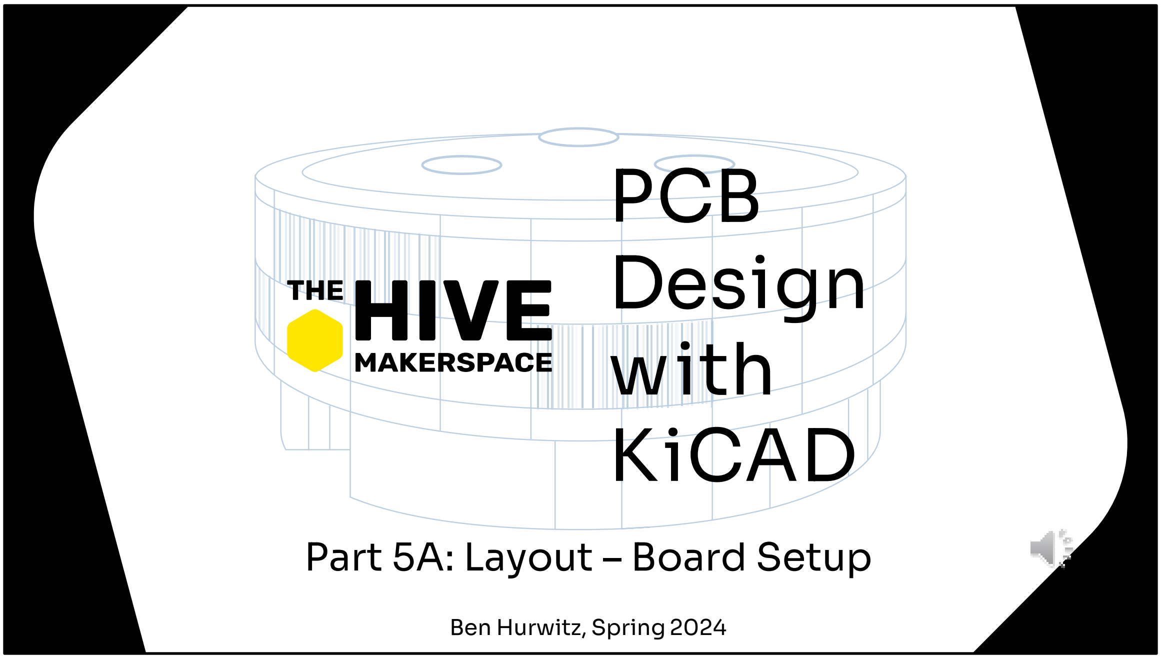


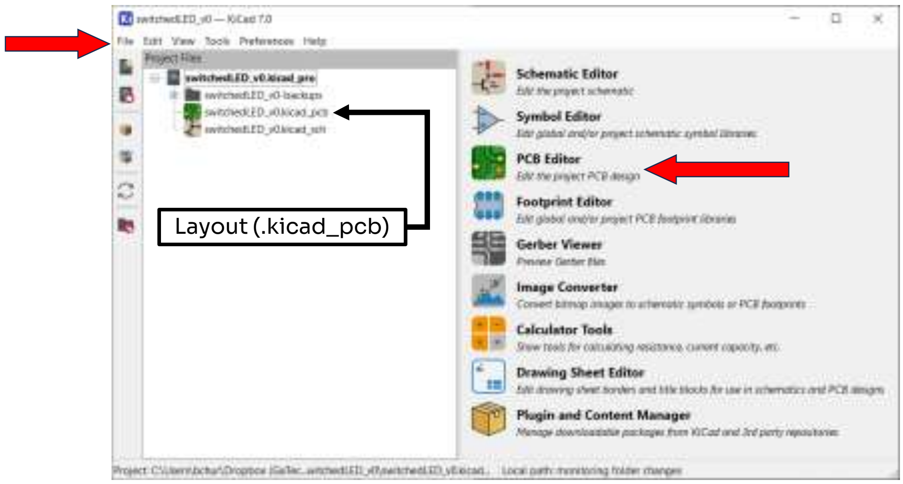
Hi all, welcome to The Hive's series on PCB Design with KiCAD. My name is Ben Hurwitz, and in this series, we've been walking through the PCB design process using KiCAD as our electronics design software.

In the last video, part 4E, we finished off the schematic capture with a look at the ERC, or the electrical rules check, and a few bits of schematic miscellany.

In these next few videos, we'll shift focus to the layout portion of the design process, which KiCAD calls the PCB view, in which we'll actually physically align and orient the actual components on the board and connect them with traces.

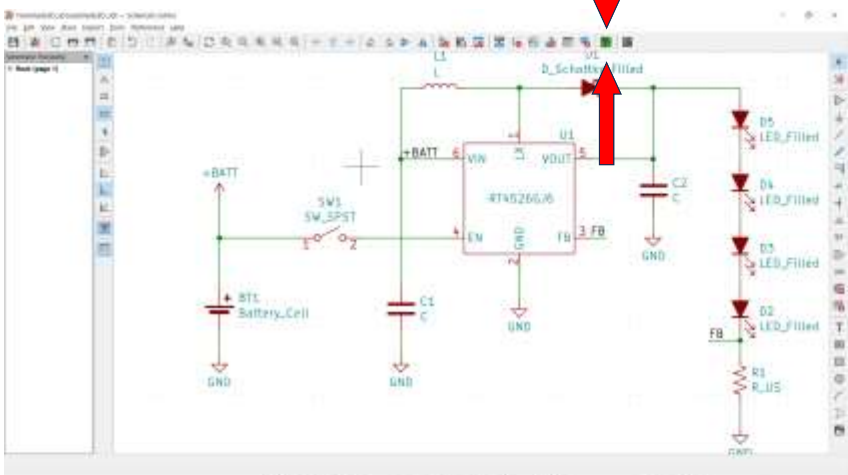
This section, part 5A, will introduce the board editor and look at setting up the board's defaults and design rules. Let's get started.

Main Window - Working Project



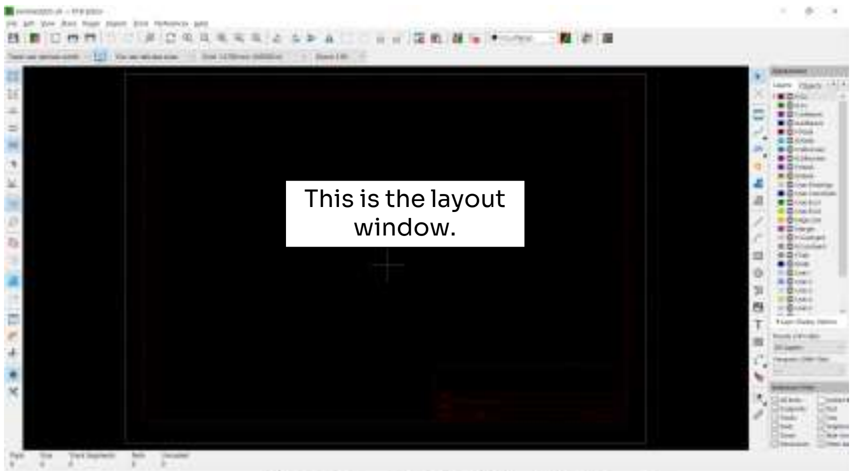
If you're just opening KiCAD for this video, go ahead and *load the project via the "File" menu, and open the PCB editor by ether *double clicking on the layout file with extension .kicad_pcb, or by *clicking on the PCB editor icon on the right, which will open the current project's board view by default.

Layout quick-switch

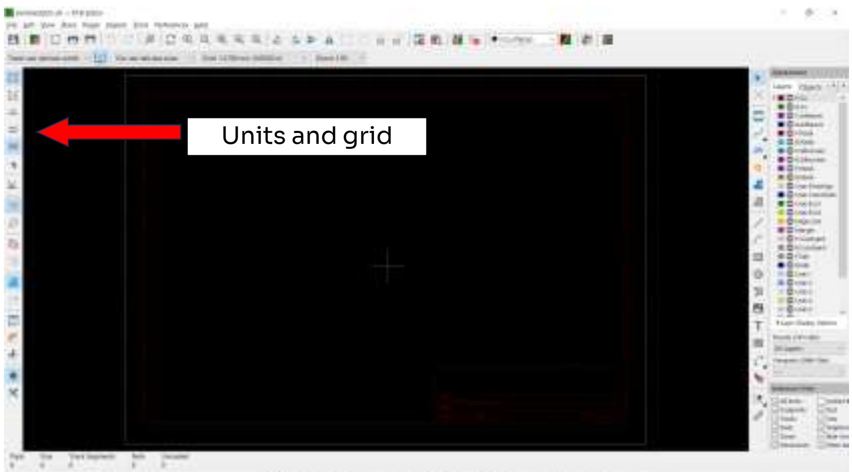


If you're coming from the previous video, part 4E, or you have the schematic editor up for whatever reason, you can also use the icon highlighted here to open the PCB view.

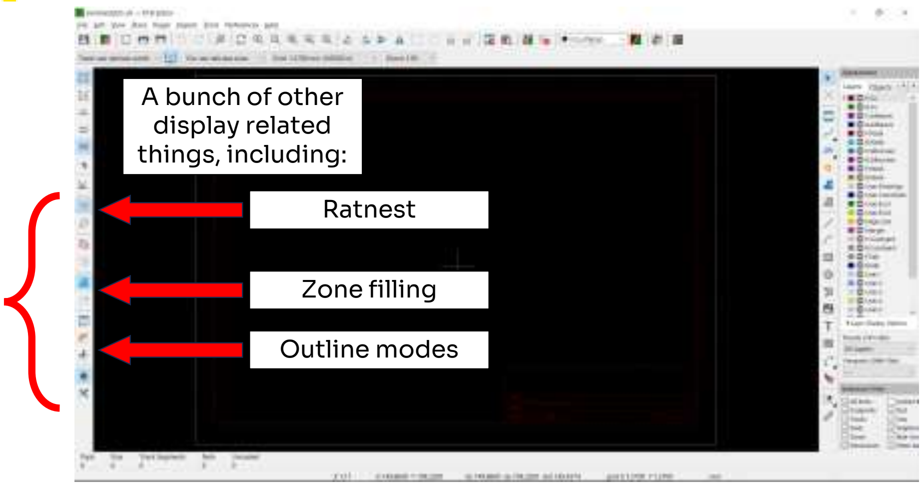
Layout



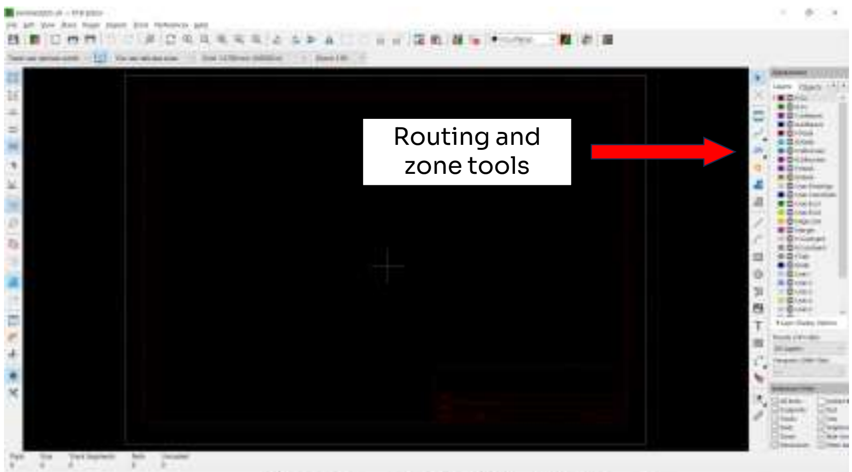
Layout



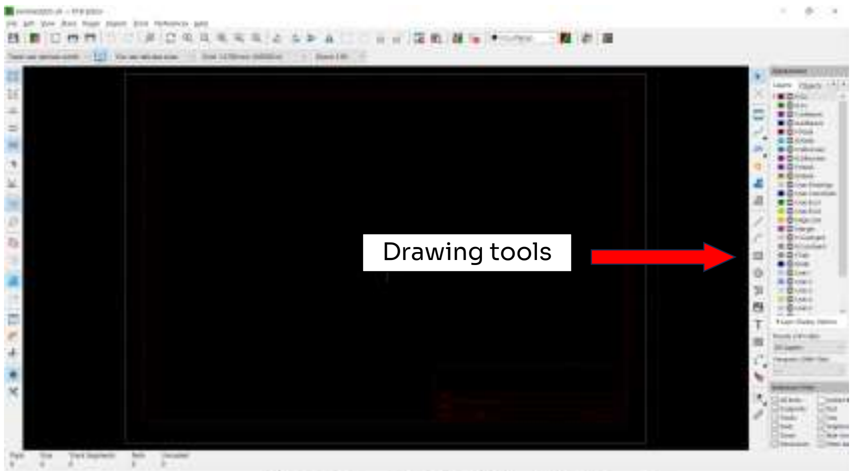
Layout


A bunch of other display related things, including:
Ratnest
Zone filling
Outline modes

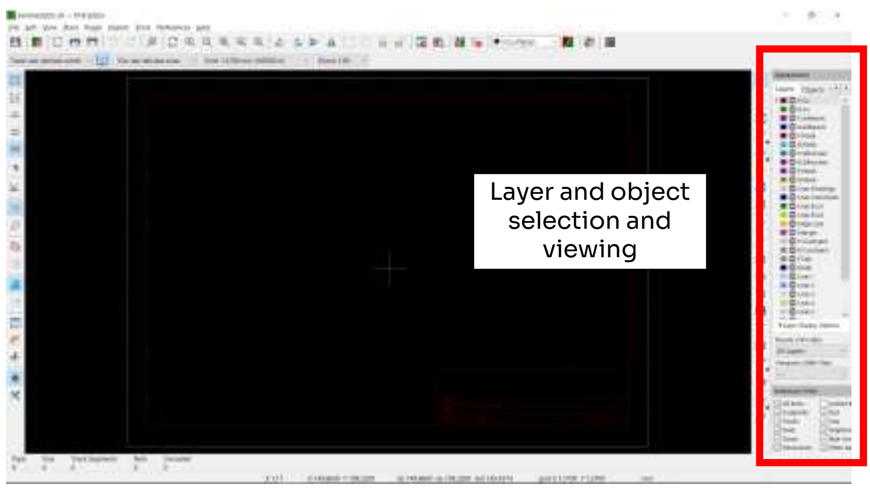
Layout



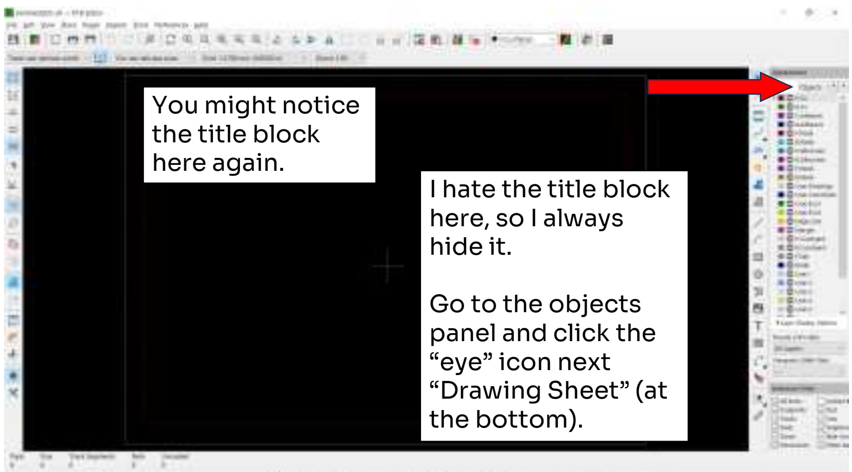
Layout



Layout

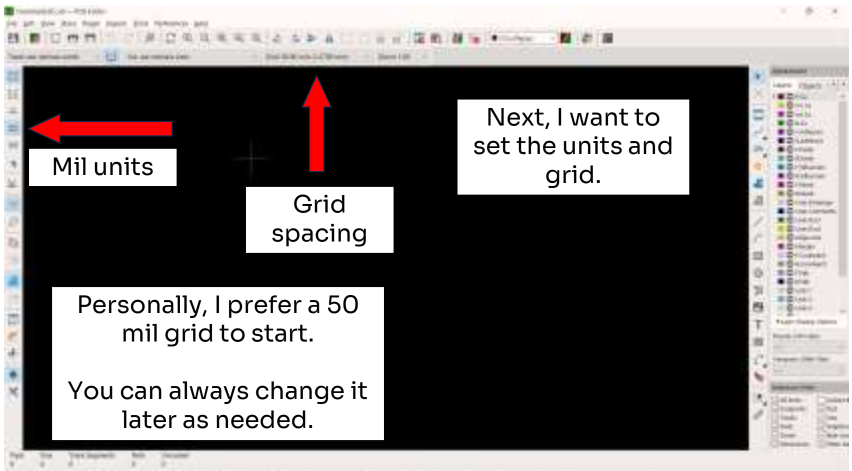


Layout



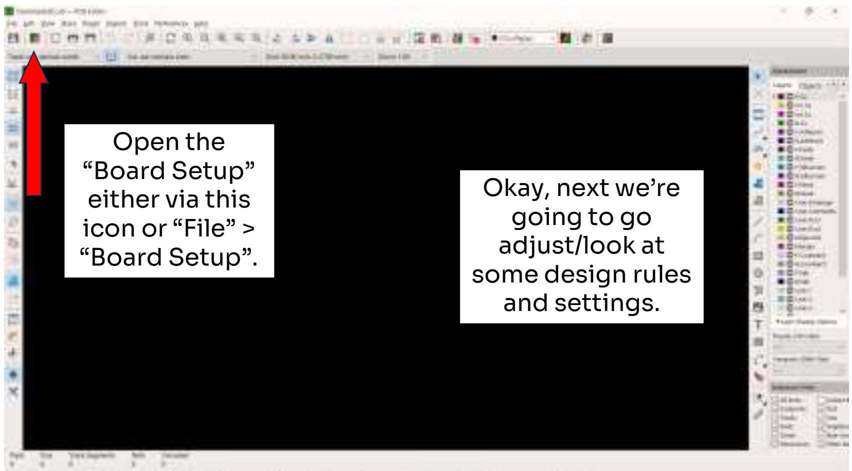
Similar to the Schematic editor, removing the title block requires making a new drawing sheet, which is beyond the scope of this tutorial.

Layout



Unlike the schematic in which the grid is really important to KiCAD's functionality, the grid is highly flexible in the layout editor, and can (and often will) be changed frequently.

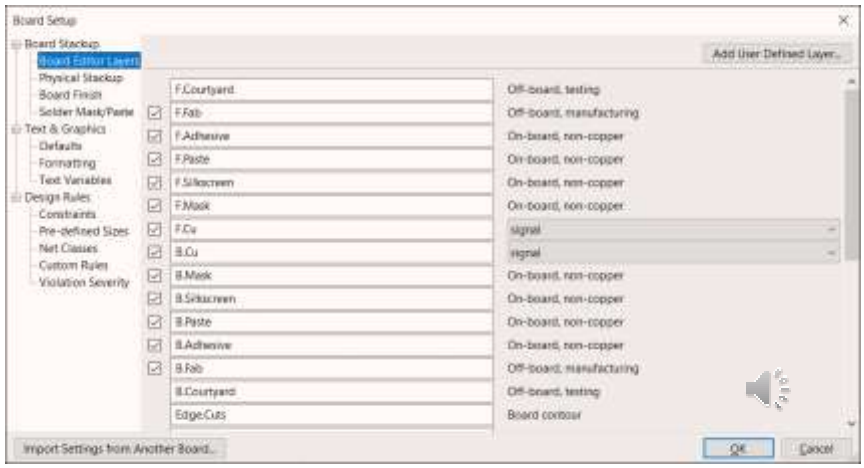
Layout


Open the "Board Setup" either via this icon or "File" > "Board Setup".
Okay, next we're going to go adjust/look at some design rules and settings.

Layout – Board Setup

The “Board setup” window offers you many settings to adjust for your design.

There is a similar window for the schematic, but it’s much more important to understand some of these settings in the layout view.

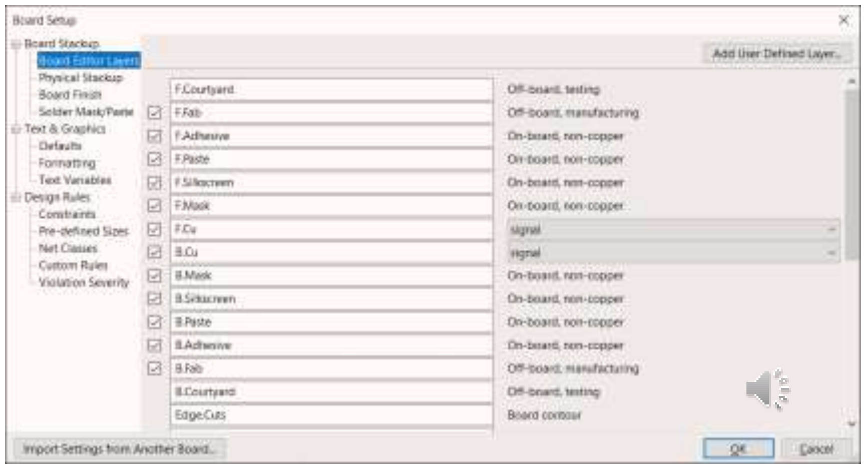


Layout – Board Setup

The first section allows you to select the layers available to your design.

You can select/deselect these as you see hit in your design.

I believe they will not be used at all if they're deselected, versus just not being visible.

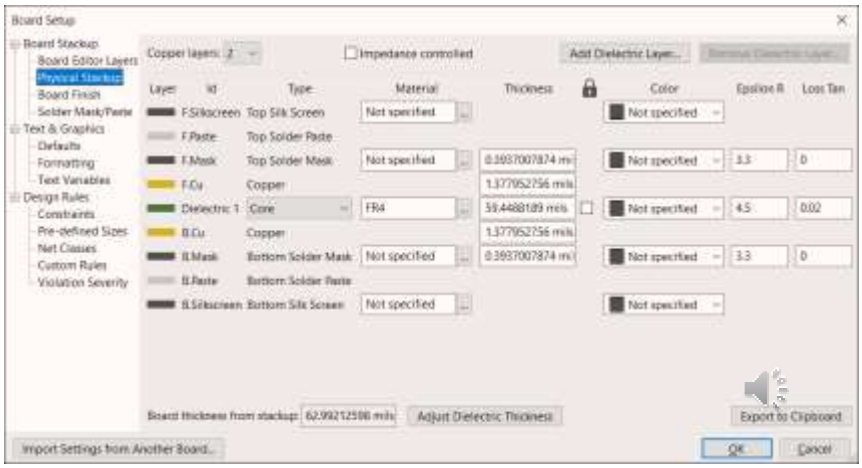


This can be useful if it's important that you don't put silkscreen or copper on the backside of the board, for example. By removing them from the layers entirely, you minimize changes of error. You can also add user-defined layers here if you want or need, perhaps for a custom assembly process, required by your fab house, or for specific additional information.

Layout – Board Setup

“Physical stackup” refers to the the physical (rather than digital) layers of the boards.

This is where you would set your board to be 4 layers, or 8 or whatever, using the “Copper layers” drop down.



You can do some really advanced setup in here, though I’m honestly not sure how much of it is used in other parts of the software versus just being for your (and you boss’s) knowledge.

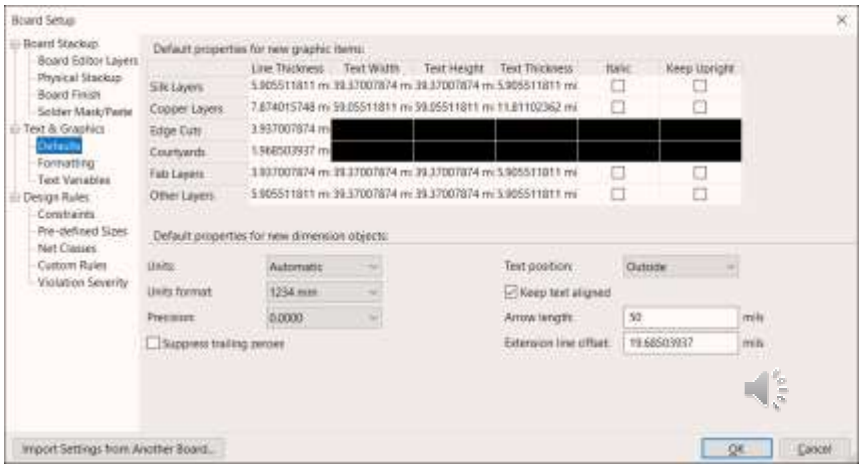
The units in this pane and in all other panes are derived from the units specified in the main layout editor window. KiCAD is a metric-based design tool, which is why the values are so wonky in mils.

Layout – Board Setup

Here you can set some defaults for text and graphics.

The defaults are normally okay, but it can be hard to gauge what size you need.

Good rule of thumb is that height = width = 6*thickness.

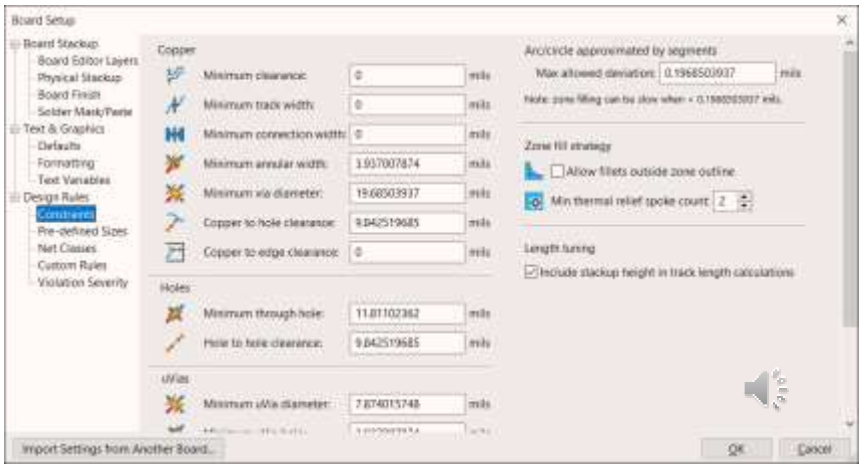


It can be a useful exercise to make a dummy layout only with text on it at various height/width and thickness ratios for your reference. If you print that board (CTRL + P, or “File” > “Print”), it can be handy to keep nearby for scale.

Layout – Board Setup

The "Constraints" pane is where all our design rules go, things like minimum spacings and sizes.

These are defined by your chosen fabrication house (i.e. where you're getting the board made). Read their instructions closely.

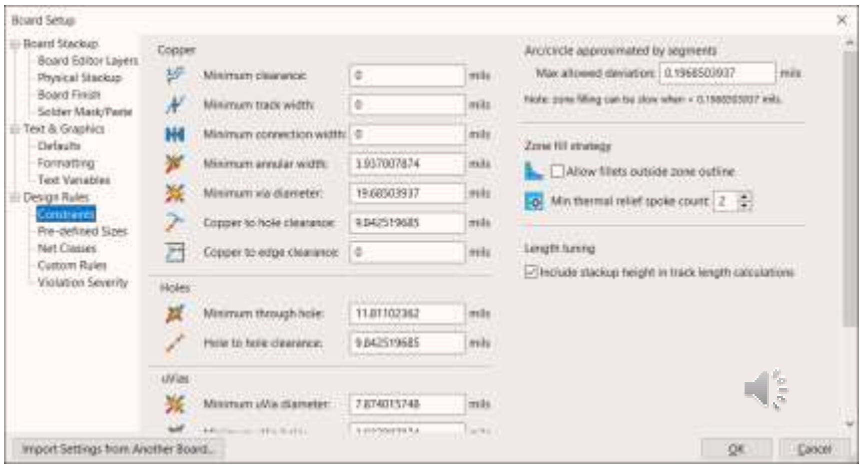


Many fab houses have software-specific instructions these days. Read them carefully! This is an easy way to make your design un-fabricatable, and require re-doing the layout.

Layout – Board Setup

I’m not going to adjust any of this now because I’m not getting this fabbed, but do not forget to set this.

If you set it at the end, that’s okay, but be prepared for a bunch of DRC errors and potentially significant re-design.



Layout – Board Setup

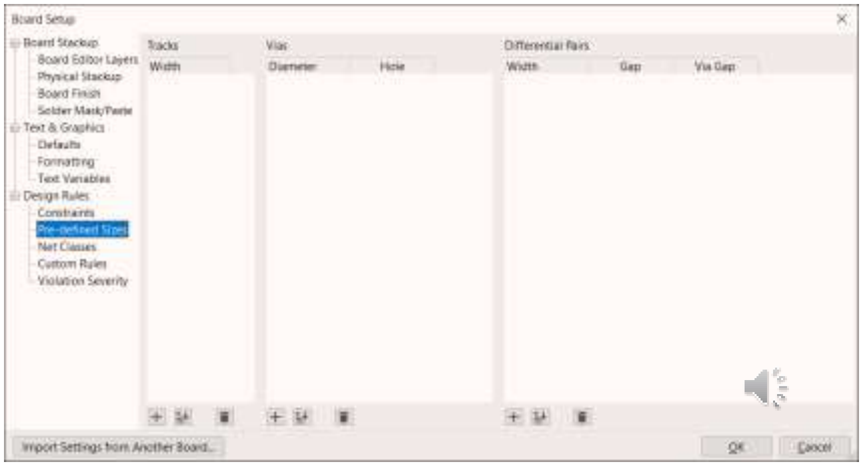
You can adjust the severity of the DRC rules (warning, error, or ignore) under the “Violation Severity” tab.

Don’t do this unless you’re confident of the result, or you may end up with a bad board.

Layout – Board Setup

In "Pre-defined Sizes", we can set some pre-defined sizes for traces and holes.

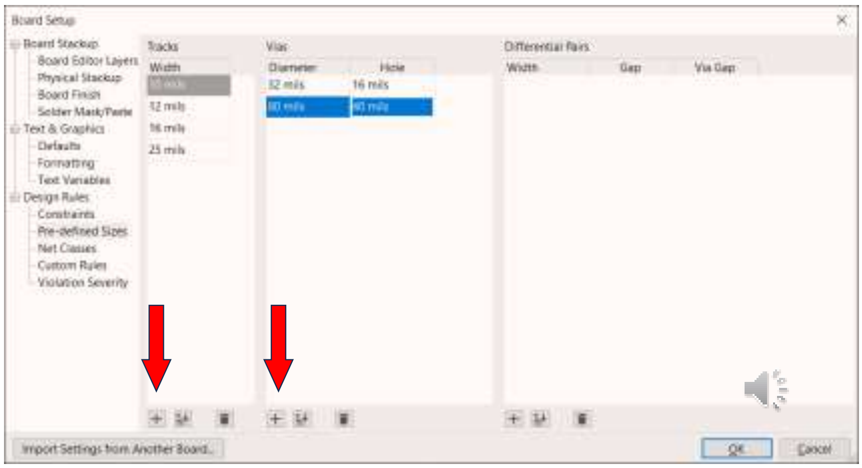
This will make your lives easier later by allowing hot-changing of sizes.



Layout – Board Setup

Use the “+” button to add:

- Track widths of 10 mil, 12 mil, 16 mil, and 25 mil.
- Via diam/holes of 32/16 mil and 80/40 mil



The parameter “diameter” for holes and vias refers to both the drill hole diameter itself plus the annular ring. It’s generally a good starting point to have the ring diameter to be twice the hole diameter, and larger can make soldering easier.

These default values are not required by anything.

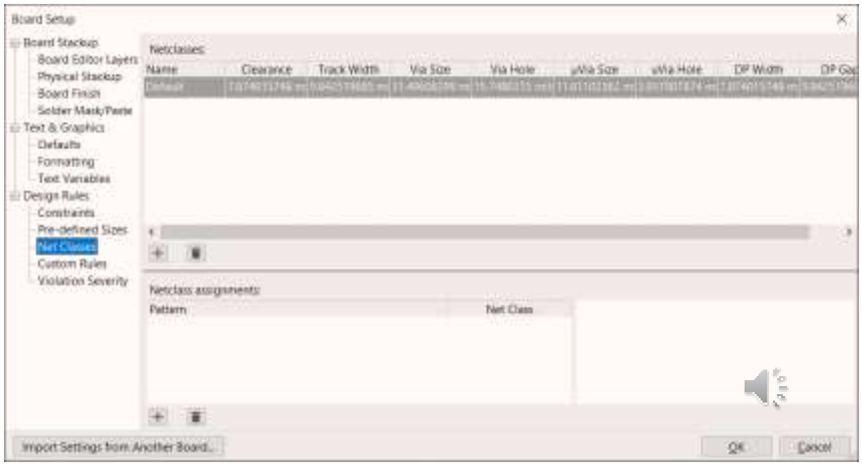
I like to keep my traces above the minimum allowable if possible, which is normally around 6 mils, to reduce the possibility of broken or damaged minimum-width traces that are more likely to have issues. For signals, I try to use 12 mils generally and neck-down, meaning shrink briefly, to 10 mils. Power traces should have the largest reasonable width possible. 25 mils is a good balance, and according to the trace-width calculator at Advanced Circuits, can be used to pass 1A on external 0.5 oz copper with just a 10 degree Celsius rise in temperature. 16 mils is a good necking size for brief lengths over which 25 mils can’t fit.

For the vias, the 16 mil diameter hole for vias is a classic default, and is good for small currents and signals. The 40 mil hole is about 1mm, and will fit a standard pin header if you need a via but do not have electroplating. It’s also good for a lot more current. Not necessary typically, since you can use multiple small vias instead of one large one.

Layout – Board Setup

You can group nets together into *netclasses* to give them all the same default settings.

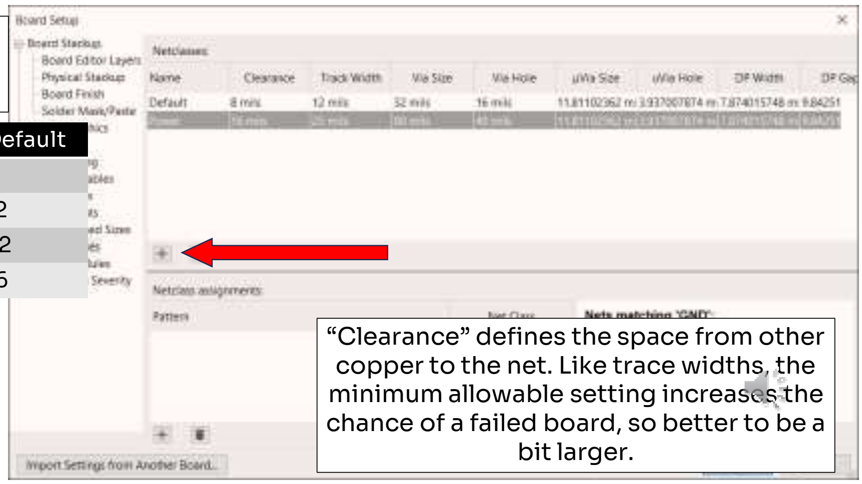
This is very useful if you want all your power traces to be wide, or some traces to be very narrow.



Layout – Board Setup

Let's define a "Power" netclass, and adjust "Default":

Parameter	Power	Default
Clearence	16	8
Track Width	25	12
Via Size	80	32
Via Hole	40	16



"Clearance" defines the space from other copper to the net. Like trace widths, the minimum allowable setting increases the chance of a failed board, so better to be a bit larger.

*Clearance… blah blah blah….

For The Hive's tool, because we don't offer a protective soldermask layer, it's important that this clearance is set quite large, 30-50 mils, to reduce the chances of accidentally jumping traces or a plane when soldering.

The uVia, or microvias, are extra-tiny vias that are typically used with BGA-style components with high-density ball-style pads underneath the package. These will add to your board cost, and are usually not needed unless you really can't find an alternative package.

DP stands for differential pairs, which are used when impedance matching is important, like for USB data or antennas.

Layout – Board Setup

Assign your power net (the one out of your battery) and the ground net to the “Power” class.

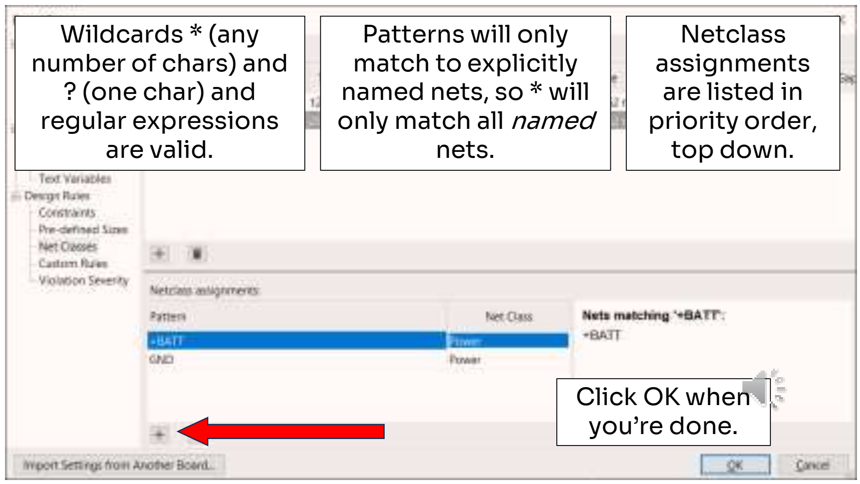
Must be one pattern per assignment line.

ALT+Tab into the schematic to double-check the names.

Wildcards * (any number of chars) and ? (one char) and regular expressions are valid.

Patterns will only match to explicitly named nets, so * will only match all *named* nets.

Netclass assignments are listed in priority order, top down.



Click OK when you’re done.

Netclass assignments can actually also be defined in the schematic by right-clicking a node or net.

And that ends part 5A of the KiCAD design tutorial series in which we introduced the layout editor and a number of pre-layout board setup settings. A PDF of this video is available as well, linked in the description and hosted on The Hive's Wiki.

In the next video, part 5B, we'll go through placement of the components and routing. See you then.