# **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:

nplate: None				Show a	list of errors o	or "No Errore" r	nessage	
indite	`				lise of errors e		nessaye	
Check Clearances	🗹 Check	Sizes	🗹 Er	nable Real-tim	e DRC			
learances Sizes Re	al-time DRC Optio	ns						
		_				-	<b>D</b> 1	
	<b>—</b>	Trace:	Via:	Pad:	SMD:	Copper:	Drill:	
Bottom	Trace:	6						mil
	Via:	6	6					mil
	Pad:	6	6	6				mil
	SMD:	6	6	6	6			mil
	Copper:	6	6	6	6	6		mil
	Drill:	10	10	10	10	10	10	mil
	Board:	20	20	20	20	20	20	mil

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:

Design Rules				×
Template: None	~	Show a	a list of errors or "No Errors" mes	sage
Check Clearances Clearances Sizes Real-time	Check Sizes	🗹 Enable Real-tin	ne DRC	
All Layers	Minimum Trace: Drill: Ring Size:	6 mil 10 mil 5 mil	Maximum Plated Hole: Non-plated Hole: Ring Size:	200 mil 200 mil 2001.312 mil
			ОК	Cancel

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:

Design Rules		
Template: None	~	Show a list
Check Clearances	Check Sizes	🗹 Enable Real-time D
Clearances Sizes Real-time	DRC Options	
Check Class-to-Class Rules		Check Jumper Wires
Check Length Matching		🗹 Check Same Pattern Pads
Check Copper Pours		Check Same Net Pads
Check Route Keepouts		Check Silk over Pads

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".

Export Gerber RS-274X			×	
Layers:				
Top Assy Top Silk	Objects Traces	Files	Apertures	
Top Mask Top Paste Top Bottom Paste Bottom Mask Bottom Silk Bottom Assy Board Outline Board Top Dimension Bottom Dimension	<ul> <li>Pads</li> <li>Vias</li> <li>Pad/Via Holes</li> <li>Mt Holes</li> <li>Text</li> </ul>	Settings Recognize Accuracy: Board Outline Width: Solder Mask Swell:	3 mil 5.512 mil 4 mil	
	<ul> <li>Tables</li> <li>Pictures</li> <li>Dimensions</li> </ul>	Paste Mask Shrink:     4     mi       ✓ Paste Mask for SMT Pads Only		
	<ul> <li>Drill Symbols</li> <li>Plated</li> <li>Non-plated</li> <li>All Holes</li> <li>Set Symbols</li> <li>Add Comments</li> </ul>	<ul> <li>Mirror</li> <li>✓ Flip Text</li> <li>Enable G54</li> <li>Offset</li> <li>X: 394 mil Y</li> <li>Use Design Origin</li> </ul>	Units Inches Metric 394 mil	
Preview		Export Expo	rt All Close	

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**.

Export N/C Drill		×
Layers:	Tools:	
Top Rottom	Hole	Tool
Boccom	Diameter 24 mil	
	Diameter 33 mil	
	Diameter 35 mil	
	Diameter 106 mil	
	Auto	Number: T
Objects ☑ Pads ☑ Vias	Manufacture Type Drilling (Round H Milling (Oval Hole	oles) s)
Mt Holes	Plating ✓ Plated	Units Inches
Mirror	Non-plated	OMetric
Offset		
X: 394 mil	Y: 394 mil	
Use Design Origin		
Preview	Export Expo	ort All Close

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.

는 mydesign.zip - WinR4	AR.			_		×
<u>File</u> <u>Commands</u> Tools	Fav <u>o</u> rites Opt	io <u>n</u> s <u>H</u> elp				
Add Extract To	Test View	Delete	Find Wizard	Info VirusScan C	Comment 2	SFX »
🗈 📄 mydesign.zi	p - ZIP archive, un	packed size S	56,402 bytes			~
Name	Size	Packed	Туре	Modified	CRC32	
			Local Disk			-
Through.drl	592	205	DRL File	20/8/2017 12:47 AM	48967464	!
BoardOutline.gbr	288	183	GBR File	20/8/2017 12:47 AM	5C52B04C	
Bottom.gbr	21,593	6,573	GBR File	20/8/2017 12:47 AM	8F25287A	
BottomAssy.gbr	189	137	GBR File	20/8/2017 12:47 AM	DC1CDC	
BottomDimension.gbr	1 <b>99</b>	141	GBR File	20/8/2017 12:47 AM	0BED4B77	
BottomMask.gbr	484	245	GBR File	20/8/2017 12:47 AM	059D7813	
BottomPaste.gbr	189	134	GBR File	20/8/2017 12:47 AM	92752156	
BottomSilk.gbr	188	137	GBR File	20/8/2017 12:47 AM	02A2F9AC	
📄 Top.gbr	23,708	7,231	GBR File	20/8/2017 12:47 AM	7E809ABB	
TopAssy.gbr	186	132	GBR File	20/8/2017 12:47 AM	9A647369	
📄 TopDimension.gbr	196	136	GBR File	20/8/2017 12:47 AM	9C12BE44	
📄 TopMask.gbr	1,582	550	GBR File	20/8/2017 12:47 AM	63251885	
📄 TopPaste.gbr	1,281	468	GBR File	20/8/2017 12:47 AM	923FE83C	
📄 TopSilk.gbr	5,727	1,599	GBR File	20/8/2017 12:47 AM	F055F7C7	
<b>D • C</b>			Total 56,40	02 bytes in 14 files		

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.

OSH Pa Processing U	ark pload	
Detected 2 layer board of	1.26x1.51 inches (31.90x38.25mm). \$9.45 for three.	IV-25 Driver
Your upload has finished pro all the individual layers to ma	pressing. Enter the project details below and we'll move on to checking ake sure that they're correct.	
Design notes:		
<ol> <li>Processing mydesig</li> </ol>	gn.zip as DipTrace ZIP file.	
<ol> <li>Removed empty file</li> </ol>	e "BottomSilk.gbr".	
<ol> <li>Removed empty file</li> </ol>	e "BottomPaste.gbr".	
<ul> <li>Your project doesr</li> <li>2 layer board of 1.2</li> </ul>	ı't contain a bottom silk screen. 26x1.51 inches.	
Name	mydesign.zip	
Description	Enter a short description of your project	
Email	Enter your email address (required).	
	Start Over C Cont	tinue 🗲

**NOTE:** 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!

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