

- [OSHPARK](#)
- [Services](#)
- [Support](#)
- [Services](#)
- [Support](#)
- [OSH Park Docs](#)
- [Fabrication Services](#)
- [4 Layer Prototype Service](#)



- [OSH Park Docs](#)
- [Fabrication Services](#)
- [4 Layer Prototype Service](#)

4 Layer Prototype Service

Pricing

\$10 per square inch, which includes three copies of your design. For example, a 2 square inch board would cost \$20 and you'd get three copies of your board. You can order as many copies as you want, as long as they're in multiples of three.

Turn Times

Orders will be shipped within 12 calendar days of ordering.

You can get a quote, approve a design, and pay for an order at [OSH Park](#).

We do not offer Super Swift Service for 4 layer boards.

Need more than 100 square inches of boards? Our [4 Layer Medium Run Service](#) is a less expensive option for larger orders.

Common Specs

These specs apply to all our PCB services.

Spec	Value
Manufactured in the United States	Yes
Lead Free compatible	Yes
RoHS Complaint	Yes
High Temp	Yes, 175 Tg or higher (see Material Specs)
PCB Finish	ENIG (Gold), compliant with IPC-4552
Soldermask Type	SMOBC (Soldermask Over Bare Copper), both sides
Silkscreen Type	High Res DLP, both sides

Stackup

Thickness	Layer	Tolerance
1mil (0.0254mm)	silkscreen	+/-0.2mil (0.00508mm)
1 mil (0.0254mm)	solder resist	+/-0.2mil (0.0051mm)
1.4 mil (0.0356mm)	1 oz copper	
6.7 mil (0.1702mm)	FR408 prepreg	+/- .67mil (0.017mm)
0.7 mil (0.0178mm)	0.5 oz copper	
47 mil (1.1938mm)	FR408 core	+/-4.7mil (0.1194mm)
0.7 mil (0.0178mm)	0.5 oz copper	
6.7 mil (0.1702mm)	FR408 prepreg	+/- .67mil (0.017mm)
1.4 mil (0.0356mm)	1 oz copper	
1 mil (0.0254mm)	solder resist	+/-0.2mil (0.0051mm)
1mil (0.0254mm)	silkscreen	+/-0.2mil (0.00508mm)

Material Specs

Spec	Value	
Substrate	190Tg FR408-HR	ISOLA FR408-HR Datasheet PDF
Board Thickness	63mil (1.6mm) nominal	
Dielectric	3.69 at 1GHz	
Soldermask Color	Purple	Mask Datasheet
Minimum soldermask web	4 mil (0.1016mm)	
Maximum soldermask alignment	3mil (0.0762mm)	Covers retraction, expansion, and shift
Silkscreen minimum line width	5 mil (0.127mm) (recommended minimum) 3 mil (0.0762mm) (short lines, text, graphics)	Silkscreen Datasheet
Maximum board size	16in (406.4mm) by 22in (558.8mm)	

Spec	Value
Minimum board size	0.25in (6.35mm) by 0.25in (6.35mm)

Copper Specifications

Spec	Value
Copper Layers	4
Copper Weight	1oz outer 1/2oz inner
Trace Spacing	5mil (0.127mm)
Trace Width	5mil (0.127mm)
Annular Ring	4mil (0.1016mm)
Board Edge Keepout	15mil (0.381) from nominal board edge
Via Plating Thickness	1mil (0.0254mm)

Drill Specifications

Spec	Value	
Minimum Annular Ring	4mil (0.1016mm)	
Minumum Drill Size	10mil (0.254mm)	
Minimum Slot Size	20mil (0.508mm) (drill slot only)	Additional information on slots
Drill Size tolerance	Max: +/- 2.5mil (0.0635mm) Typical: +/- 1.0mil (0.0254)	
Drill Positional Tolerance	Max: 2mil (0.0508mm) Typical: <1mil (0.0254mm)	
Via Tenting	Yes (filled hole and flat surface not guaranteed)	
Buried Via	No	
Blind Via	No	
Overlapping drills	Allowed, but not guaranteed. May result in missing or slotted holes. 5 mil (0.127mm) clearance is recommended between holes.	
Castellations	Allowed, but not guaranteed	Details and recommendations
Maximum Drill Size	None	Drill sizes above 250mil (6.35mm) will be fabbed, but with larger milling tolerances.

Layer Naming

We support the default file names for most design tools, and fab the internal layers in the order intended by the design tool.

If your tool or files requires configuration or manual renaming of layers, we suggest this pattern to ensure your layer order is interpreted correctly.

Layer	Suggested File Extension	
Layer 1	.GTL	Top or Front layer
Layer 2	.G2L	Internal layer (adjacent to Top)
Layer 3	.G3L	Internal layer (adjacent to Bottom)
Layer 4	.GBL	Bottom or Back layer

For additional info or other layers, see our [Suggested Naming Pattern](#)

Internal Plane Polarity

When submitting gerbers, we need the “positive” internal planes, meaning that lines represent copper, not the absence of copper.

Some CAD tools will generate the internal planes as power planes with “negative” polarity so the lines indicate where copper should be removed. To work around this, declare the internal planes as signal layers and use a copper pour to define the power plane.

Take a look at our [Positive and Negative Gerbers][invertedgerbers] page for more information.

- COMPANY
 - [Home](#)
 - [Blog](#)
 - [Shop](#)
- SERVICES
 - [Upload Your File](#)
 - [Prototypes](#)
- HELP
 - [Support](#)
 -
 - If you can't find what you're looking for, please contact us at support@oshpark.com
- CONNECT
 - [Shared Projects](#)
 - [Sign In / Sign Up](#)

Follow us

-
-
-
-
-
-

•
•
© Copyright 2017 Oshpark LLC | [Privacy](#)