



PCA 40 QFN 5X5 NDG 40 package
 QFN EXPOSED DIE PAD DIMENSIONS = 3.5 mm +/- 0.10 mm
 QFN I/O PAD LENGTH = 0.4 +/- 0.10 mm max
 QFN I/O PAD WIDTH = 0.17 mm min 0.20 mm typ 0.25 mm max

PCA 40 QFN 5x5 NDG 40 package the land pattern dimensions should be as follows.

PCB THERMAL LAND DIMENSIONS = 3.5 mm
 PCB I/O PAD LENGTH = 0.6 mm
 PCB I/O PAD WIDTH = 0.20 mm

Thermal Via

Maximum number of vias allowed by PCB design
 Diameter of 0.3 mm
 Via pitch of 1.0 mm

Note:

This is 0.4mm pitch package with PAD max width of 0.25 mm. Solder mask web between pads may not be possible. If this is the case, use one large opening for all 10 pads (one side) and no solder mask between pads. Attention to stencil aperture size-to-land size is required. Thermal via grid is required for all devices that have amplifier.

Recommendation:

Solder Mask Design:

Use Non-Solder Mask Defined (NSMD) for I/O lands
 Use Solder Mask Defined (SMD) for thermal land

Inspection:

QFN's do not have "Toe fillets", so X-ray should be used for inspection.

Thermal Via:

Thermal via grid should be placed on devices to maximize QFN thermal performance.

Source:

<http://www.intersil.com/data/tb/TB389.pdf>

http://assembly.idt.com/pdf_dwg/plastic/P4171.pdf