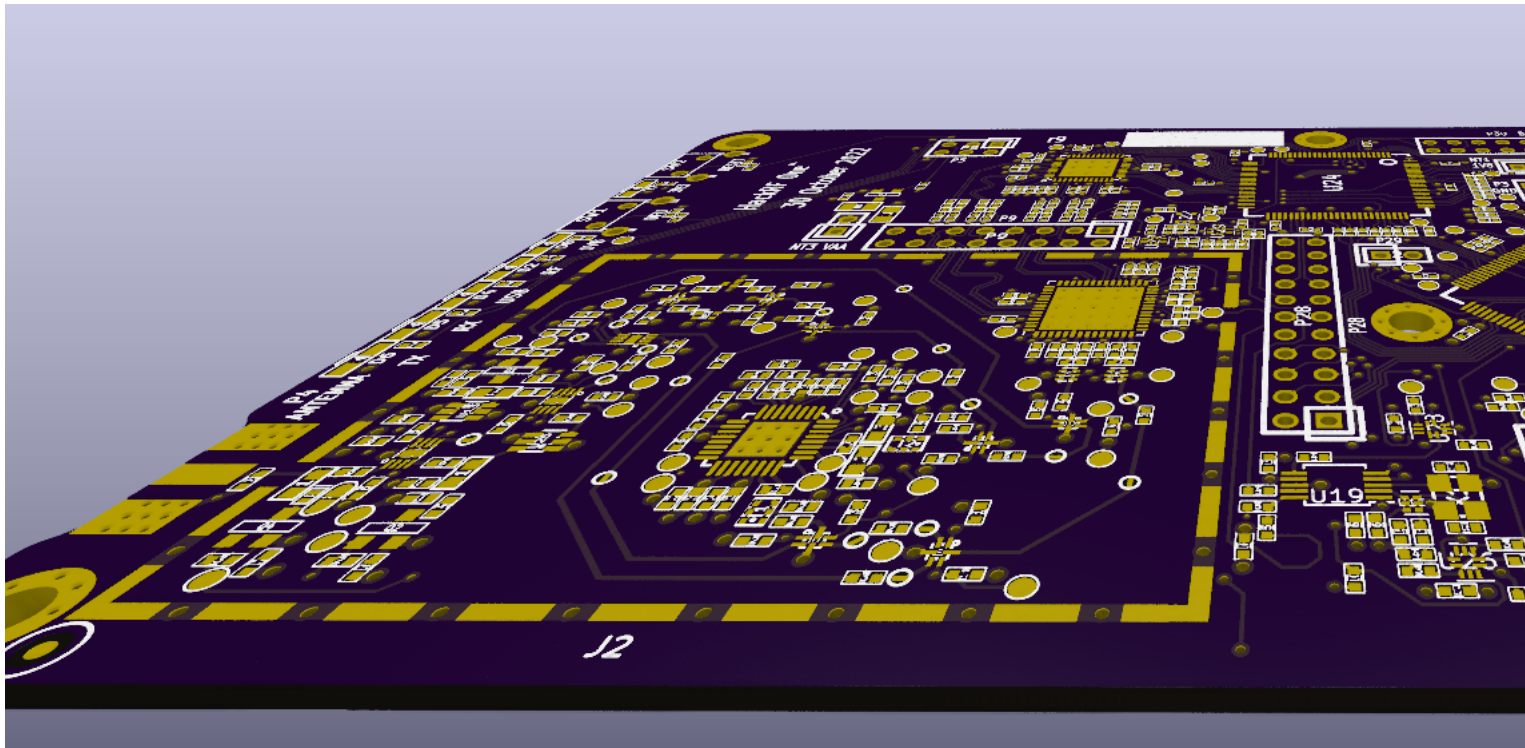- [OSH**PARK**](#)
- [Services](#)
- [Support](#)

- [Services](#)
- [Support](#)

- [OSH Park Docs](#)
- [Design Tool Help](#)
- [KiCad](#)
- [Design Rule Setup](#)

- [OSH Park Docs](#)
- [Design Tool Help](#)
- [KiCad](#)
- [Design Rule Setup](#)

Design Rule Setup

[KiCad](#)
[Open Source](#)

Table of Contents

- [Design Rules Editor](#)
- [Board Stackup > Physical Layers](#)
- [Board Stackup > Soldermask/Paste](#)
- [Design Rules > Constraints](#)
- [Design Rules > Net Classes](#)

Related Pages

- [Generating Gerbers](#)
- [KiCad](#)
- [Design Rule Setup](#)

Other Resources

- [Tutorials](#)
- [Kicad Forum](#)
- [Contextual Electronics](#)

The design rules describe manufacturing constraints, and help Kicad ensure that the design you create can be manufactured correctly.

Design Rules Editor

The manufactuing config dialogue box is found under `File > Board Setup`.

Most values provided below have conversions, but Kicad will automatically convert to your preferred units if you enter a unit indicator. EG, entering `0.01in`, `0.254mm`, or `10 mil` will provide the same end result.

Board Stackup > Physical Layers

This page lets you indicate the PCB layer count, and physical properties of the PCB.

`Copper Layers` should be set to `2`, `4`, or `6`. Using other layer counts will often result in issues in exporting files for production.

All other values: These depend on which service you need for production, and provided here: [OSH Park Services](#). Entering these values is only required if your design will use Kicad's DRC/DRU tools to calculate controlled impedence. During production, we will disregard user-provided values, and use the stackup indicated for the selected service.

Board Stackup > Soldermask/Paste

`Solder Mask Expansion` should usually be set to 2mil (0.0508mm), which pairs well with our production tolerances. This can be decreased as needed, but increasing this value is not recommended. For additional information, see [Understanding Stop Mask Expansion](#).

`Solder Mask Expansion Web Width` should usually be set to `0`.

`Tent Vias` is optional, but typically recommended. When checked, vias will be inaccessible for soldering; When unchecked, they will be exposed. Vias and through holes used for components will always be exposed as indicated in their footprint.

`Solder Paste` options will vary, and depend on your assembler or stencil manufacturer.

Design Rules > Constraints

These vary slightly based on production service.

Setting	2 Layer Services	4 Layer	6 Layer
Minimum Clearance	6mil (0.1524mm)	5mil (0.127mm)	5mil (0.127mm)
Minimum track Width	6mil (0.1524mm)	5mil (0.127mm)	5mil (0.127mm)
Minimum Connection Width	6mil (0.1524mm)	5mil (0.127mm)	5mil (0.127mm)
Minimum Annular Ring	5mil (0.127mm)	4mil (0.1016mm)	4mil (0.1016mm)
Minimum Via Diameter	20mil (0.508mm)	18mil (0.4572mm)	16mil (0.4064mm)
Copper to hole clearance	5mil (0.127mm)	5mil (0.127mm)	5mil (0.127mm)
Minimum Through Hole	10mil (0.254mm)	10mil (0.254mm)	8mil (0.2032mm)
Hole to hole clearance	5mil (0.127mm)	5mil (0.127mm)	5mil (0.127mm)
Minimum uVia diameter	20mil (0.508 mm)	18mil (0.4572 mm)	16mil (0.4064 mm)
minimum uVia Hole	10mil (0.254mm)	10mil (0.254mm)	8mil (0.2032mm)
Silkscreen Min Item Clearance	user preference		
Silkscreen Min Text Height	user preference		
Silkscreen Min Text Thickness	5 mil (0.127mm)	5 mil (0.127mm)	5 mil (0.127mm)

Design Rules > Net Classes

These can be configured to user preferences, and allow customized DRC depending on the type of signal you're routing.

The "Default" net class will use the minimum DRC values.

- 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](#)
- [Log in / Sign Up](#)

Follow us

-
-
-
-
-
-
-

© Copyright 2023 Oshpark LLC | [Privacy](#)