

PCBFast

Local Factory Capabilities

Revision 1.3 – October 2016

Standard Manufacturing Panel Sizes

- 1 Layer NPTH & 2 Layer PTH Small – usable area 180 x 260mm
- 1 Layer NPTH & 2 Layer PTH Standard – usable area 415 x 260mm
- Multi-Layer PTH Standard – usable area 380 x 250mm

Regular Lead times & Order Cutoff Time

- 1 day service: 1 Layer NPTH/2 Layer PTH Small – order cutoff 2pm (VIC) the afternoon before Day 1
- 2 day service: 1 Layer NPTH/2 Layer PTH Small – order cutoff 8am (VIC) Day 1
- 3 day service: 1 Layer NPTH/2 Layer PTH Small – order cutoff 8am (VIC) Day 1
- 3 day service: Multi-Layer PTH - order cutoff 2pm (VIC) the afternoon before Day 1
- 4 day service: All layer counts – order cutoff 9am (VIC) Day 1
- 5 day service: All layer counts – order cutoff 10am (VIC) on Day 1
- 8+ day service: All layer counts – order cutoff 10am (VIC) on Day 1

Design & Test

- Maximum 1 – 2 layer PCB size 550 x 400mm
- Maximum Multilayer PCB size 450 x 300mm
- Minimum finished 1 – 2 layer PCB thickness 0.25mm
- Minimum finished 4 layer PCB thickness 0.5mm
- Minimum finished 6 layer PCB thickness 0.6mm
- Minimum annular ring 6 mil / 0.15mm
- Minimum Pad size equals hole/via size + 0.3mm (e.g. 0.3mm via has 0.6mm pad)
- Minimum 6 mil / 0.15mm Trace Width (1 oz copper finish)
- Minimum 6 mil / 0.15mm Air Gap (1 oz copper finish)
- Minimum 8 mil / 0.2mm Trace Width (2 oz copper finish)
- Minimum 8 mil / 0.2mm Air Gap (2 oz copper finish)
- Minimum 14 mil / 0.35mm internal drill to copper clearance for Vias
- Minimum 18 mil / 0.45mm internal drill to copper clearance for Component Holes
- Minimum 20 mil / 0.5mm copper to board edge clearance
- Minimum 40 mil / 1.0mm copper to v-groove line clearance
- IPC 6012 Class 2
- Flying Probe Gerber Electrical Test

Multi-Layer PCBs

- 4, 6 & 8 Layers – through hole vias only, Blind and / or Buried Vias are not available
- 140°Tg Laminate & prepregs only – high temperature materials not available

PCBFast

Local Factory Capabilities

Revision 1.3 – October 2016

Factory Certification

- Underwriters Laboratories Inc (UL) Manufacturer File E234060

Drill & Rout

- X-Ray Drill Optimization
- NC Drill – 0.25mm to 6.20mm Finished Size
- NC Rout – Min 0.6mm Finished PTH Slot Width
- NC Rout - Min 0.8mm NPTH Slot width
- NC Rout - 2.00mm Standard Final Rout
- NC V-Groove
- NC Jump V-Groove – 50mm clearance run in & run out required

Materials

- Rigid FR4 Laminate (135 – 140 Tg only)
- Limited processing of other materials – customer supplied only

Plate

- 4oz / 140um Maximum Average Surface Copper, subject to min trace/air gaps of 12mil / 0.3mm
- Plated Edges
- Immersion Silver (RoHS Lead Free)
- 0.03µm Immersion Gold over 0.03µm Immersion Silver (RoHS Lead Free)
- HAL Solder (tin/lead)
- 0.5mm minimum for Castellated Holes (plated half holes)

Solder Mask

- Liquid Photo-imageable (LPI) Ink Satin Finish – Standard Green with Red, Black, Blue & White option
- Via Tenting using standard LPI – biggest via tented 0.4mm
- Via Plugging Not Available

Silkscreen Overlay

- White Ink Jet Print – subject to minimum physical text height of 0.8mm & line width 0.15mm
- LPI – White, Yellow, Black, Red, Blue or Green & subject to design minimum physical text height of 0.5mm & line width 0.125mm

Tolerances

- Air Gap / Trace: +/- 20%
- Solder mask Clearance over PAD size: 5 mil / 0.125mm
- Board thickness: +/- 10%
- Route: +/- 0.2mm