with only through hole vias (i.e. without microvias) can be used with an IPC conform standard-pad size of 0.60 mm (see figure 1).

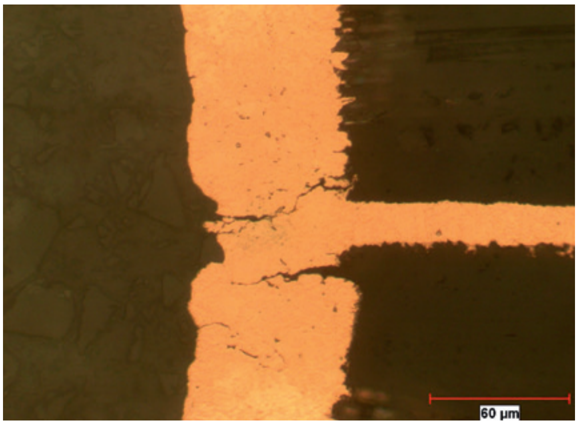


Figure 2: Barrel Crack

Fine pitch BGA

This is hardly feasible for Fine Pitch BGA components. If for such features the aspect ratio is too critical or the drill diameter is too small then reliability problems threaten. Due to different expansion coefficients of copper and FR4 material the barrels of the vias could crack all too soon, either during multiple lead free solder processes or at the latest by reliability tests (see figure 2).

The solution: laser drilled microvias

If there is not enough space for a suitable via pad size, the much smaller microvias should be used. Microvias can be used up to a drill hole depth of approx. 100 µm and due to their short barrel they do not have any problems with different expansions (see figure 3). For this reason, microvias are more reliable than small through hole vias as a general rule.

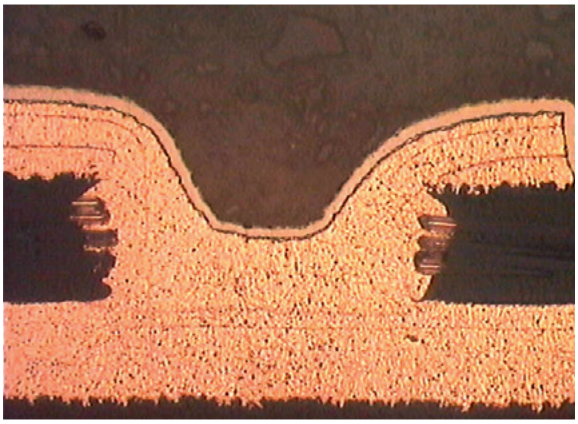


Figure 3: Laser drilled microvias

BGA 0.80 mm pitch

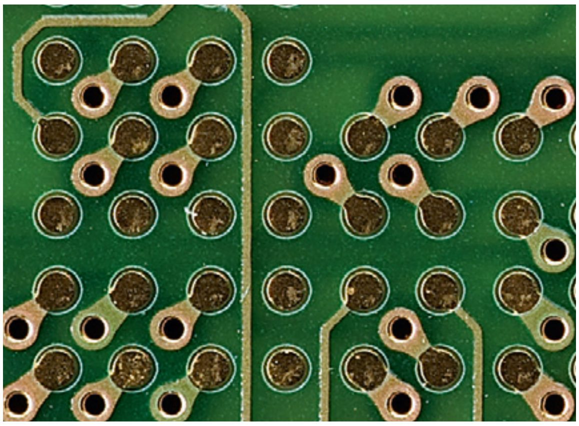
For 0.80 mm pitch BGAs and PCBs of not too high complexity (max. thickness 1.80 mm, max. 12 layers) it is possible to use through hole vias. In order to achieve this the via pad size has to be reduced to 0.50 mm. Using such a pad size a reliable production is possible with limited manufacturing tolerances.

Microvia solutions are generally recommended and for complex boards or for complex BGA components with a high number of connected pins they are absolutely necessary.

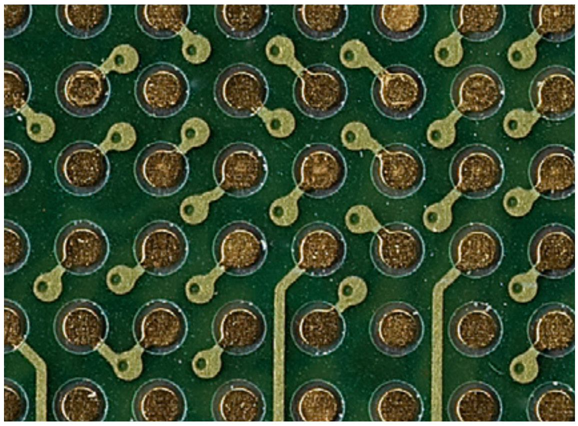
Variant 2 shows microvias connected with dogbones. The advantage here is an absolutely planar surface of the solder pads as microvias are not set into the BGA pads. The disadvantage of this variant is that routing tracks between the pads on the outer layer is hardly feasible.

Variant 3, via in pad, is the most frequently used technique. With this variant the outer layers can be completely used for routing.
The dimples caused by the microvias do indeed somewhat raise the risk of voiding, but with suitable soldering conditions this can be safely controlled. For special cases an additional microvia filling process could be used to avoid the dimples and the risk of voiding, but there are extra costs involved in this.

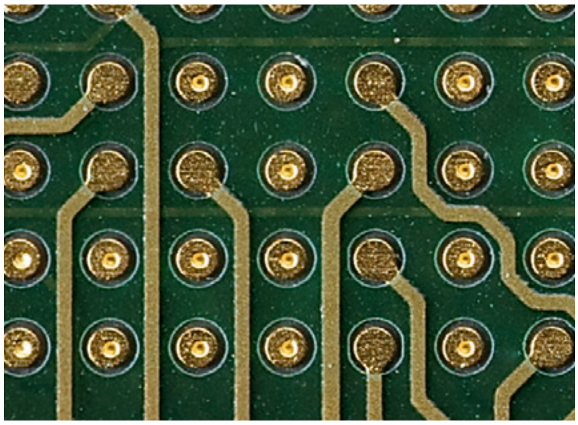
Combinations of the three variants are of course also possible.



Variant 1: Dogbone with through-hole vias



Variant 2: Dogbone with microvias

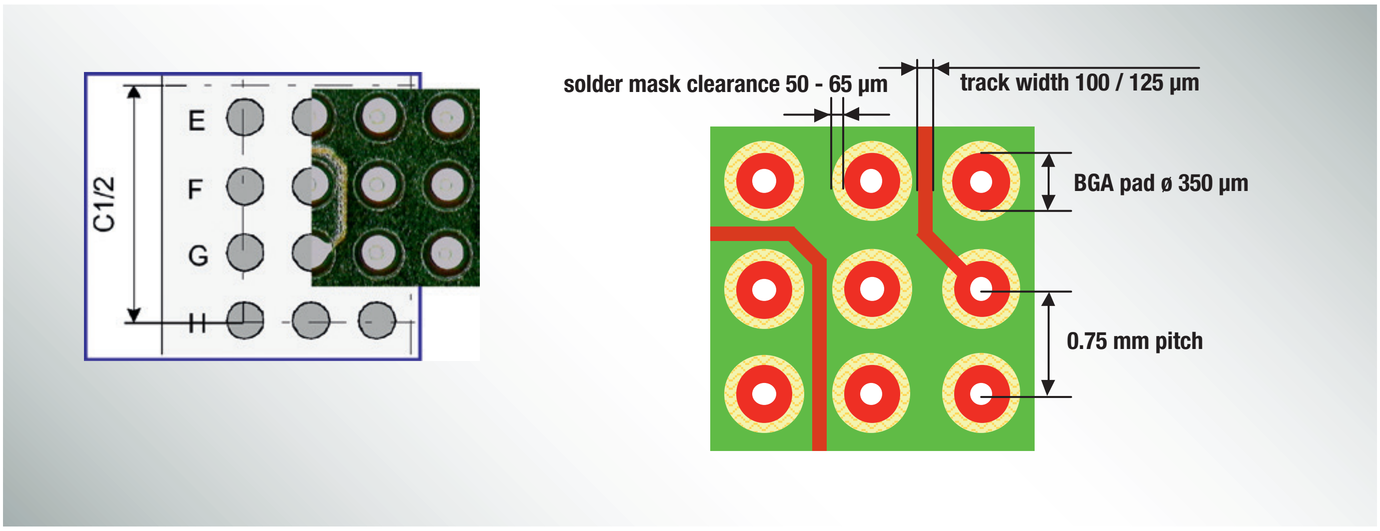


Variant 3: Microvia in pad

	Variant 1	Variant 2	Variant 3
BGA solder pad	max. 400 µm	-	max. 500 µm
Solder mask clearance	50 µm	≥ 50 µm	50 µm
Via pad size BGA area	500 µm	-	-
Microvia pad outer layers	-	300 / 350 µm	300 / 350 µm
Microvia pad inner layers	-	300 / 350 µm	300 / 350 µm
Track width / spacing outer layers	≥ 100 µm	≥ 100 µm	≥ 100 µm
Track width / spacing inner layers	≥ 100 µm	≥ 100 µm	≥ 100 µm

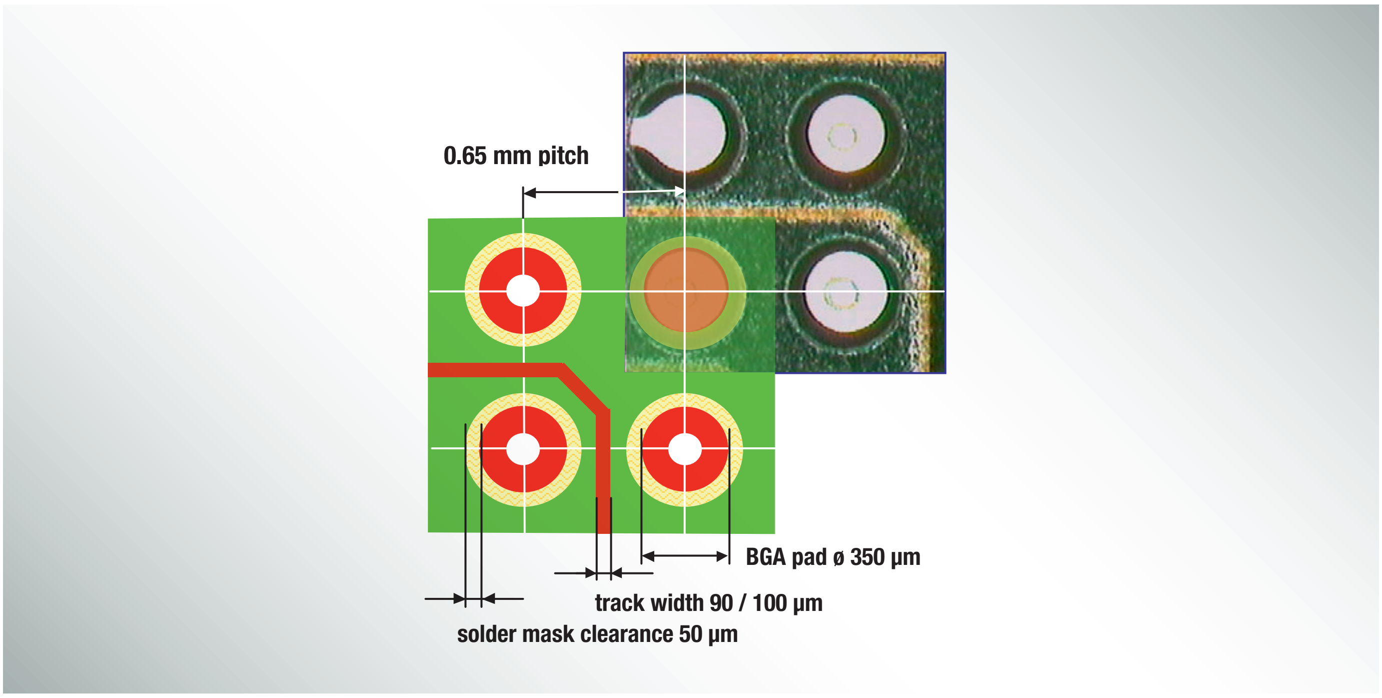
BGA 0.75 mm pitch

With 0.75 mm pitch BGAs a sufficient pad size for through hole vias is hardly feasible within the BGA area. In order to ensure reliability, Würth Elektronik recommends using microvias for such components. In this way standard parameters for track widths, spacing, pad sizes and solder mask clearances can be utilised.



BGA 0.65 mm pitch

For 0.65 mm pitch BGAs microvias are definitely required. The track width in the BGA area may need to be reduced to 90 µm, or in rare cases even less than this. This depends on the BGA pad size used.



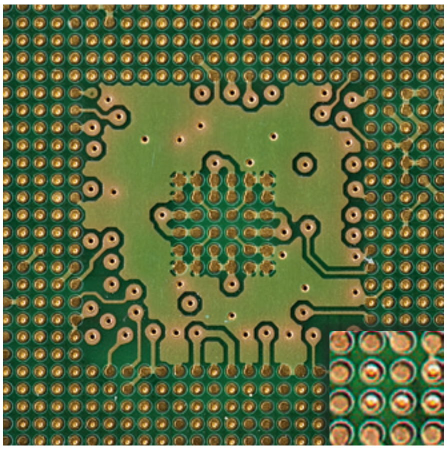
BGA 0.50 mm pitch

With Variant 1 fine line structures will definitely be required for a 0.50 mm pitch BGA, we recommend 75 µm (3 mil). It will also be necessary to decrease the microvia pad size, at least on the inner layers, to 275 µm. For 75 µm fine line structures the final copper thickness on the surface is limited to approximately 25 µm.

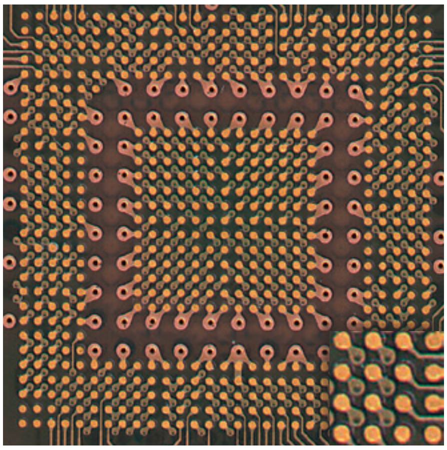
Würth Elektronik recommends **variant 1** described above, without tracks between the solder pads on the outer layer. This avoids the need to use fine line structures on the outer layers.

Variant 2 gives the advantage of a planar surface (lower risk of voiding), but with a reduced solder pad size.

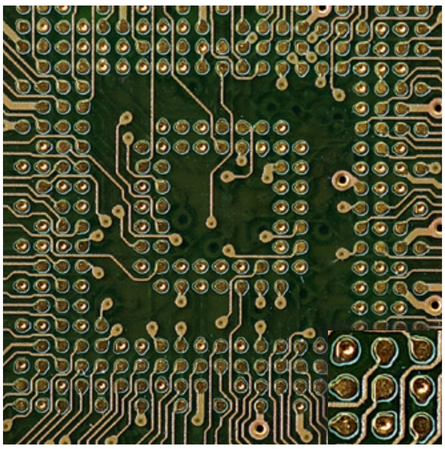
With **variant 3,** 75 µm structures are needed on the outer layer as well. This increases the production effort and the production costs. Moreover, the solder mask clearance has to be reduced to 35 µm. This variant could probably help to save one microvia layer. Generally the number of the microvia layers required, and therefore the kind of stack-up, depends on the complexity of the component.



Variant 1: Via in pad



Variant 2: Dogbone



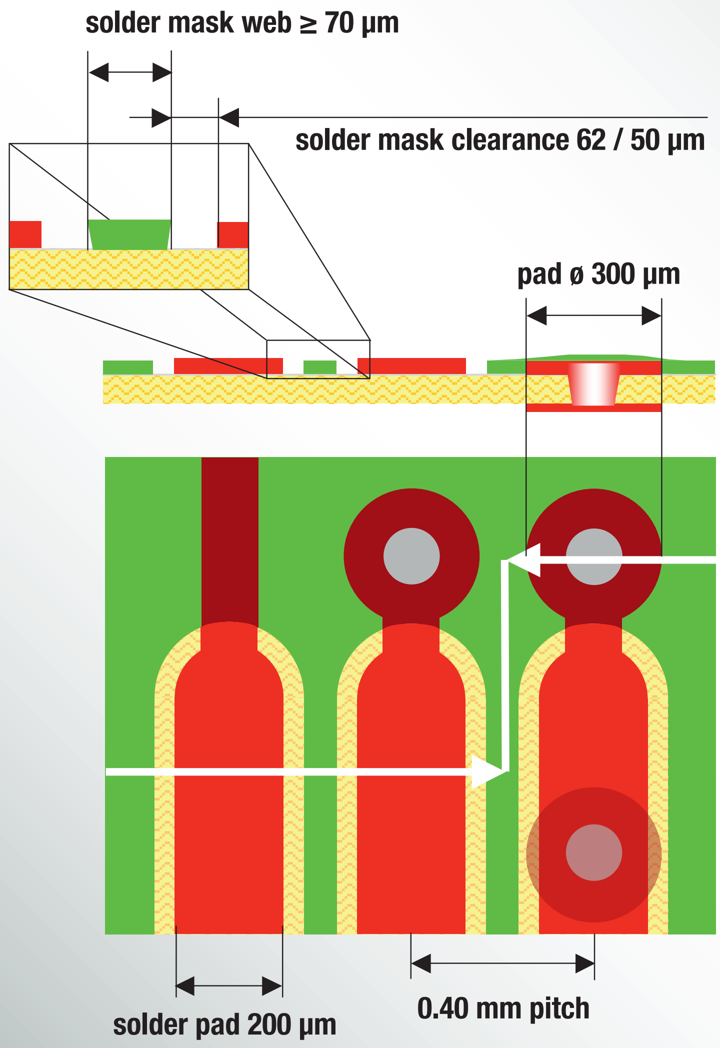
Variant 3: Via in pad

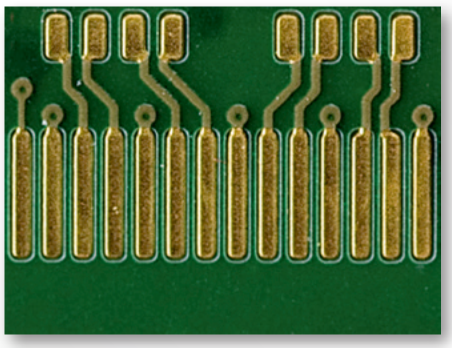
	Variant 1	Variant 2	Variant 3
BGA solder pad	300 - 330 µm	240 µm	275 µm
Solder mask clearance	50 µm	40 µm	35 µm
Microvia pad outer layers	≥ 300 µm	275 µm	275 µm
Microvia pad inner layers	275 µm	275 µm	275 µm
Track width / spacing outer layers	≥ 100 µm	80 / 90 µm	75 µm
Track width / spacing inner layers	75 µm	75 µm	75 µm

QFP 0.40 mm pitch

Microvias could also be of use for QFP components with a 0.50 mm and more particularly with a 0.40 mm pitch.
If the amount of space available permits, microvias can be placed outside of the solder pads using standard pad sizes. If the routing area is not large enough to allow this, then a microvia pad can be set on top of the narrower solder pad.

The fact that a minimum solder mask web width of not less than 70 µm is used must be taken into consideration. With a usual solder pad width of 200 µm it is also possible for 0.40 mm pitch QFPs to have a solder mask web between all solder pads.

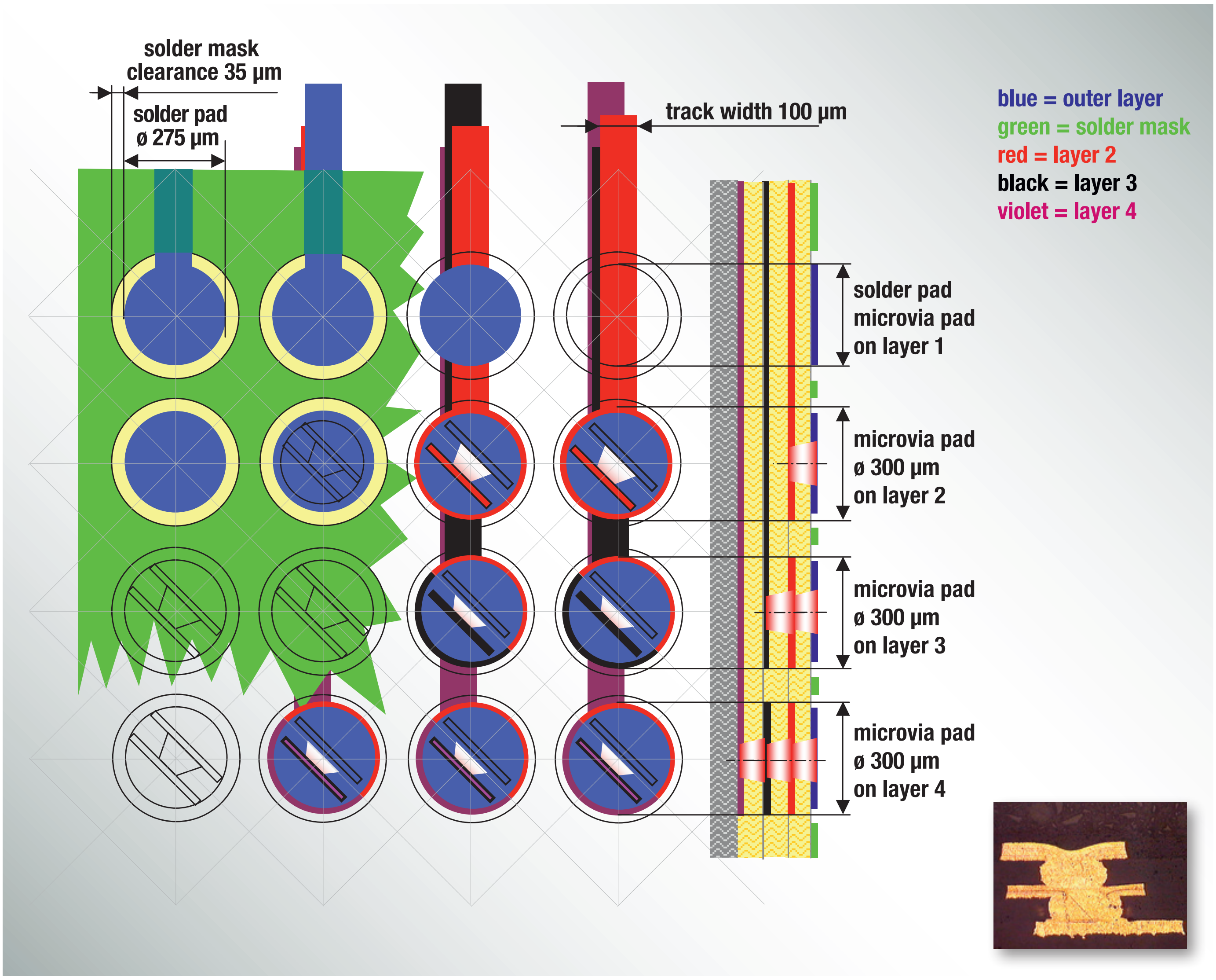




BGA 0.40 mm pitch

When dealing with a 0.40 mm pitch BGA it is possible to utilise stacked microvias in order to avoid fine line structures below 75 µm. There are, however, additional costs involved with this, as the microvias on the inner layers have to be filled and a more complex stack-up may be necessary. But in this way 0.40 mm pitch BGAs can be routed using well established technologies.

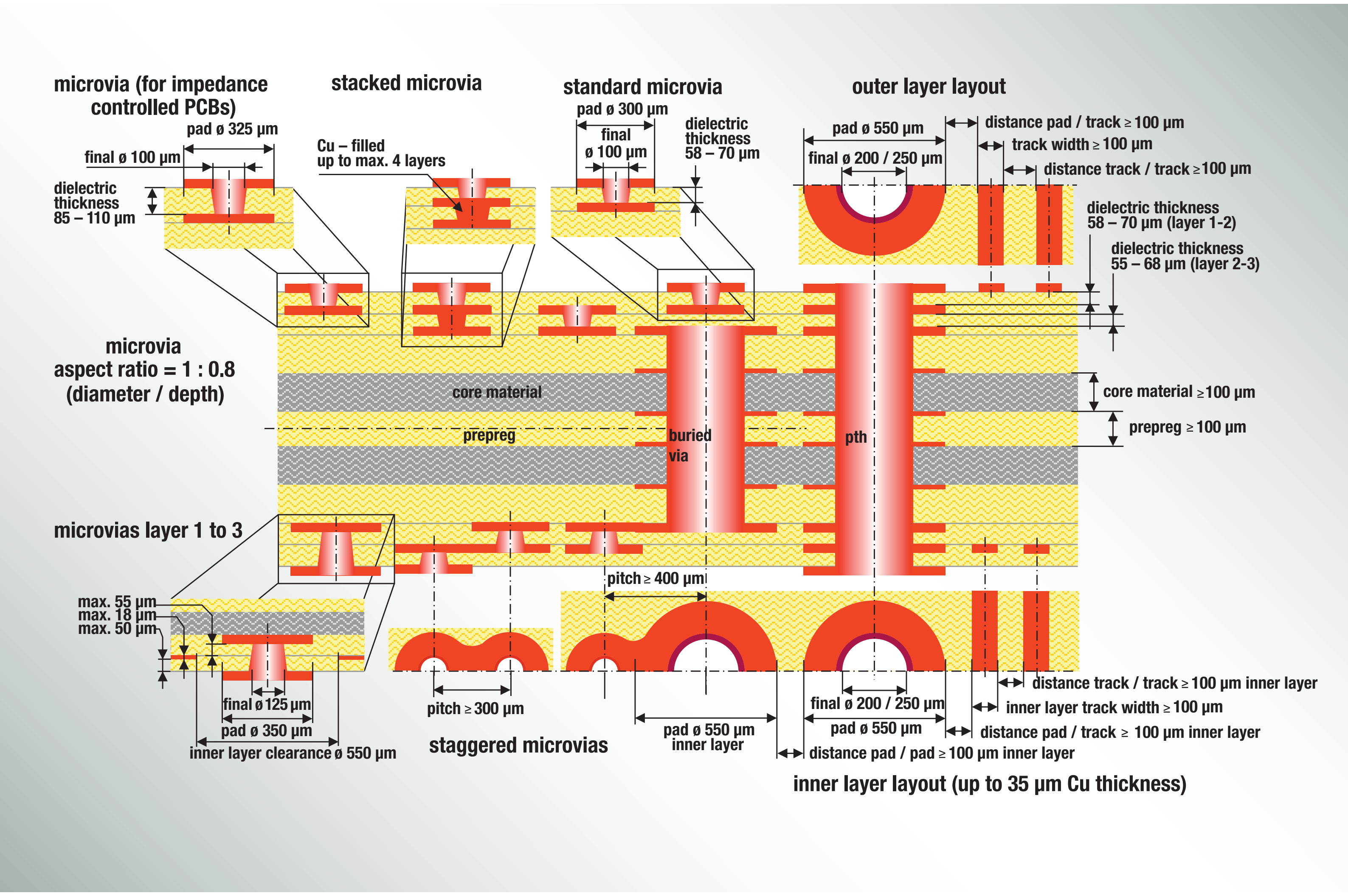
The kind of stack-up and the number of microvia layers required depends on the complexity of the component. As with all other BGA components we strongly recommend the use of Non Solder Mask Defined pads for 0.40 mm pitch BGAs, particularly due to the high mechanical stability of such a solder joint.



BGA solder pad	275 µm
Solder mask clearance	35 µm
Microvia pad inner layers	300 µm
Track width / spacing outer layers	≥ 100 µm
Track width / spacing inner layers	≥ 100 µm

WÜRTH ELEKTRONIK

HDI Microvia Standard Design Rules



For more information about HDI Microvia Technology visit our website: www.we-online.com/microvia

Würth Elektronik GmbH & Co. KG
Circuit Board Technology
Salzstraße 21
74676 Niedernhall · Germany
Tel: +49 (0) 7940 946-0
Fax: +49 (0) 7940 946-550000
cbt@we-online.de