



Característica	Capacidad
Material Type	
Flex Material	Copper thickness 12µm; resin thickness between 65 and 100µm (Shengy material)
LDPP (Laser Drill PP)	IT-180A 1037 (2 mils) and IT-180A 1086 (3 mils). ITEQ Material
FR-4 (standard Tg)	S1141
FR-4 Halogen Free (standard Tg)	S1155
FR-4 Halogen Free (alto Tg)	S1165
FR-4 (High Tg)	FR408, IT180A, PCL-370HR, N4000-13, N4000-13S
Hydrocarbon ceramic high frequency material	Rogers4350, Rogers4003, 25FR, 25N
PTFE high frequency material	ROGERS, TACONIC, ARLON, NELCO
PTFE Bonding film	RO3001 (1.5 mils), HT1.5 (1.5 mil), CuClad6700 (1.5 mil)
Surface Finish type	
Lead Free	HAL Flash Gold (for base copper ≤ 35 µm) ENIG (Electroless Niquel Inmersion Gold) Inm. Ag (Chemical Silver) Inm. Sn (Chemical Tin) OSP (Organic Soldering Preservative) Hard gold Gold fingers (with any of previous finishings)
Surface plating or coating thickness	
Tin thickness (HAL)	2 - 40 µm
Flash Gold	Ni: 3 - 5 µm; Au ≥ 0,025 µm
ENIG	Ni: 3 - 5 µm; 0,05 µm ≥ Au ≥ 0,10 µm
Inm.Sn	Sn ≥ 1,0 µm
Inm.Ag	0,10 - 0,30 µm
OSP	0,20 - 0,30 µm
Hard gold	Au ≤ 2,5 µm
Carbon Thickness	100 - 350 µm
Solder Mask thickness	10 - 18 µm (on copper area) and 5 - 8 µm (on via pad and line corner)
Peelabls solder mask thickness	200 - 500 µm
Hole	
Hole size (final diameter = d) for mechanical drilling	0,10 ≤ d ≤ 6,5 mm For PTFE material: d ≥ 0,25 mm For blind / buried via holes: d < 0,30 mm For plugged holes with resin: 0,10 mm ≤ d ≤ 0,40 mm
Hole size (final diameter = d) for laser drilling	For blind holes plugged with resin: 0,075 mm ≤ d ≤ 0,15 mm For blind holes plugged with copper: 0,075 mm ≤ d ≤ 0,127 mm
Aspect Ratio = thickness "e"/diametre "d"	e ≤ 0,60 mm (for d = 0,10 mm) e ≤ 1,20 mm (for d = 0,15 mm) Aspect ratio ≤ 16 (for d > 0,20 mm)
Hole position tolerance	± 3 mil
PTH size tolerance	± 3 mil
NPTH size tolerance	± 2 mil (distribution: +0 /-2 mil o + 2 /-0 mil)
Relationship between hole size (finished) for plugging with resin and board thickness	Thickness ≤ 1,6 mm : 0,15 mm Thickness ≤ 2,4 mm : 0,20 mm Thickness ≤ 2,8 mm : 0,25 mm Thickness ≤ 3,2 mm : 0,30 mm



Característica	Capacidad
Minimum laser drill hole size	Depth ≤ 65 μm: 0,10 mm Depth ≤ 100 μm: 0,13 mm
Countersink hole	Standard. For diameter ≤ 3,175 mm: Angle 130° Standard. For diameter 3,175 - 6,5 mm: Angle 165° Special. For diameter 0,30 - 10 mm: Angle 82°, 90° and 120°
Countersink angle tolerance	± 10°
Countersink hole size tolerance	± 0,20 mm
Countersink depth tolerance	± 0,15 mm
Slot size tolerance	± 0,15 mm
Depth tolerance for blind NPTH slot routing	± 0,10 mm
Minimum pad size for laser drilling	Drill depth ≤ 65 μm : 10 mil Drill depth ≤ 100 μm : 11 mil
Minimum pad size for mechanical drilling	14 mils (via 8 mil, Base Cu 17 - 35 μm) 20 mils (via 8 mil, Base Cu 70 μm) 24 mils (via 8 mil, Base Cu 70 μm)
Minimum pad size for BGA	7 mils
Pad size tolerance	5% / -10%
Conductor width and space capability	
Inner layers	3/3 mils (Base Copper 12 and 17 μm) 3/4 mils (Base Copper 35 μm) 5/5 mils (Base Copper 70 μm) 6/7 mils (Base Copper 105 μm) 7/11 mils (Base Copper 140 μm) 10/16 mils (Base Copper 170 μm)
Outer layers	3/3 mils (Base Copper 12 μm) 3,5/3,5 mils (Base Copper 17 μm) 4,5/5 mils (Base Copper 35 μm) 6/8 mils (Base Copper 70 μm) 8/14 mils (Base Copper 105 μm) 10/16 mils (Base Copper 140 μm) 12/20 mils (Base Copper 170 μm)
Line width tolerance	≤ 10 mil : ± 1,0 mil > 10 mil : ± 1,5 mil
Space	
Minimum gap between hole wall to line (blind and buried via PCB)	9 mil
Minimum gap between hole wall to line (none blind and buried via PCB)	≤ 8 layers: 6 mil ≤ 14 layers: 8 mil ≤ 28 layers: 9 mil
Minimum gap between laser hole to line (HDI pcb)	6 mil
Minimum gap between outline and pattern for no copper exposure after routing	8 mil
Minimum gap between hole wall and hole wall (same net)	8 mil

**ZVBELZV S.L.**

CIRCUITOS IMPRESOS - PRINTED CIRCUIT BOARDS

Capability for Outsourcing Partner

Característica	Capacidad
Minimum gap between pads for immersion gold	4 mil
Minimum gap between gold fingers	6 mil
Minimum gap between pads for HAL	7 mil (10 mil on large copper area)
Minimum gap between peelable mask and pad	16 mil
Minimum gap between legend and pad	6 mil
Minimum gap between carbon pads	15 mil
Other	
Minimum core thickness	No blind neither buried via pcb: 0,05 mm Blind and buried via pcb: 0,13 mm
Number of layers	2 - 40
Thickness	0,13 - 7,0 mm
Maximum board size	890 x 584 mm
Register tolerance between layers	≤ 5 mil
PCB thickness tolerance	Thickness ≤ 1,0 mm : ± 0,10 mm Thickness > 1,0 mm : ± 10%
Impedance tolerance	± 5 Ω (< 50 Ω) ± 10% (≥ 50 Ω); can be ± 5% under request
Outline dimension tolerance	± 0,1 mm
Outline location tolerance	± 0,1 mm
Minimum warpage	± 0,1%
Maximum finished copper on inner layers	350 μm
Minimum isolation thickness between layers	2 mils (only for Base Copper 17 μm)
Minimum legend width	4 mils
V-Cut angle tolerance	± 5°
V-Cut symmetrical tolerance	± 4 mils
V-Cut rest thickness tolerance	± 4 mils
Outline machining	Routing, V-Cut
Minimum soldermask bridge width	4 mils for green solder mask; 5 mils another soldermask colour
Solder mask colours	Green, yellow, black, blue, red, white
Legend colours	White, yellow, black
Angle tolerance of Gold Finger	± 5°
Maximum test voltage	500 volts
Maximum test current	200 mA

REMARKS

Information according to manufacturer's Technical Data Sheets