

Technology - HF-Printed Circuits

For an up to date version, please visit www.leiton.de



Options and Characteristics - Rigid Printed Circuits - PCBs	Online calculation//Standard	on explicit enquiry
Quantity	1 piece up to 0,4m ² total area	from 1 piece
Layer quantity	1 to 2 layers	up to 8 layers
Materialdicke RO4003C (1- and 2 layers)	0,203mm; 0,3mm; 0,41mm; 0,51mm; 0,81mm; 1,524mm;	0,203mm; 0,305mm; 0,406mm; 0,508mm; 0,813mm; 1,524mm
Materialdicke RO4350B (1- und 2 layers)	0,168mm; 0,254mm; 0,508mm; 0,762mm; 1,524mm	0,168mm; 0,254mm; 0,338mm; 0,422mm; 0,508mm; 0,762mm; 1,524mm
Materialdicke Panasonic Megtron6 (1- und 2 layers)	0,25mm; 0,74mm; 1,46mm	0,15mm; 0,20mm; 0,30mm; 0,40mm; 0,50mm; 0,925mm
Material thickness (4, 6 and 8 layers)	not possible	with above mentioned cores up to 4,0mm
Copper thickness (1 and 2 layers)	35µm	35µm, 70µm
Copper thickness (4, 6 and 8 layers)	not possible	outer: 35µm, 70µm / inner: 18µm, 35µm
Material colour	white	white
Maximum operating temperature (Rogers)	ca. 280° C	ca. 280° C
Maximum operating temperature (Megtron6)	ca. 170° C (TG 185° C)	ca. 170° C (TG 185° C)
Minimum operating temperature	ca. -40° C	ca. -40° C
Silk print layer	none, top, bottom, double sided	none, top, bottom, double sided
Solder mask colour	green, white, black, blue, red	green, white, black, blue, red and transparent (individual colours on exact colour value - RAL scale)
Silk print colour	white	black, blue, yellow, red
Via-Filling (no copper lid)	possible	possible
Electric test	possible (flying probe)	possible, also adapter
Plugging (with copper lid, e.g. for "via-in-pad technology")	not possible	possible
Peelable mask	not possible	top, bottom or double sided
Beveling	not possible	not possible
Surface finish	HAL lead free, immersion tin, immersion gold (ENIG and ENEPIG)	HAL lead free, immersion tin, immersion gold (ENIG and ENEPIG)
Finger gold	not possible	possible
Maximum printed circuit board size 1 and 2 layers PCB	430 x 275 mm ²	580 x 420 mm ²
Maximum printed circuit board size for multilayer PCBs	not possible	430 x 275 mm ²

Technology - HF-Printed Circuits

For an up to date version, please visit www.leiton.de



Minimum singulated circuit board size	3cm ² , smaller can be calculated but will be panelized	>1cm ² <3cm ² on enquiry
Minimum circuit board size in v-cut panels	5 x 5 mm ²	5 x 5 mm ²
Minimum circuit board size in routing panels	10 x 10 mm ² , or 1cm ²	10 x 10 mm ² , or 1cm ²
Minimum printed circuit board width	5mm	5mm
Lead time options 1 and 2 layers printed circuits	5WD, 7WD, 9WD, 12WD	2WD
Lead time options 4 to 8 layers multilayer printed circuits	not possible	10WD
Routing	always	always, except punching
V-cut	possible	possible
Jump-v-cut	possible	possible
Punching	not possible	possible
Counter-sink-holes	not possible	possible
Z-axis milling	not possible	possible
Multilayer special stack-up	not possible	possible

Panel Production - Rigid Printed Circuit Boards - PCBs	Online calculation/Standard	on explicit enquiry
V-cut panel	possible	possible
V-Cut - Routing panel (combination)	possible	possible
Multi panels (more than one layout per panel)	possible	possible
Panel setup (chosen by Leiton)	possible	possible
Panel setup (according to drawing)	possible	possible

PTH-drills (plated drills) - Rigid Printed Circuit Boards - PCBs	Online calculation/Standard	on explicit enquiry
Smallest drill 35µm (final diameter)	0,20mm, standard is 0,30mm	0,15mm
Smallest drill 70µm (final diameter)	not possible	0,20mm
Smallest annular ring 35µm	0,15mm	0,10mm
Smallest annular ring 70µm	not possible	0,12mm
Possible drill diameters	0,20mm to 5,5mm in 0,05mm steps	0,15mm to 5,5mm in 0,05mm steps
Drills >5,5mm	are routed	are routed

Technology - HF-Printed Circuits

For an up to date version, please visit www.leiton.de



Smallest drill distance 0,20mm to 2,00mm hole size (drill edge to drill edge)	0,40mm	0,30mm
Smallest drill distance 2,05mm to 5,50mm hole size (drill edge to drill edge)	0,50mm	0,40mm
Intersecting drills	not possible, will be routed	1,0mm to 2,0mm
Half-open PTH on PCB edge	not possible	possible

NPTH-drills (non-plated drills) - Rigid Printed Circuit Boards - PCBs	Online calculation/Standard	on explicit enquiry
Smalles drill size (final diameter)	0,60mm	0,30mm
Possible drill sizes	0,60mm to 5,5mm in 0,05mm steps	0,30mm to 5,5mm in 0,05mm steps
Copper clearance / distance to copper	0,20mm	0,20mm
Drills >5,5mm	are routed	are routed
Minimum drill-to-edge distance	0,50mm	0,50mm
Smallest drill distance 0,20mm to 2,00mm hole size (drill edge to drill edge)	0,40mm	0,20mm
Smallest drill distance 2,05mm to 5,5mm hole size (drill edge to drill edge)	0,50mm	0,50mm
Intersecting drills	not possible, will be routed	1,0mm to 2,0mm
NPTH drills in copper area (without clearance)	not possible (will be cleared from copper by min. 0,2mm)	on explicit requirement only

Blind Vias - Rigid Printed Circuit Boards - PCBs	Online calculation/Standard	on explicit enquiry
Smallest blind via (final diameter)	not possible	0,30mm
Smallest Aspect-Ratio	not possible	1
Smallest annular ring	not possible	0,125mm

Buried Vias - Rigid Printed Circuit Boards - PCBs	Online calculation/Standard	on explicit enquiry
Smallest buried via (final diameter)	not possible	0,20mm

Routing (non-plated) - Rigid Printed Circuit Boards - PCBs	Online calculation/Standard	on explicit enquiry
Inner routing NPTH	possible	possible
Smallest inner routing NPTH	0,70mm	0,70mm
Available routing diameters NPTH	up to 2,0mm in 0,10mm steps	up to 2,2mm in 0,10mm steps
Smallest radius (inner corners) NPTH	0,35mm	0,35mm

Technology - HF-Printed Circuits

For an up to date version, please visit www.leiton.de



Routing (plated) - Rigid Printed Circuit Boards - PCBs	Online calculation/Standard	on explicit enquiry
Inner routing PTH	partly possible	possible
Smallest inner routing PTH (final diameter)	0,60mm	0,60mm
Edge plating (outer edge)	not possible	possible
Special routing paths with plating (inner)	partly possible	possible
Available routing diameters PTH (final diameter)	up to 1,9mm in 0,10mm steps	up to 1,9mm in 0,10mm steps
Smallest radius (inner corner, final) PTH	0,30mm	0,30mm
Smallest annular ring	0,20mm	0,20mm

Copper Layers (outer) - Rigid Printed Circuit Boards - PCBs	Online calculation/Standard	on explicit enquiry
Smallest trace 18µm	not possible	0,07mm
Smallest trace 35µm	0,10mm; 0,125mm or 0,15mm	0,08mm
Smallest trace 70µm	not possible	0,15mm
Smallest trace-to-trace distance 18µm	not possible	0,07mm
Smallest trace-to-trace distance 35µm	0,10mm; 0,125mm or 0,15mm	0,08mm
Smallest trace-to-trace distance 70µm	not possible	0,15mm
Smallest drill pad	0,50mm bei 0,20mm Bohrungen, Standard ist 0,60mm	0,38mm
Smallest copper clearance to inner routing	0,30mm	0,25mm or 0,0mm (plated)
Smallest copper clearance to board edge (routing)	0,30mm	0,25mm or 0,0mm (plated)
Smallest copper clearance to board edge (v-cut)	0,50mm	0,40mm

Copper Layers (inner) - Rogers-Multilayer Printed Circuit Boards - PCBs	Online calculation/Standard	on explicit enquiry
Smallest trace 18µm	not possible	0,09mm
Smallest trace 35µm	not possible	0,10mm
Smallest trace 70µm	not possible	0,15mm
Smallest trace-to-trace distance 18µm	not possible	0,08mm
Smallest trace-to-trace distance 35µm	not possible	0,08mm
Smallest trace-to-trace distance 70µm	not possible	0,15mm

Technology - HF-Printed Circuits

For an up to date version, please visit www.leiton.de



Smallest drill-pad diameter	not possible	0,40mm
Smallest copper clearance to outer edges (routed)	not possible	0,25mm
Smallest copper clearance to inner edges (routed)	not possible	0,25mm
Smallest copper clearance to drills	not possible	0,25mm

Solder Mask - Rigid Multilayer Printed Circuit Boards - PCBs	Online calculation/Standard	on explicit enquiry
Smallest solder mask web (straight)	 0,12mm	0,10mm
Smallest solder mask web (round)	 0,075mm	0,05mm
smallest solder mask size around copper	 0,075mm	<0mm
Smallest text lines	0,25mm	0,20mm

Silk Print - Rigid Printed Circuit Boards - PCBs	Online calculation/Standard	on explicit enquiry
Smallest lines	0,20mm	0,15mm
Smallest distance between lines	0,20mm	0,15mm
Minimum clearance to copper pads	0,20mm	0,15mm

Carbon Print - Rigid Printed Circuit Boards - PCBs	Online calculation/Standard	on explicit enquiry
Smallest pad-to-pad distance	not possible	0,50mm

Tolerances, Values, Marks & Norms - Rigid Printed Circuit Boards - PCBs	Online calculation/Standard	on explicit enquiry
Max. offset drill centre to centre of reference	0,07mm	0,05mm
Max. offset solder mask / copper structures	0,12mm	0,075mm
Finished drill sizes PTH (0,55 to 3mm)	+/-0,10mm	+/-0,05mm
Finished drill sizes PTH (>3mm)	+/-0,12mm	+/-0,10mm
Finished drill sizes NPTH (up to 6mm)	+/-0,10mm	+/-0,05mm
Finished drill sizes NPTH (>6mm)	+/-0,12mm	+/-0,10mm
Outline	+/-0,20mm	+/-0,10mm
Max. offset outline/copper structures	+/-0,15mm	+/-0,10mm

Technology - HF-Printed Circuits

For an up to date version, please visit www.leiton.de



V-cut depth	+/-0,30mm	+/-0,20mm
Z-axis milling	not possible	+/-0,05mm
V-Cut position to copper structures	+/-0,25mm	+/-0,15mm
Etch tolerance copper thickness 18µm	not possible	+0/-0,02mm
Etch tolerance copper thickness 35µm	+0/-0,03mm	+0/-0,03mm
Etch tolerance copper thickness 70µm	not possible	+0/-0,05mm
Material thickness tolerances	<=1,0mm: +/-15% >1,0mm: +/-10% 1,55mm: 1,6mm +/-10%	differs, please enquire
Copper thickness tolerances	+20% / -15%	+/-10%
Immersion tin thickness	>=0,7µm	>=1,0µm
HAL lead free thickness	>= 8- bis 10µm, Kanten >0,5µm	>= 8- bis 10µm, Kanten >0,5µm
HAL lead thickness	not possible	>= 8- bis 10µm, edges >0,5µm
ENIG - Immersion gold for soldering (nickel thickness)	2,5µm to 5µm	3µm to 6µm
ENIG - Immersion gold for soldering (gold thickness)	0,05µm to 0,075 µm	0,07µm to 0,12 µm
ENEPIG - Immersion nickel-palladium-gold for gold-wire bonding (nickel thickness)	4µm to 8µm	4µm to 8µm
ENEPIG - immersion nickel-palladium-gold for for gold-wire bonding (palladium thickness)	0,1µm to 0,3µm	0,1µm to 0,3µm
ENEPIG - imersion nickel-palladium-gold for gold-wire bonding (gold thickness)	0,02µm to 0,08µm	0,02µm to 0,08µm
Immersion gold for gold-wire bonding (nickel thickness)	not possible	3µm to 6µm
Immersion gold for gold-wire bonding (gold thickness)	not possible	0,4µm to 6µm
Immersion gold for aluminum-wire bonding (nickel thickness)	2,5µm to 5µm	3µm to 6µm
Immersion gold for aluminum-wire bonding (gold thickness)	0,05µm bis 0,075 µm	0,07µm bis 0,12 µm
Electrolytic connector gold - soft, bonding possible (nickel thickness)	not possible	4µm to 8µm
Electrolytic connector gold - soft, bonding possible (gold thickness)	not possible	0,3µm to 5µm
Electrolytic connector gold - hard, bonding NOT possible (nickel thickness)	not possible	4µm to 8µm
Electrolytic connector gold - hard, bonding NOT possible (gold thickness)	not possible	0,8µm to 5µm
Wet solder laquer thickness	>15µm	>15µm
Copper thickness in plated hole, traces 35µm	minimum 18µm	minimum 20µm
Copper thickness in plated hole, traces 70µm	not possible	minimum 20µm

Technology - HF-Printed Circuits

For an up to date version, please visit www.leiton.de



Winding	max. 1%	max. 0,5%
Warping	max. 1%	max. 0,5%
Beveling angle	not possible	not possible
Base materials RoHS-compliance	yes, always	yes, always
Surface finish RoHS-compliance	yes, always	always, unless "HAL lead" is explicitly required
IPC-norm	IPC-A-600 - Class 2	IPC-A-600 - Class 1, 2 or 3
UL-approval of printed circuits (UL number, logo, date code)	not possible	not possible
UL-approved base material	yes, always	yes, always
Insertion of date code	possible, please mention in enquiry	possible, please mention in enquiry
Insertion of supplier logo (Leiton)	possible, please mention in enquiry	possible, please mention in enquiry
DIN EN ISO 9001 certification of printed circuit board production	yes	yes
DIN EN ISO 14001 certification of printed circuit board production	yes	yes
DIN EN ISO 16949 certification of printed circuit board production	no	possible