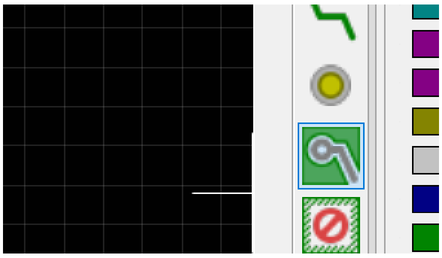
PCB Guide Part 6 - Fill zones, decoration and production

Now that the routing is finished, let's tune the PCB up.

Step 11. Ground fills

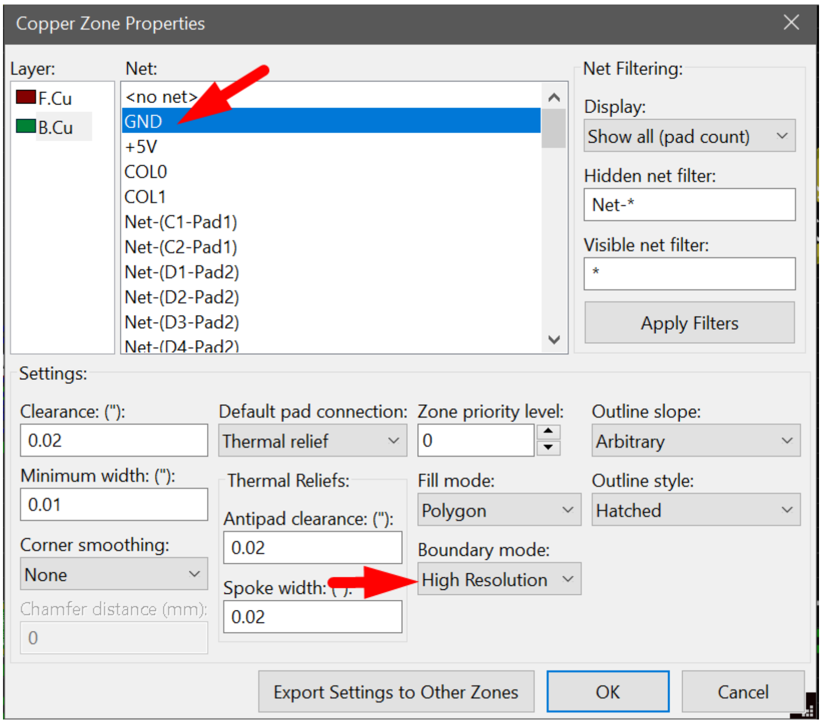
You may remember the mention about how the crystal zone is sensitive.
We can protect it, along with other components, from external electrical noise through the use of ground fills.

Select create fill zones to begin.



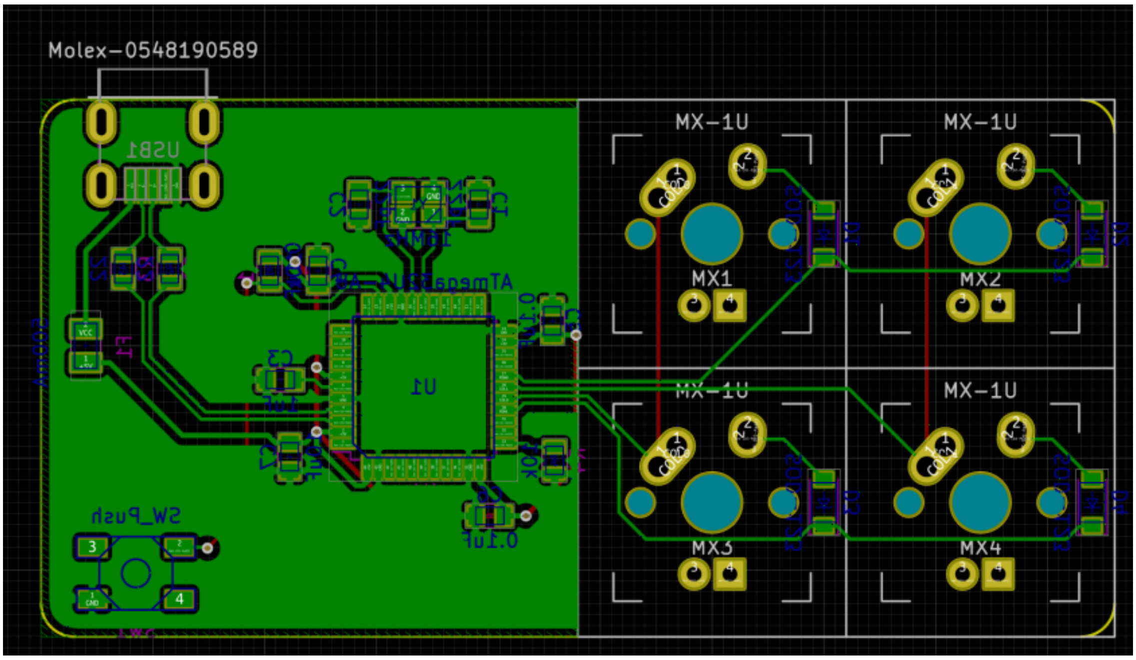
We will draw two zones: One around the MCU, and one for the USB connector.

Then click on the first corner of the zone to surround to bring up a menu.



Select the ground net for the fill net, and high resolution for prettier edges.

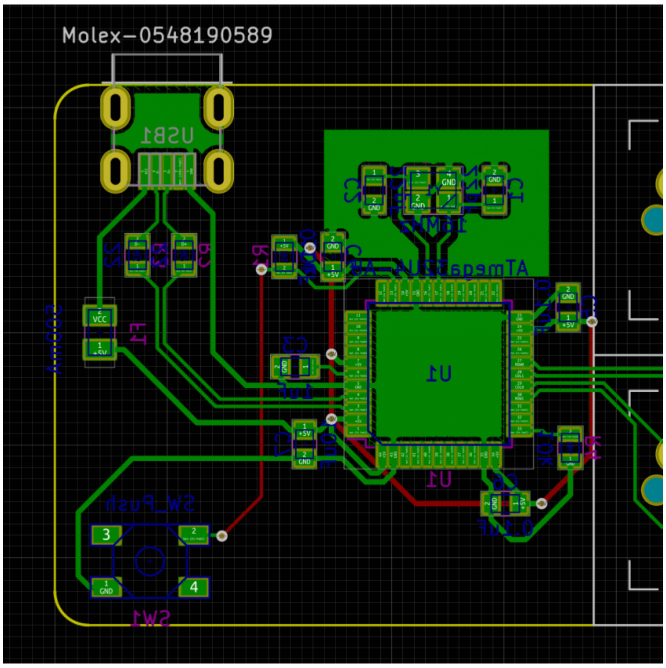
Click at the corners, and double click at the last corner to end.
Then, press B to fill the zone.



The ground fill question is always an interesting one.
On one hand, it reduces EMI interference both entering and exiting the board.
However, it adds a significant amount of copper, making it more difficult to solder due to the necessity to raise temperatures further for ground-connected components.
It also makes the PCB opaque. This can be a major disadvantage if you are wishing for the translucent PCB aesthetic.

I will say two things regarding this topic:

- If doing full ground fill, make sure there is no ground current interference (i.e. Ground current from LED doesn't take a path that cuts across MCU/Crystal zone).
- If doing minimal ground fill, at least do the following:
    - Around the crystal
    - Under the MCU, possibly a bit around
    - Below the USB connector
- Also, in this case, it is critical that the USB lines are routed properly: Not close and parallel to pulsing matrix and LED lines, crossing perpendicularly to other traces as much as possible.



Whether you choose aesthetic or function is up to you. Such is a design decision left for the designer.
My personal recommendation is to use fill around the entire MCU zone or entire board until you are used to PCB design and have a few functional prototypes.

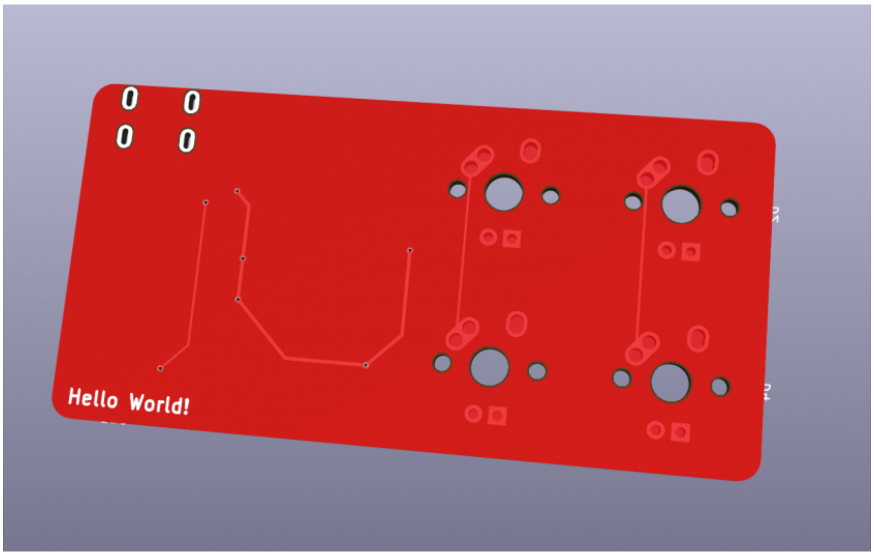
Step 12. Decorations

Let's celebrate your first PCB by making it yours.

The following layers can be used for aesthetic designs:

- F/B.SilkS - Silkscreen print. The best choice for most decorations.
- F/B.Cu - The copper. Add stealth logos and text that aren't as obvious.
- F/B.Mask - The soldermask. If no copper exists below, you will end up with a tan color exposed (PCB material). If copper below, you will get a shiny copper artwork or text.

For example, select the text tool on the layer F.SilkS, and place a block of text on the PCB.
Then, press Alt+3 to render the PCB.

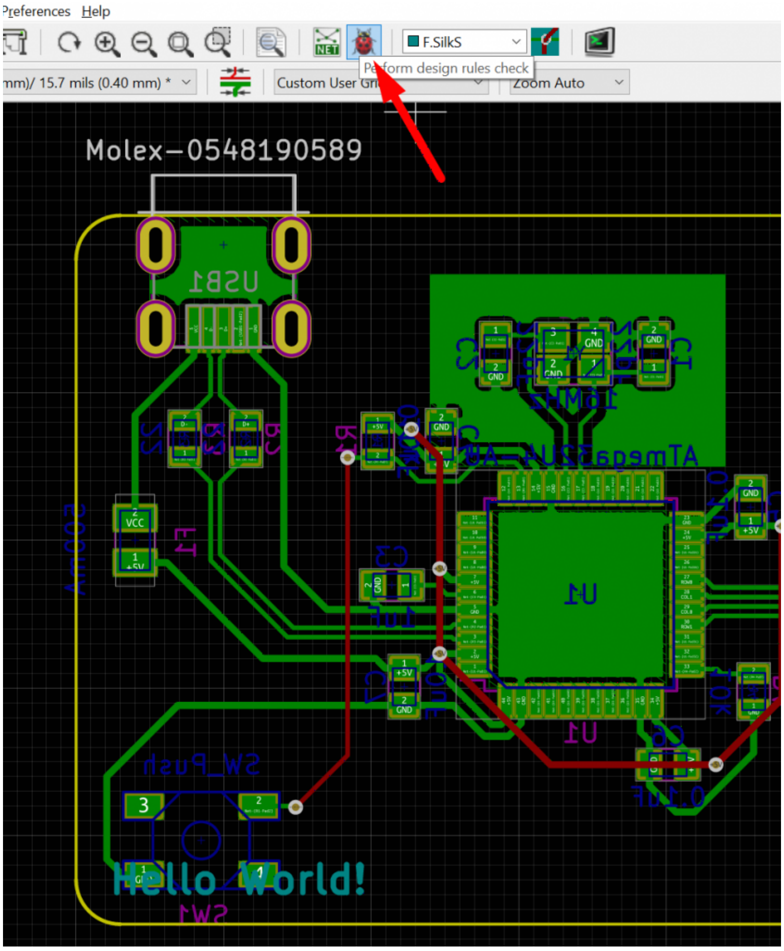


Artwork can be done using Bitmap2Component. This will be discussed in a future chapter.

Step 13. Error Check

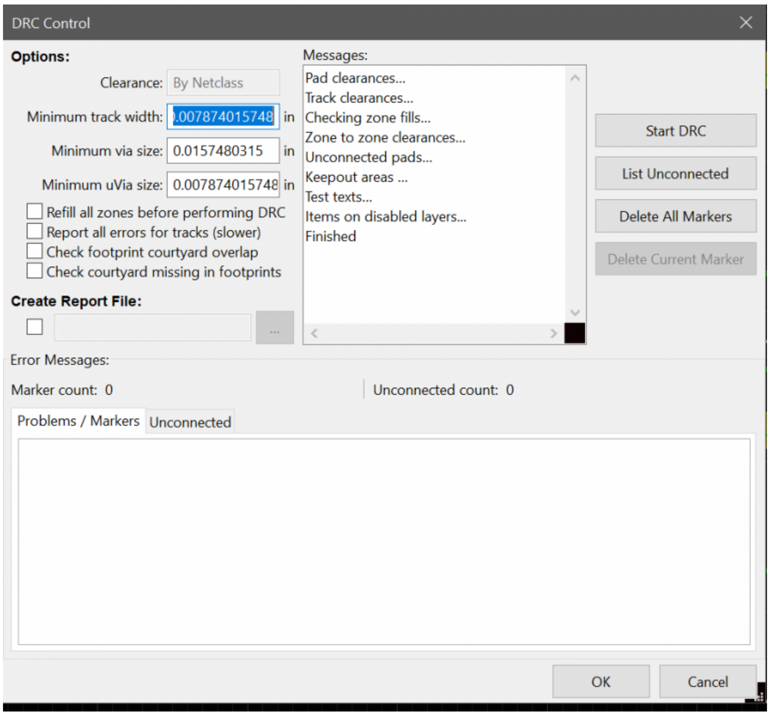
Are you sure that you haven't made any mistakes?

Use DRC to find out.



Press "Start DRC", then wait a few moments for KiCad to find things to complain about.

If it returns empty, you're all set.



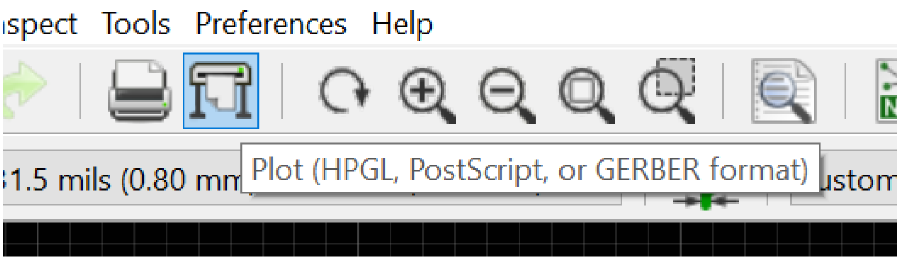
If not, you've made a mistake.
Double click the error to figure out what's wrong. (It will jump to the error point.)

Repeat until KiCad complains no more.

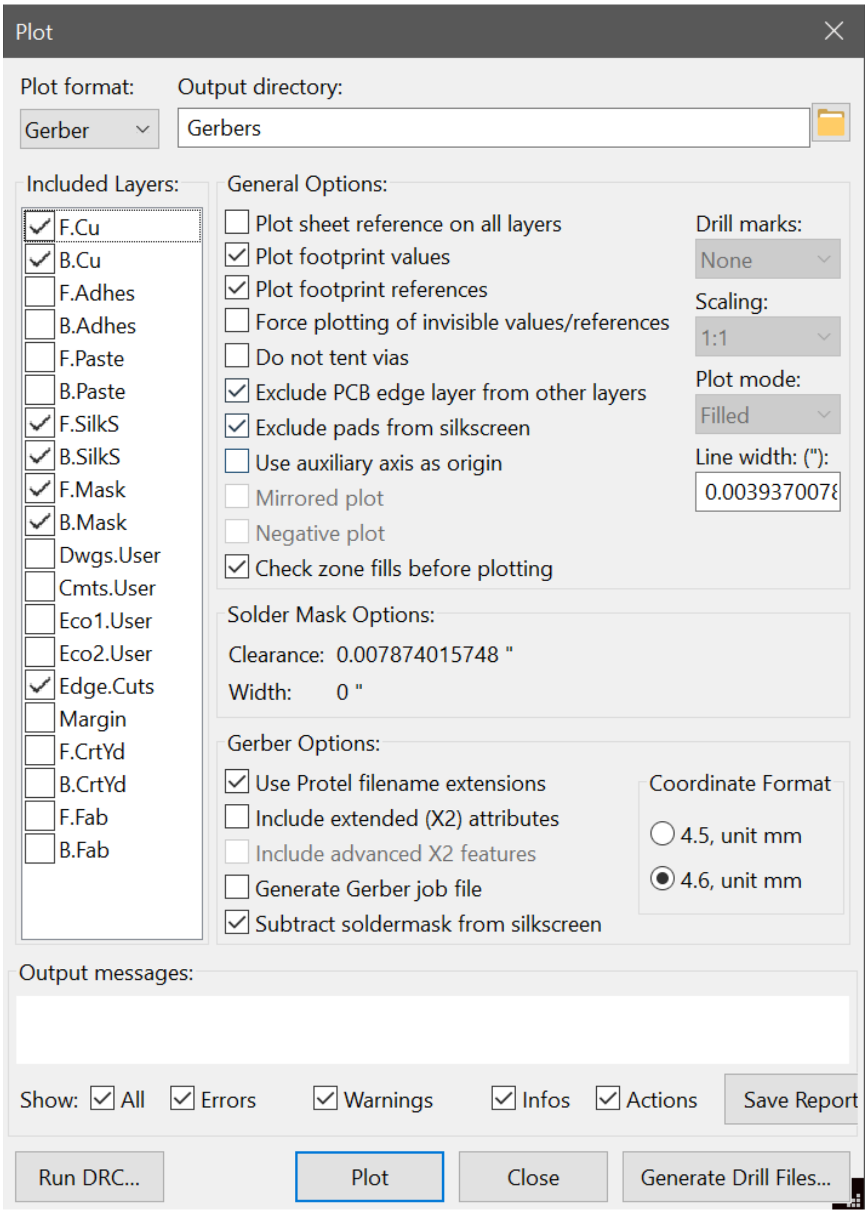
Step 14. Production

Let's get this board produced and in your hands.
Gerber files are the standard for PCB data. We will export this from KiCad.

Open the Plot menu from the toolbar.

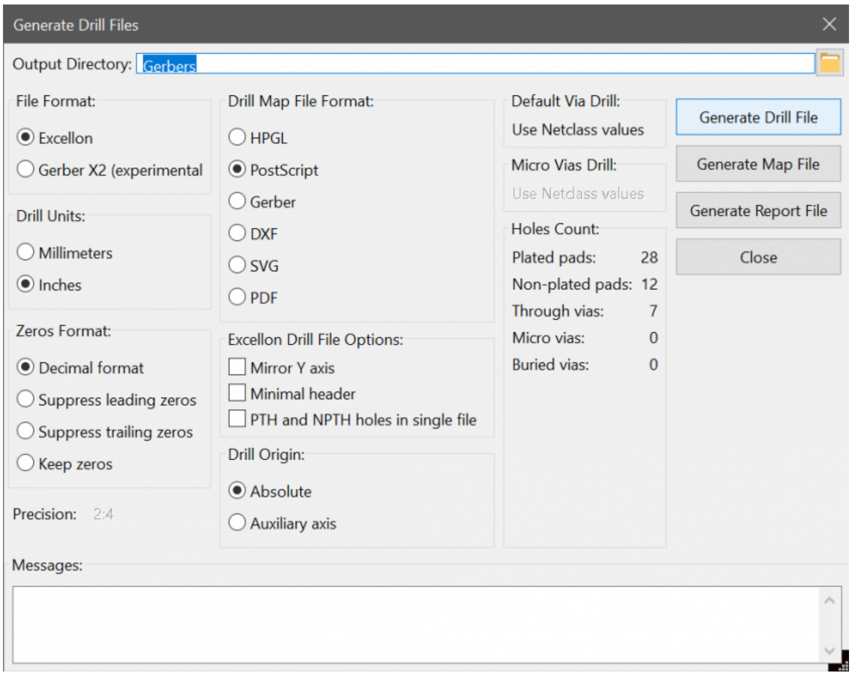


Here are the generic settings I use for exporting to most factories.

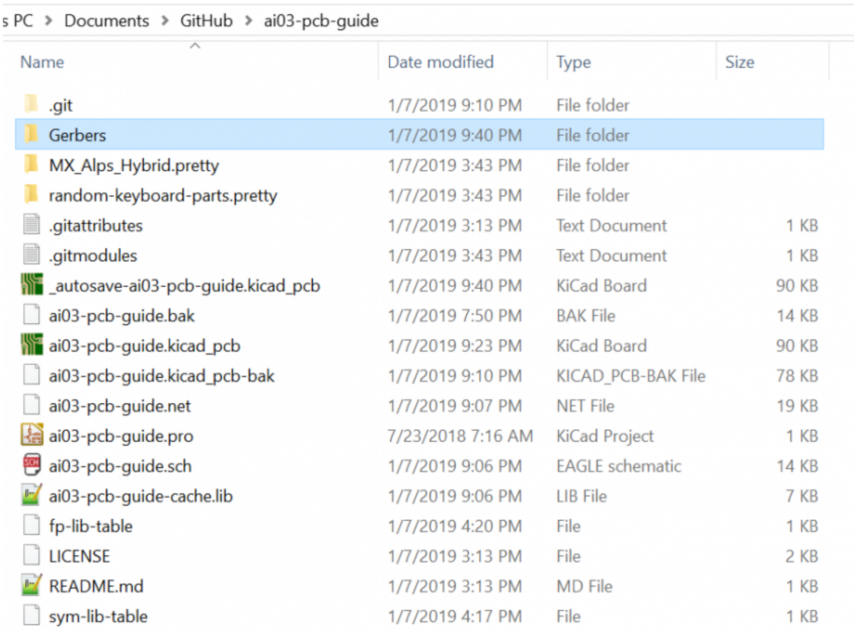


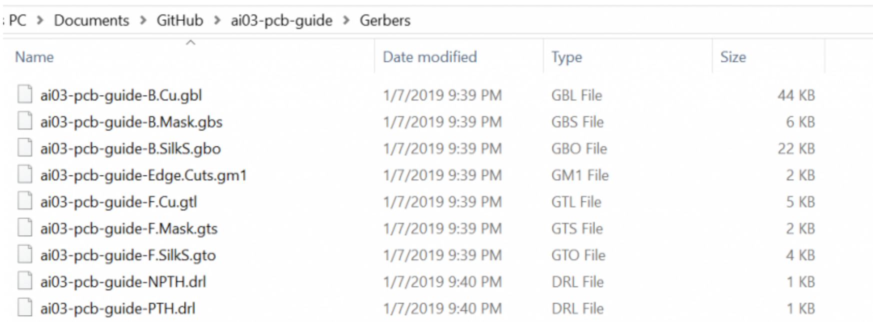
Once everything is set, press Plot to generate the data.

Now click Generate Drill Files.
Stock settings should work fine here. Simply generate the files and close all dialogs.



Check your repository folder - You'll notice that there are gerber files in the directory you specified.



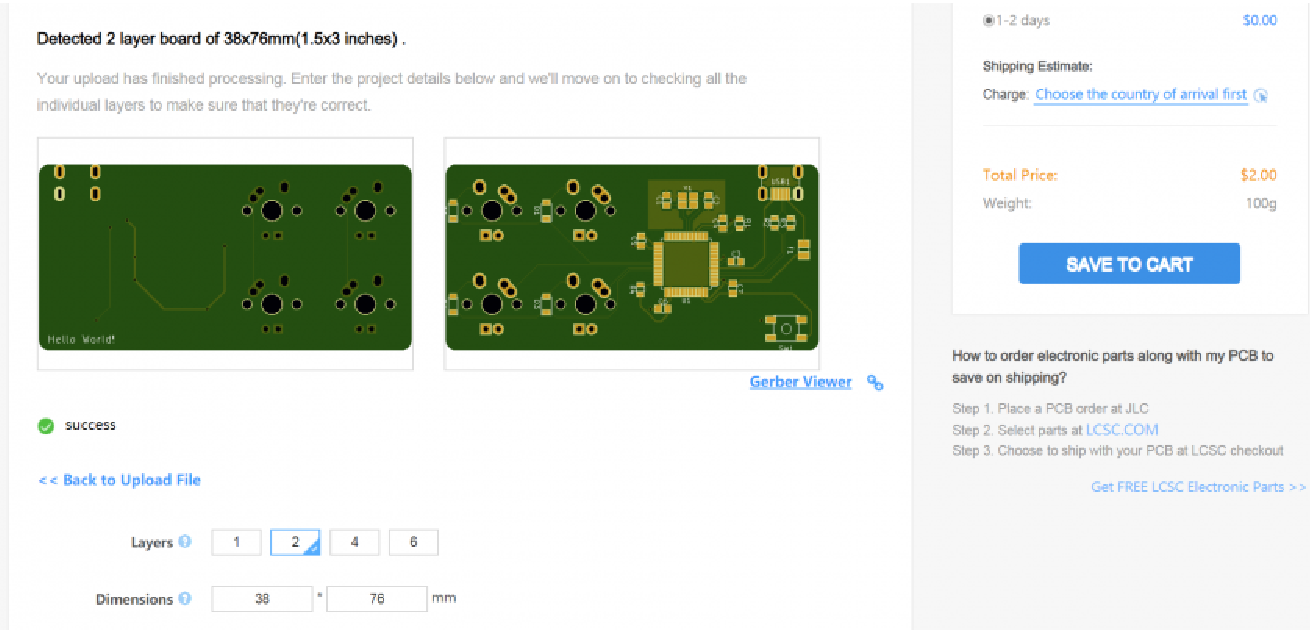


Add all these gerber files to a zip file.

ai03-pcb-guide-B.Cu.gbl	1/7/2019 9:39 PM	GBL File	44 KB
ai03-pcb-guide-B.Mask.gbs	1/7/2019 9:39 PM	GBS File	6 KB
ai03-pcb-guide-B.SilkS.gbo	1/7/2019 9:39 PM	GBO File	22 KB
ai03-pcb-guide-Edge.Cuts.gm1	1/7/2019 9:39 PM	GM1 File	2 KB
ai03-pcb-guide-F.Cu.gtl	1/7/2019 9:39 PM	GTL File	5 KB
ai03-pcb-guide-F.Mask.gts	1/7/2019 9:39 PM	GTS File	2 KB
ai03-pcb-guide-F.SilkS.gto	1/7/2019 9:39 PM	GTO File	4 KB
ai03-pcb-guide-NPTH.drl	1/7/2019 9:40 PM	DRL File	1 KB
ai03-pcb-guide-PTH.drl	1/7/2019 9:40 PM	DRL File	1 KB
Gerbers.zip	1/7/2019 9:41 PM	Compressed (zipped)...	18 KB

Congratulations, you can use this file for production.
Commit a celebratory commit to GitHub so you don't lose your work when KiCad decides to have a bad hair day.

For example, you can upload the zip at JLCPCB, and it will generate a quick screenshot of the PCB data.



Step 15. Ordering parts

Let's finish the guide up by buying all the parts necessary to assemble this board.

Component	Quantity	Package	Sample Part
Ceramic Capacitor, 0.1uF	3	0805	1276-2450-1-ND
Ceramic Capacitor, 10uF	1	0805	1276-6455-1-ND
Ceramic Capacitor, 1uF	1	0805	1276-1066-1-ND
Ceramic Capacitor, 22pF	2	0805	1276-2605-1-ND
Diode, Generic	4	SOD-123	1N4148WTPMSCT-ND
Polyfuse, 500mA hold, 1A trip	1	1206	507-1803-1-ND
Resistor, 10k	2	0805	P10KDACT-ND
Resistor, 22	2	0805	P22ACT-ND

Low Profile Tactile Switch	1	5.2x5.2mm	CKN10361CT-ND
ATMEGA32U4 Microcontroller	1	44TQFP 10x10mm	ATMEGA32U4-AU-ND
Molex Mini-B USB C Receptacle	1	Molex-0548190589	WM3895CT-ND
Crystal, 16MHz	1	3.2x2.5mm, 4 pad	1253-1698-1-ND?

The guide will be expanded sooner or later to cover LEDs and such.
For now, you can refer to the advanced info.

Also, feel free to join ai03's Discord server.
It provides a safe place for all to discuss and design.

Revision #16
Created 8 January 2019 05:10:45 by ai03
Updated 29 June 2020 01:50:39 by ai03