DipTrace: Design Your PCB for OSHpark.com

OSH Park is a PCB manufacturer tailored to hobbyists. They take orders for very small quantities of boards (minimum 3) and they are able to do this by taking small hobbyist projects and compiling them into a big batch. A typical two layer board costs US$5 per square inch.

In this short tutorial you will learn how to a export DipTrace PCB Layout for OSH Park.

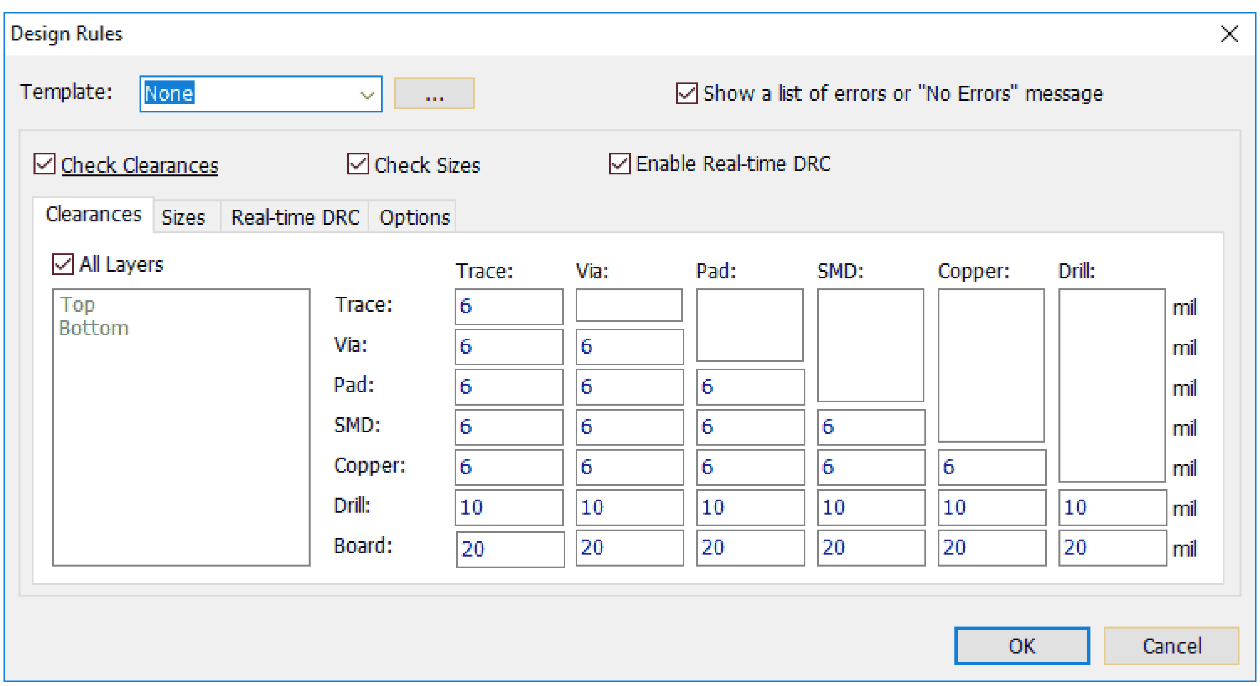
I love Diptrace, but OSH Park doesn't accept its native "dip" file format. That being said, they do take in Gerber files. But before we send PCBs to the fab, we must design a board that complies with OSH Park specifications; which are:

- Minimum 6 mil trace width
- Minimum 6 mil trace spacing
- Minimum 10 mil drill size
- Minimum 5 mil annular ring
- Minimum 15 mil keep-out distance from traces to the board edge.
- Cutouts must at least 100 mil wide

Setting up Design Rules in Diptrace

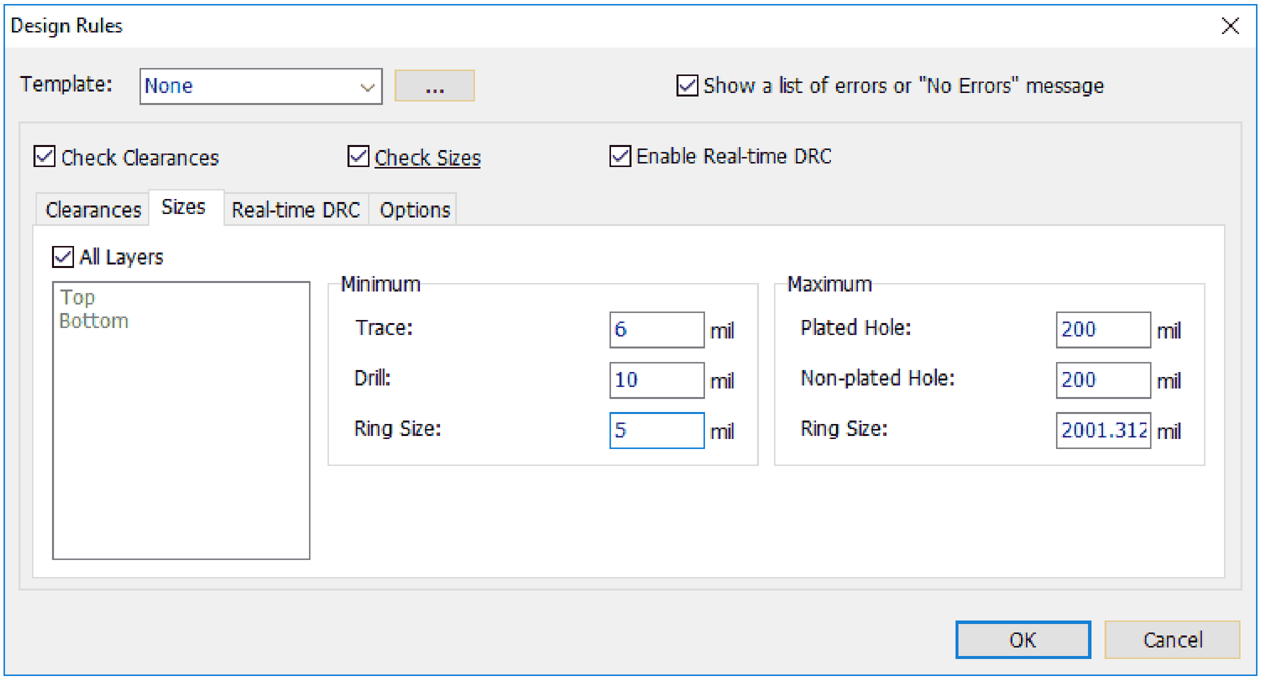
To make sure you are respecting these rules, in Diptrace PCB Layout go to **verification/design rules**.

By default, Diptrace tends to be more conservative than what OSH Park can deliver; but there are few things to change:



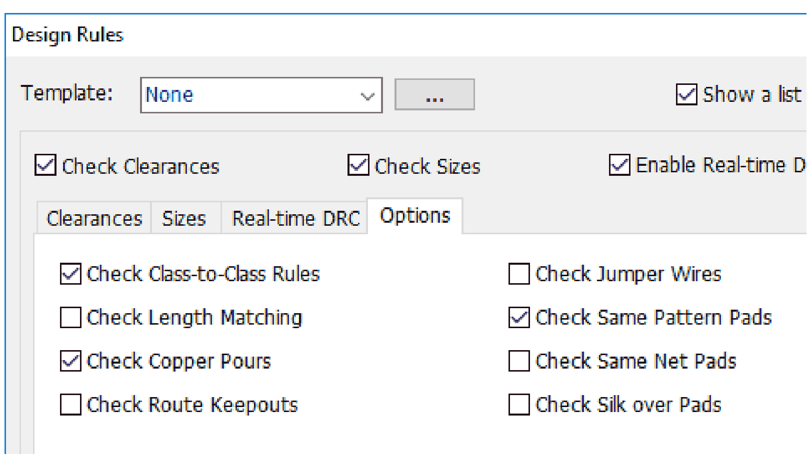
You can set the board clearance to 15 mil technically as it is what OSH Park requires, but I like the extra 5 mil buffer so it is set to 20 here. 20 mil is ridiculously small when you think about it (half a millimeter!).

The second tab is very important because you need to specify the minimum sizes. Again, DipTrace here was by default more conservative than what OSH Park can manufacture:



And adding on to that, as an hobbyist it is very unlikely you'll ever need traces smaller than 10 mil, even for fine pitch QFN or TSSOP packages, but it doesn't hurt to set the parameters right regardless.

Finally on the last tab, don't forget to tick the box "check copper pour" which is disabled by default:



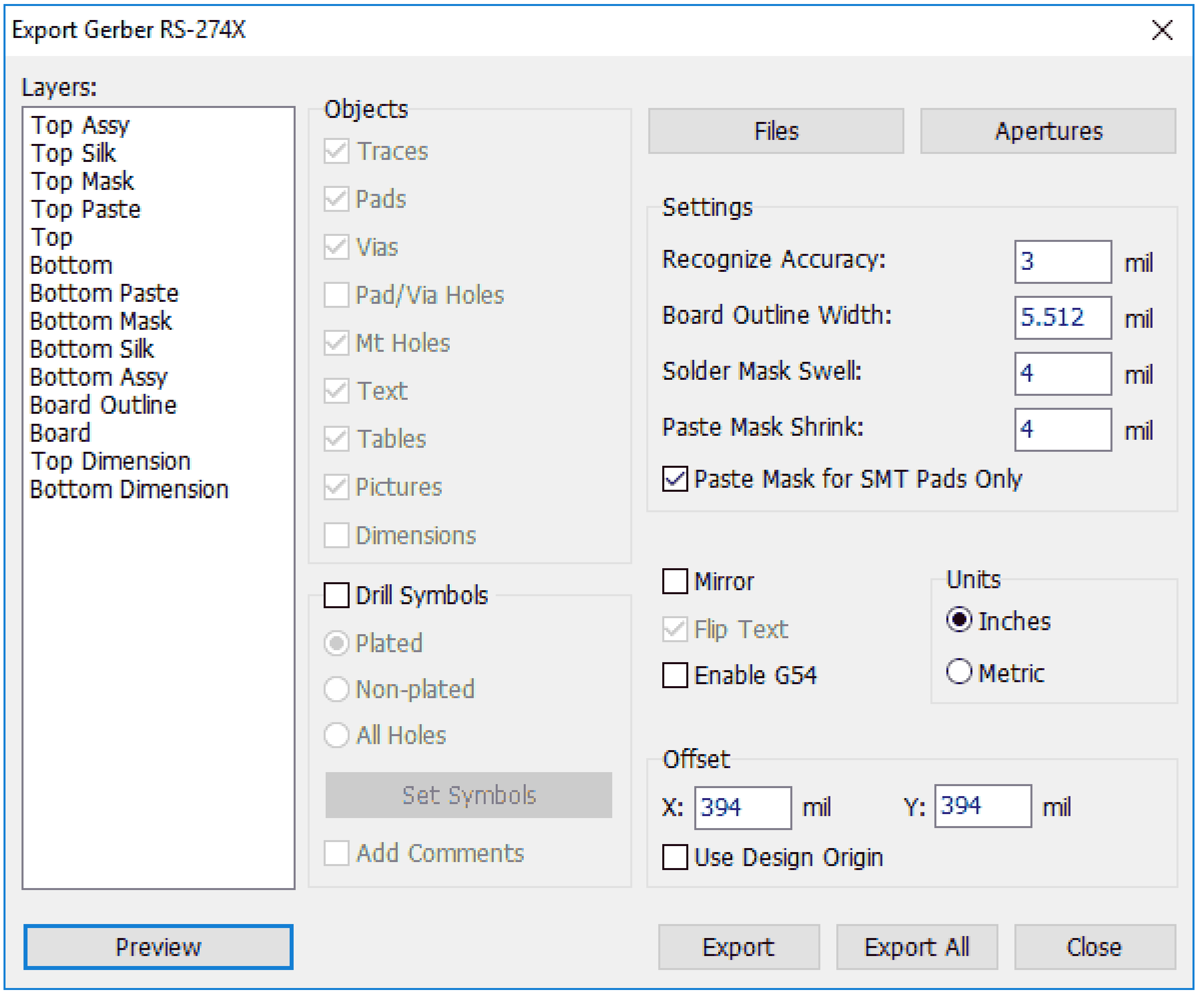
In my experience it is the most important setting because it's very easy to run copper pour along the edges and forget about the clearance here.

Once in a while press **F9** or **Verification/Check Design Rules** to make sure your PCB is correct; or enable **Real-time DRC**.

Exporting your PCB for OSH Park

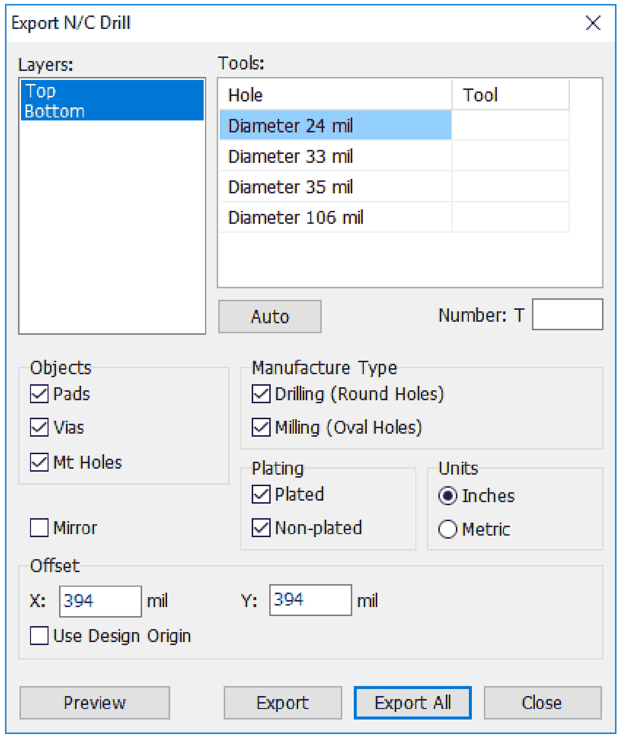
One you're happy with your design select **File**/**Export**/**Gerber**…

Leave all by default options and click "**Export All**".

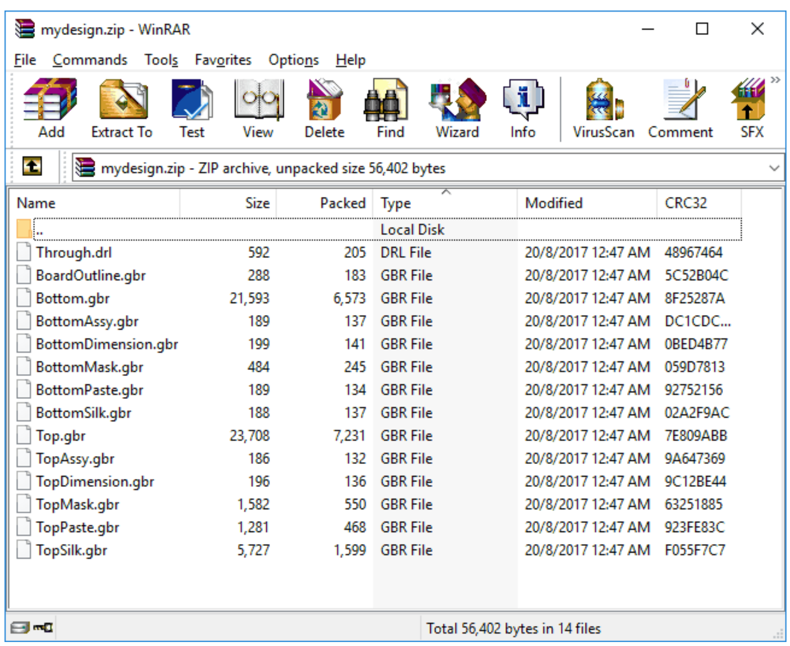


Next, click **File**/**Export**/**N/C Drill**…

Again, click on "**Export All**". When Diptrace asks you if you want to set tools automatically, click **Yes**.



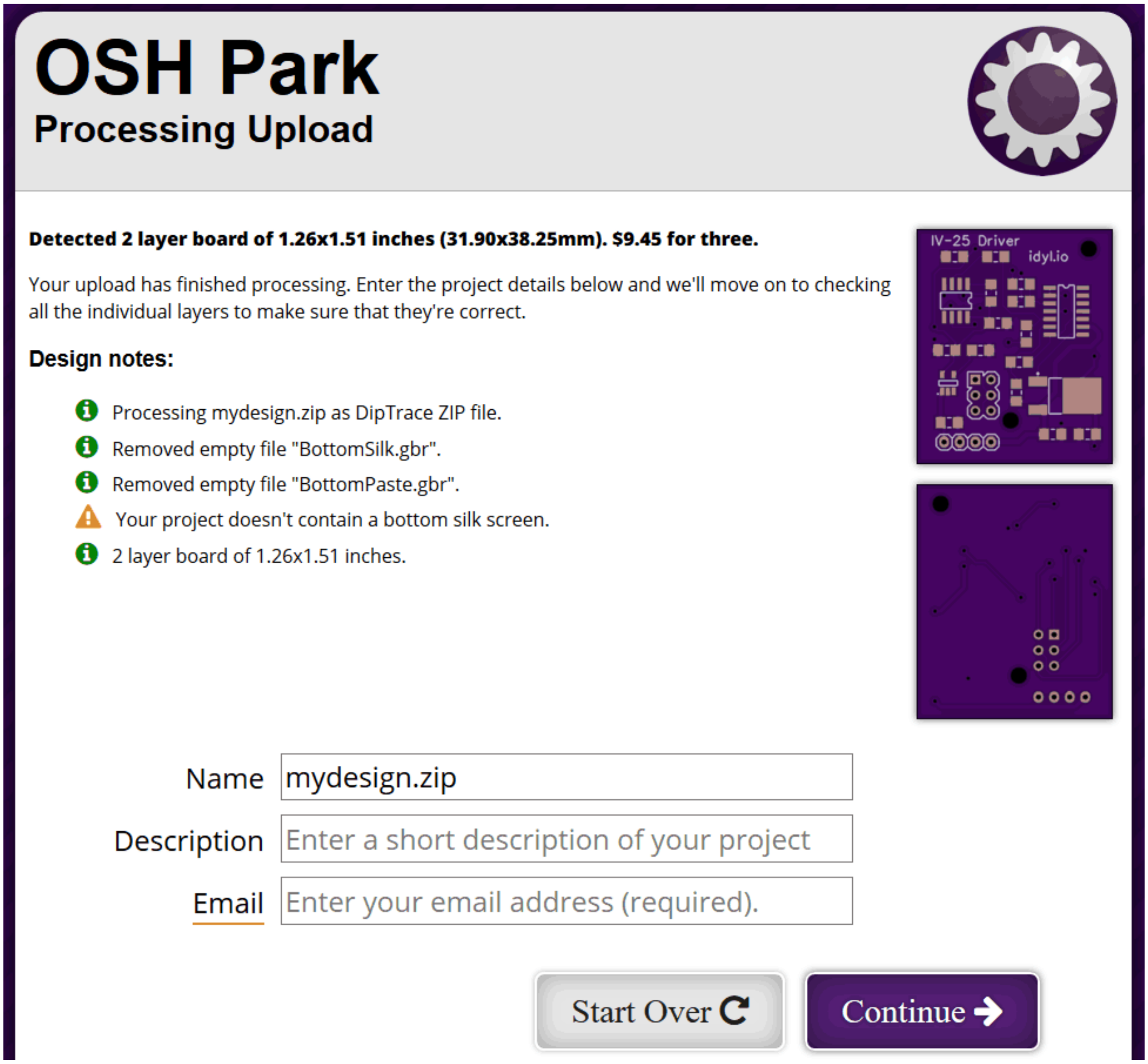
You should now have a folder that contains a lot of "gbr" files alongside a single drill file named "Through.drl". Zip them all using your favourite archive tool.



You are now ready for OSH Park!

Uploading your design to OSH Park

If everything was exported and zipped correctly, OSH Park will automatically detect your board and display an image of both sides in their signature purple silk mask.



<u>NOTE:</u> You don't need a bottom silk screen, but OSH Park will remind you that it's empty. Make sure everything is correct and just order!

Diptrace for OSH Park: easier than expected!

As you can see, while it is not as convenient as simply dropping your CAD Eagle file to OSH Park, exporting the Gerber & Drill only takes a few seconds. A small price to pay to be able to carry on using your favourite PCB layout tool!

<u>Credits:</u> https://idyl.io/2017/08/23/diptrace-design-your-pcb-for-oshpark-com/