ERABOM PCB Capabilities

Complete Manufacturing Specifications & Process Parameters

Process Item	Scope / Value	Details
Layers	1-24 layers	Number of signal layers in design files. Supports 1-24 layers through-hole boards, HDI blind/buried vias, half-holes, etc.
Solder Mask Ink	Guangxin Ink / Sun Ink / Rongda Ink	Green solder mask: Guangxin / Rongda series; White solder mask: Sun 2000 series
Board Material	FR-4 / Aluminum / Copper / High TG	FR-4 materials (Shengyi, Nanya, Kingboard, Guoji), Aluminum/Copper base materials (Lianmao/Guoji GL11 thermal conductivity 1.0W)
Max Board Size	600 x 400 mm	Standard size 520 x 415 mm; boards larger than this are considered extra-long and require special evaluation
Outline Dimension Tolerance	±0.10 mm	CNC outline tolerance ±0.10 mm; V-cut board outline tolerance ±0.10 mm
Board Thickness Range	0.2 – 8.0 mm	Multilayer boards require lamination structure reference; special materials require procurement
Board Thickness Tolerance (t ≥ 1.0mm)	±10%	Note: Generally positive tolerance due to plating/mask process
Board Thickness Tolerance (t < 1.0mm)	±0.1 mm	Note: Generally positive tolerance due to plating/mask process
Impedance Tolerance	±5Ω or ±10%	Optimized per customer impedance requirements; if not achievable, will communicate
Minimum Line Width	3 mil (0.075 mm)	4 mil is standard; 3 mil depends on board design and needs optimization
Minimum Line Spacing	3 mil (0.075 mm)	4 mil is standard; 3 mil depends on board design and needs optimization
Warpage	Max 0.2%	Standard boards 0.75%; HDI or asymmetric boards: 1%; Military grade: 0.2%-0.3%
Finished Outer Copper Thickness	35μm–210μm (6 OZ)	1 OZ = 35μm, 2 OZ = 70μm
Finished Inner Copper Thickness	18μm–140μm (4 OZ)	Inner layers above 3 OZ require line width/spacing evaluation
ENIG Thickness	1U"–3U"	U" (microinch) imperial unit; 1U" = 0.0254 μm
Drill Hole Diameter	0.10 – 6.3 mm	0.10mm laser drill for HDI; mechanical min 0.15mm; standard min 0.2mm
Annular Ring (Single Side)	≥0.153 mm (6 mil)	Minimum single side annular ring is 0.153 mm
Finished Hole Diameter (Machine Drill)	0.20 – 6.5 mm	Finished hole smaller than drill hole due to plating
Hole Diameter Tolerance (Machine Drill)	±0.05 mm	Example: design 0.6mm, finished 0.55–0.65mm acceptable
Solder Mask Type	Photosensitive Ink	Most used; thermosetting ink for low-end single-sided boards
Minimum Character Width	≥0.125 mm	Smaller may cause unclear characters on finished board
Minimum Character Height	≥25 mil (0.635 mm)	Smaller may cause unclear characters on finished board
Trace to Board Edge Clearance	≥0.3 mm (12 mil)	For routed boards: min 0.3mm; for V-cut panel: ≥0.4mm to V-cut center
Panelization: Gapless Panel Gap	0 mm	Gapless panelization, zero gap between boards
Panelization: Gapped Panel Gap	≥1.6 mm	Minimum gap for gapped panelization for easier routing
Pads Copper Fill Method	Hatch Fill	Manufacturer uses subtractive copper fill

Process Item	Scope / Value	Details
Slots in PADS Software	**Draw in Drill Drawing Layer**	If many non-metalized slots exist, draw in Drill Drawing layer
Open Window Layer in Protel/DXP	**Solder Layer**	Not paste layer — paste layer is not processed
Outline Layer in Protel/DXP	**Keepout or Mechanical Layer**	Only one outline layer allowed per file; delete unused outline layers
Half Hole Process	**Min half hole diameter 0.5 mm**	Special process; panelization method should be noted
Solder Mask Bridge	**0.2 mm**	For IC pins spacing, ensure >0.2mm for solder mask bridges
Gold Fingers	**45° chamfer, 0.5mm remaining**	Chamfer tolerance ±5°, remaining thickness tolerance ±0.13 mm
HDI Capability	**Up to 3-level blind/buried vias**	1st, 2nd, 3rd order blind/buried vias; price and lead time increase accordingly