



Design for Manufacturing Checklist

Production-grade rules for fabricating with FQC - Rayong, Thailand

This checklist summarizes the design rules enforced by FQC's production line in Rayong, Thailand. Values below are volume production specs, not bleeding-edge one-offs - every board we ship holds these tolerances. Submit designs following these rules to avoid DFM hold and accelerate your quote.

Questions outside this envelope? Contact marketing@fqc.co.th or submit a Gerber for free DFM review at fqc.co.th/quote.

Max layers	100	Min trace/space	2 mil / 50 um
Aspect ratio	20:1	Impedance	+/- 5%
Min mech. drill	6 mil (0.15 mm)	Min laser via	3 mil (0.075 mm)
Max Cu (outer)	15 oz UL	Max Cu (inner)	13 oz UL

01 Clearances & Spacing

Minimum trace-to-trace	2 mil (0.05 mm)	production grade
Minimum trace-to-pad	3 mil (0.075 mm)	
Copper-to-edge clearance	8 mil (0.2 mm)	minimum
Drill-to-drill	10 mil (0.25 mm)	minimum
Pad-to-hole annular ring	4 mil (0.1 mm)	minimum

02 Vias & Drills

Minimum mechanical drill	6 mil (0.15 mm)	
Minimum laser via	3 mil (0.075 mm)	CO2 or UV
Maximum aspect ratio	20:1	mechanical through-hole
Microvia stack depth	up to 5 levels	5th-order HDI
Via-in-pad	supported	Cu filling + planarization

03 Copper Features

Minimum trace width	2 mil (0.05 mm)	
Minimum annular ring	2 mil (0.05 mm)	
Maximum outer copper	15 oz	UL-listed
Maximum inner copper	13 oz	UL-listed
Teardrops	recommended	via-to-trace joints
Thieving	auto-applied	low-density areas

04 Surface Finish Selection

ENIG	fine-pitch BGA	shelf life 12 mo+
ENEPIG	wire-bond + BGA	gold wire compatible
OSP	cost-sensitive	shelf life 3-6 mo
HASL (lead / lead-free)	legacy only	not for fine-pitch
Immersion Silver	RF, high-freq	
Gold Fingers	edge connectors	wear-resistant

05 Silkscreen & Legend

Minimum text height	1.0 mm	
Minimum line width	0.15 mm	
Over solder pads	avoid	shift text >0.2 mm clear
Two-color legend	supported	on request

06 Panelization

Minimum gap between boards	2.0 mm	
V-cut	0.3 mm depth	rectangular boards
Tab-rout	1.5-2.0 mm tabs	0.8 mm mouse-bite holes
Fiducial marks	3 per panel min	1 mm diameter
Maximum panel size	610 mm x 540 mm	

07 Impedance Control

Tolerance (standard)	+/- 5%	+/-10% available on request
Stackup requirement	full stackup	with Gerber submission
Reference plane	solid	not split or hatched
Pre-production coupon	included on panel	for TDR verification

08 Documentation Deliverables

Gerber X2 (preferred) or RS-274X	required	
ODB++ or IPC-2581	accepted	
Drill file	Excellon format	
BOM	MPN + alternates	
Assembly drawing	if EMS assembly requested	
Stackup spec	impedance-controlled boards	
README	special instructions	

Not sure if your design fits? Submit your Gerbers at fqc.co.th/quote for a free DFM review within 24 hours. Our engineering team will flag any feature outside the production-grade envelope and propose adjustments.