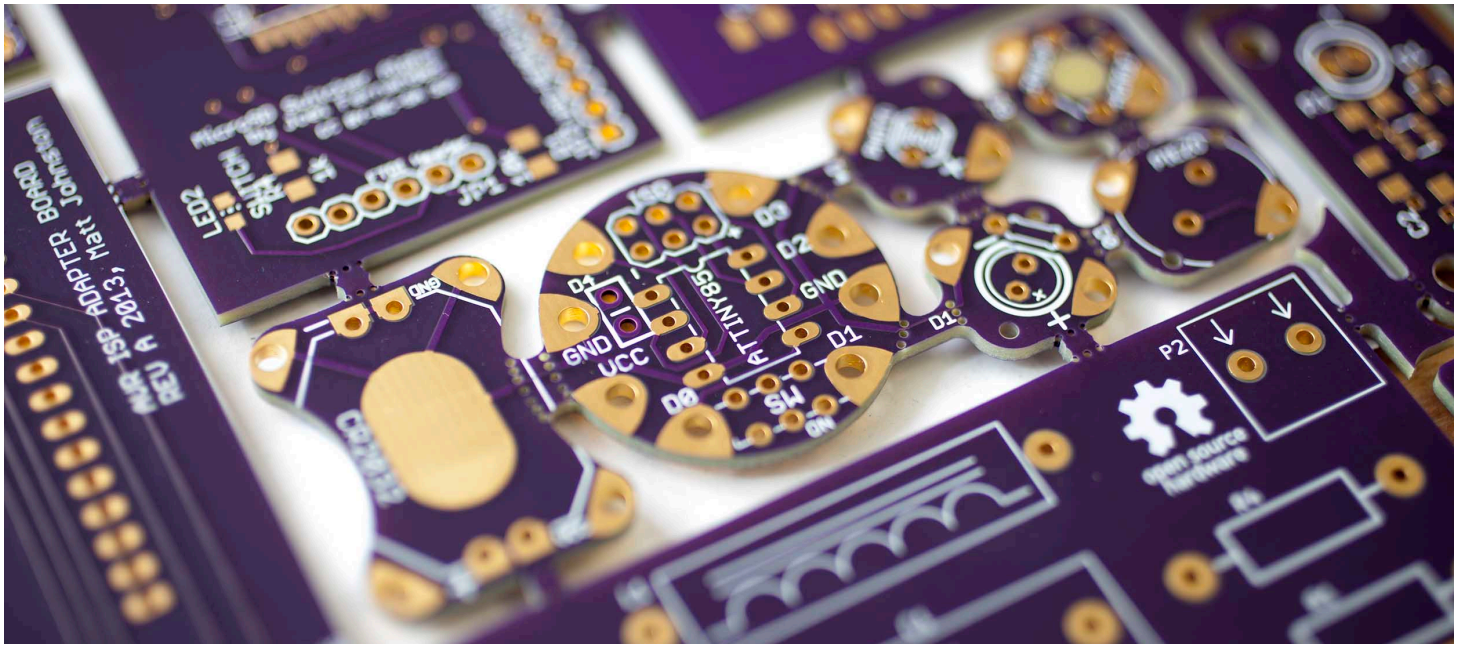


- OSH Park Docs
- Fabrication Services
- 4 Layer Prototype Service

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 ship within 9-14 calendar days of ordering.

You can get a quote, approve a design, and pay for an order at OSH Park.

For a faster turn time, you may be interested in our 4 Layer Super Swift Service.

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 Compliant	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	dk
0.6 mil (0.0152mm)	silkscreen	+/-0.2mil (0.00508mm)	
0.6 mil (0.0152mm)	solder resist	+/-0.2mil (0.0051mm)	
1.7 mil (0.0432mm)	1 oz copper clad+plated		
7.87 mil (0.1999mm)	FR408HR 2113 prepreg	+/-.797mil (0.0202mm)	3.61 at 1GHz
0.68 mil (0.0175mm)	0.5 oz copper clad		
39 mil (0.9906mm)	FR408HR core	+/-3.9mil (0.0991mm)	
0.68 mil (0.0175mm)	0.5 oz copper clad		
7.87 mil (0.1999mm)	FR408HR 2113 prepreg	+/-.797mil (0.0202mm)	3.61 at 1GHz
1.7 mil (0.0432mm)	1 oz copper clad+plated		
0.6 mil (0.0152mm)	solder resist	+/-0.2mil (0.0051mm)	
0.6 mil (0.0152mm)	silkscreen	+/-0.2mil (0.00508mm)	

Complete construction diagram

Impedence Constraint Table

Material Specs

Spec	Value	
Substrate	190Tg FR408-HR	ISOLA FR408-HR Datasheet PDF
Board Thickness	63mil (1.6mm) nominal	
Dielectric	3.61 at 1GHz	

Spec	Value	
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)	
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)
Plated Drill to Internal Layer Copper Clearance	10mil (0.254mm)

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	
Filled+Plated Vias (via-in-pad)	No	

Spec	Value	
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.
Plated Drill to Internal Layer Copper Clearance	10mil (0.254mm)	

Layer Naming

We automatically detect intended layer order from many tools, using common naming conventions and meta-data formats like Gerber X2. In most cases, renaming files should not be needed, and you can use your default export settings.

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.

Drill Internal Layer Clearance

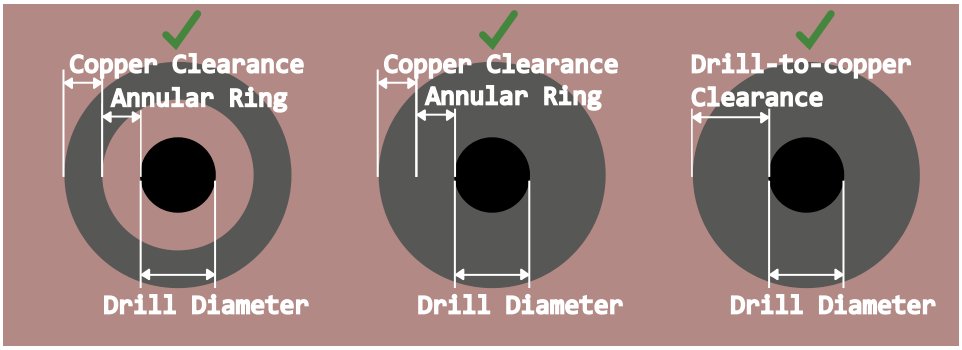
Some tools will often suppress annular rings on vias that do not connect to internal layers. However, this may not accurately capture required production specs, leading to inadvertent spec violations. While these violations do not always result in issues, they can lead to surprising defects and yield issues.

This DRC requirement is not always obvious inside design tools, with some tools defaulting to a violation, and not providing clear warnings. Care should be taken to ensure compliance. Tools handle this differently, but one of the following methods should help:

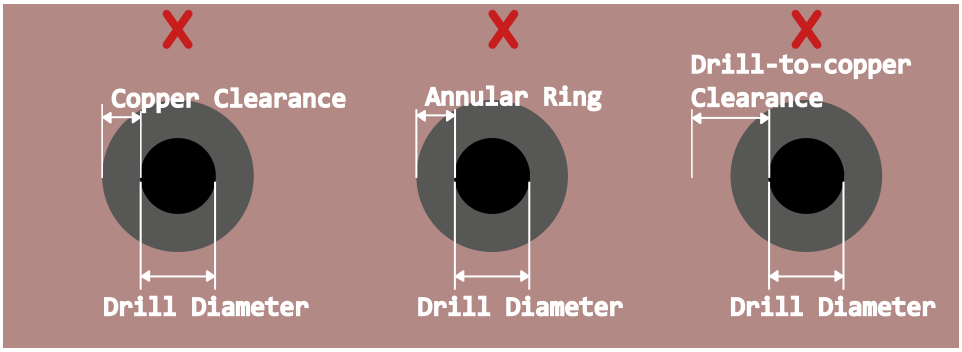
- Configuring the tool to always include annular rings, regardless of net connections
- Increasing ground plane clearance on internal layer copper pours

- Configuring custom net rules to increase isolation on internal layer copper pours.

Your design tool's documentation will assist with these steps.



Examples of drill configurations that comply with specs



Examples of drill configurations that violate specs

Changelog

25-JUL-2024 Corrected prepreg thickness from 7.96mil to 7.87mil to better match the boards as fabricated. This is a documentation correction, and does not reflect a production change.

17-JUN-2024 Added DRC item for vias passing through internal layers without generating annular rings.

Follow us

-
-
-
-
-
-
-

© Copyright 2023 OSH Park, LLC | Privacy