PCB Manufacturing & Capabilities

Know our capabilities & get your PCBs built fast

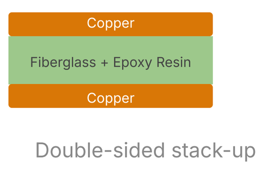
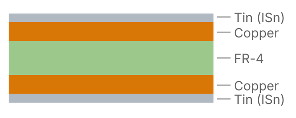
⚡ **QUICK SPECS AT A GLANCE**

LAYERS	SURFACE FINISH	MIN TRACK / SPACE	MIN DRILL	MAX DRILL
1 & 2 Layer	Immersion Tin	0.3 / 0.3 mm	0.4 mm	5.0 mm
HOLE TYPE	**MIN ANNULAR RING**	**SOLDERMASK**		
NPTH Only	0.45 mm	Green Only		

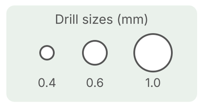
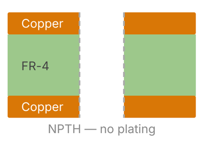
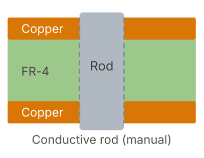
📐 **DESIGN RULES QUICK REFERENCE (FOR YOUR PCB SOFTWARE)**

Min. Track Width	0.3 mm	Min. Annular Ring	0.45 mm
Min. Clearance (Track–Track)	0.3 mm	Min. Drill Size	0.3 mm
Min. Pad–Track Clearance	0.3 mm	Max. Drill Size	5.0 mm
Min. Hole–Track Clearance	0.3 mm	Drill Step Increment	0.1 mm

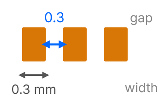
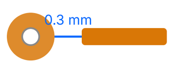
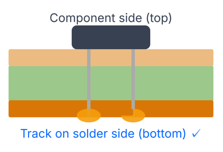
Layers & Material

Features	Capability	Description	Patterns
Layer Count	1 Layer (Single-sided) 2 Layers (Double-sided)	The number of copper layers in the PCB. Single-sided has copper on one side only; double-sided has copper on both sides.	
Base Material	FR-4	Standard fiberglass epoxy laminate (FR-4). Suitable for most general-purpose PCB applications.	
Surface Finish	Immersion Tin (ISn)	A flat, solderable surface finish where a thin layer of tin is chemically deposited on exposed copper pads. Provides excellent solderability and is suitable for fine-pitch components.	

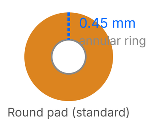
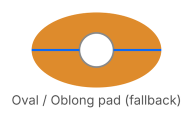
Drilling

Features	Capability	Description	Patterns
Drill Size Range	0.4 mm — 5.0 mm step: 0.1 mm	Drill sizes from **0.4 mm to 5.0 mm** in 0.1 mm increments only. Sizes outside this range or between increments are not available. Available: 0.4 · 0.5 · 0.6 · 0.7 · 0.8 · 0.9 · 1.0 · 1.1 · 1.2 · 1.3 · 1.4 · 1.5 · ... · 5.0 mm	
Hole Type	NPTH Only Non-Plated Through Holes	All drilled holes are **non-plated (NPTH)**. No copper is deposited inside the hole barrel. Suitable for mechanical mounting holes and through-hole component leads.	
Via Plating (PTH)	✕ Not Supported	Plated through holes (PTH) are **not available**. Electrical connections between the top and bottom copper layer cannot be made via plated vias. For double-sided boards, inter-layer connections must be made manually using a **conductive wire or rod** soldered through the hole.**Via sould be Not tented**	
Rectangular / Slot Holes	✕ Not Supported	Only **round edge holes/slot** are supported. Rectangular cutouts and milled slots are not available. All holes / slot must have Rounded edges.	—
Blind & Buried Vias	✕ Not Supported	Blind and buried vias are not available. All holes pass fully through the entire board from top to bottom.	—

Traces & Spacing

Features	Capability	Description	Patterns
Min. Track Width / Spacing	0.3 mm / 0.3 mm	The minimum copper track width and the minimum gap between any two copper features is **0.3 mm**. Designs with narrower traces or tighter spacing cannot be manufactured.	
Pad-to-Track Clearance	0.3 mm	Minimum clearance between a copper pad edge and any adjacent track must be at least **0.3 mm**.	
Via Hole to Track Clearance	0.3 mm	Minimum clearance between any hole edge and the nearest copper track must be at least **0.3 mm**.	—
Through-Hole Component Track Side	Opposite (solder) side	For through-hole components on double-sided boards, connecting tracks should be placed on the **opposite (bottom/solder) side** from where the component body sits. This keeps the component side clean and makes hand-soldering significantly easier.	

Pads & Annular Ring

Features	Capability	Description	Patterns
Min. Annular Ring	0.45 mm	The copper ring surrounding a drilled hole must be at least **0.45 mm** wide on all sides. This ensures reliable mechanical strength and a good solder joint for component leads.	
Pad Shape Fallback	Oval / Oblong Pad	If the **0.45 mm annular ring cannot be maintained** with a round pad (e.g. hole is too close to board edge or neighbouring pads), switch to an **oval / oblong pad shape**. This extends copper in one direction to achieve the required area without widening in both directions.	

Soldermask

Features	Capability	Description	Patterns
Soldermask Color	Green Only available color	A protective epoxy layer applied over copper to prevent oxidation and accidental solder bridges. Only **green soldermask** is available. No other colors are offered.	
Silkscreen / Legend	✕ Not Supported	No silkscreen or legend layer is available. Component labels, reference designators, polarity markers, and other text markings cannot be printed on the board.	—
Board Outline	✕ Not Included	Board outline / edge cut layer is not included in fabrication. The final board shape and dimensions must be communicated separately when placing an order.	—

Ready to Order Your PCB?

Send your Gerber files on WhatsApp and get a fast quote from YarsaPCB.

💬 WhatsApp — Get Instant Quote

📞 Contact Us: +977 9769490592