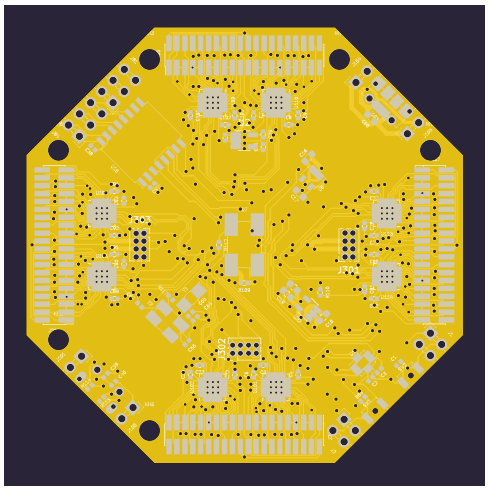
PRELIMINARY · IN DEVELOPMENT

PCB TECHNICAL INFORMATION · PRE-PRODUCTION

Four-Layer, Double-Sided PCB — Layout Data

Board-level data for the IronBCI-128 pre-production design: stack-up, copper layers, drill table and the Gerber file set. Values are read directly from the fabrication files.



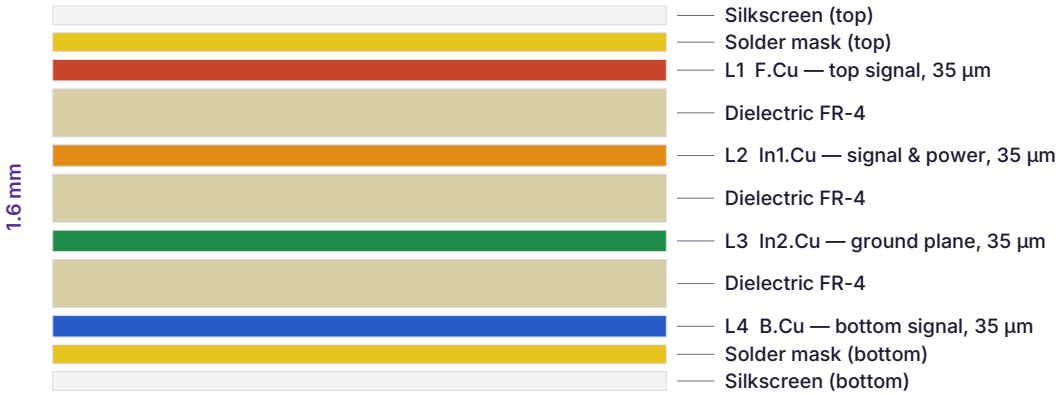
4	1.6 mm	70 × 70	0.2 mm	0.2 mm
Copper layers	FR-4 board thickness	mm octagon outline	Minimum track / gap	Minimum drill

01 FABRICATION SUMMARY

BOARD	
Base material	FR-4
Layer count	4 copper layers (signal / signal & power / ground / signal)
Finished thickness	1.6 mm
Outline	Regular octagon, 70.0 × 70.0 mm across flats, eight edges of 29.0 mm
Copper	35 µm (1 oz) on all four layers
DESIGN RULES	
Track width / gap	0.20 mm minimum on outer and inner layers
Inner plane clearance	0.508 mm region-to-track / region-to-region
Holes	608 plated (vias 0.2–0.4 mm, component holes 0.2–1.0 mm); 8 non-plated
Assembly	SMD components on both sides; through-hole headers and pads
DATA	
Design tool	KiCad 7 (Pcbnew), files generated 14 Nov 2026
Formats	Gerber X2 (RS-274X with attributes), Excellon drill, metric, absolute coordinates, board-centre origin

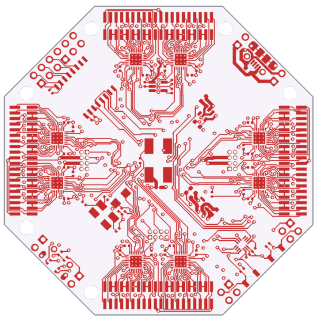
In development. Preliminary data; specifications may change. Not a medical device.

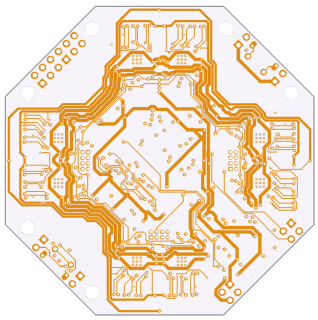
02 LAYER STACK-UP

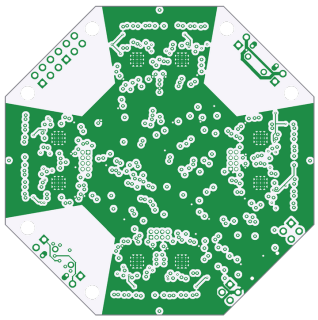


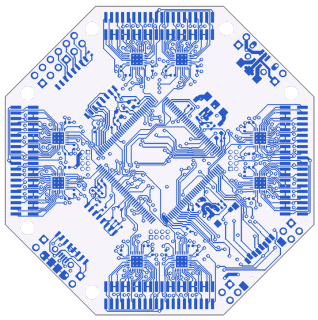
L1 and L4 carry the front-end parts and signal routing on each side. L2 distributes signals and power, and L3 is an almost unbroken ground plane under the whole board. Ask the fabricator for a standard 4-layer 1.6 mm build; the dielectric values in the job file are nominal.

03 COPPER LAYER ARTWORK

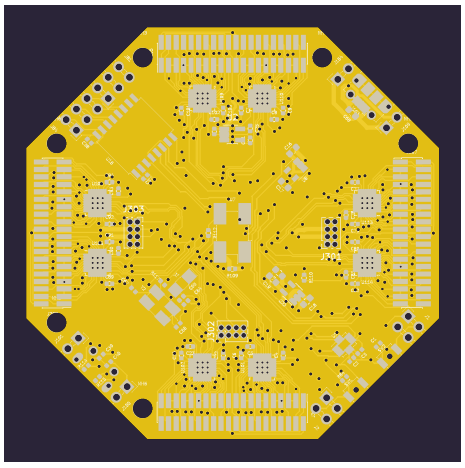

L1 F.Cu — top

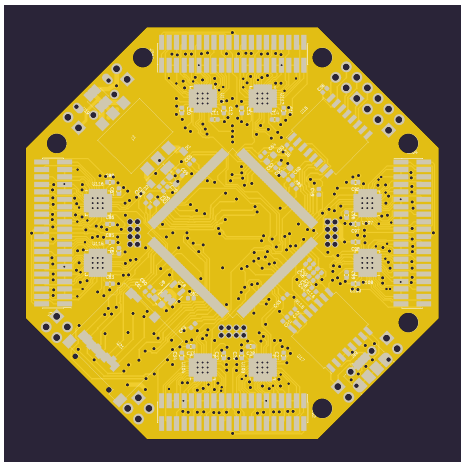

L2 In1.Cu


L3 In2.Cu — ground


L4 B.Cu — bottom

04 ASSEMBLY VIEWS


Top side


Bottom side (viewed from below)

In development. Preliminary data; specifications may change. Not a medical device.

05 DRILL TABLE

Diameter	Plating	Qty	Function
Ø 0.20 mm	Plated	152	Vias + component holes
Ø 0.30 mm	Plated	37	Vias
Ø 0.40 mm	Plated	359	Vias
Ø 0.60 mm	Plated	4	USB-C shell
Ø 0.65 mm	Plated	8	Header J302 (2 × 4)
Ø 0.70 mm	Plated	16	Headers J301, J303 (2 × 4)
Ø 0.85 mm	Plated	4	USB-C shell
Ø 1.00 mm	Plated	28	Through-hole pads (wire / test points)
Ø 0.80 mm	Non-plated	2	Locating holes
Ø 3.20 mm	Non-plated	6	M3 mounting holes (see IRB128-MD-002)
Total		616	

06 GERBER FILE SET

File	Layer	Function
IronBCI128-F_Cu.gtl	L1	Copper, top
IronBCI128-In1_Cu.g2	L2	Copper, inner 1
IronBCI128-In2_Cu.g3	L3	Copper, inner 2 (ground)
IronBCI128-B_Cu.gbl	L4	Copper, bottom
IronBCI128-F_Mask.gts / B_Mask.gbs	Top / Bot	Solder mask
IronBCI128-F_Silkscreen.gto / B_Silkscreen.gbo	Top / Bot	Legend
IronBCI128-F_Paste.gtp / B_Paste.gbp	Top / Bot	Solder paste (stencil)
IronBCI128-Edge_Cuts.gm1	—	Board profile
IronBCI128-PTH.drl / NPTH.drl	—	Plated / non-plated drill (Excellon)
IronBCI128-job.gbrjob	—	Gerber job file (stack-up, design rules)

07 RELATED DOCUMENTS

Doc. No.	Document	Contents
IRB128-DS-001	Preliminary Product Datasheet	Overview, preliminary specifications, family comparison
IRB128-MD-002	Mechanical & Dimensional Drawing	Outline, mounting holes, connector positions
IRB128-PCB-003	PCB Technical Information	Stack-up, copper layers, drill table, Gerber files

PRE-PRODUCTION DATA

These files describe the current IronBCI-128 pre-production board and may change before release. Subscribe at pieeg.com/store/ironbci-128 to be told when the final design and the assembled board are available.