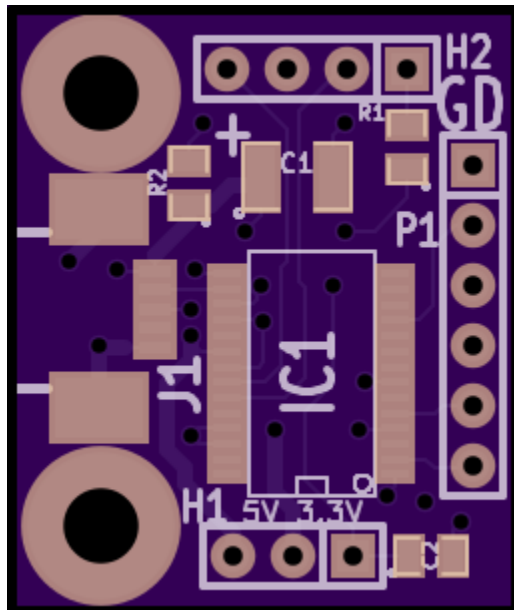


-  Doug Gilliland.
- [Home](#) [Design Rules](#) [Pricing & Specs](#) [Projects](#) [Profile](#) [Log out](#)



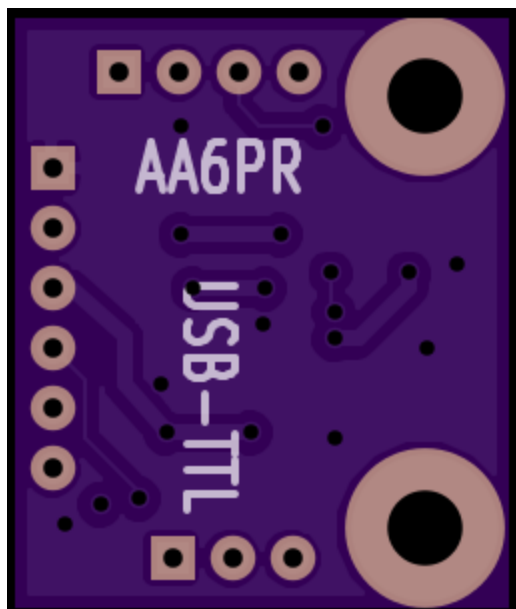
# OSH Park

## PCB Order - Verify your design



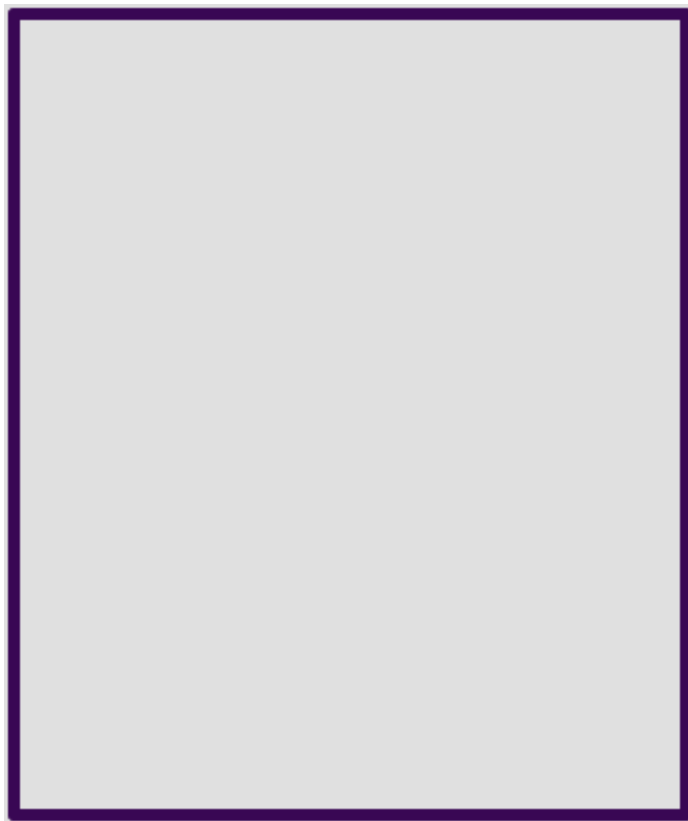
## Board Top

This is an render of what we think your board will look like after fabrication as viewed from the top.



## Board Bottom

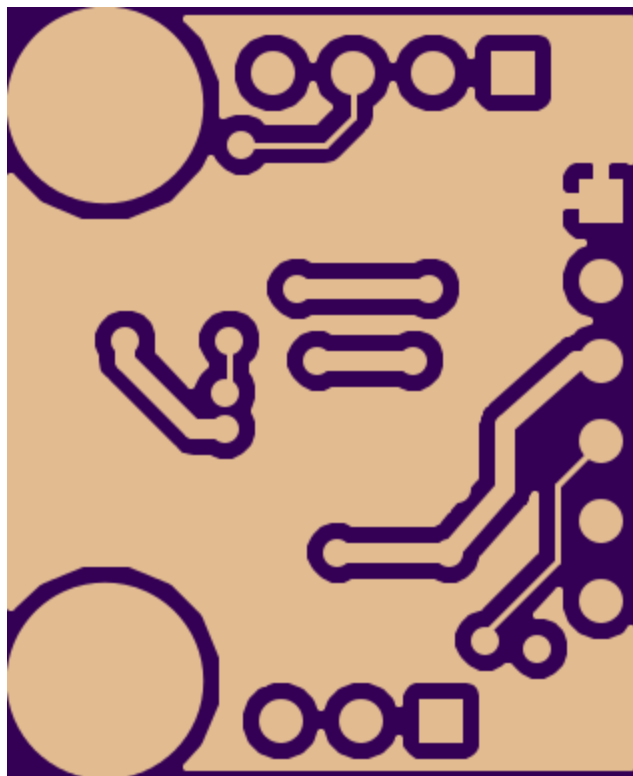
This is an render of what we think your board will look like after fabrication as viewed from the bottom.



Rendered from "Board Outline.ger"

## Board Outline

- The board outline needs to go all the way around the edge of the board such that it's "water tight" (no gaps).
- Non-rectangular board shapes are allowed, but you'll be billed for the smallest rectangle that would enclose your design. So a circle two inches in diameter would be billed at 4 square inches.
- Cutouts aren't officially supported, but the fab has been doing them pretty regularly as long as they're drawn on the board outline layer, and are at least 100 mils wide.
- To try making a cutout, draw the outline of the cutout on the board outline layer, or draw the path you'd like the milling tool to make using a 0.1" wide line. Cutouts won't be plated.



Rendered from "Bottom Layer.ger"

## Bottom Layer

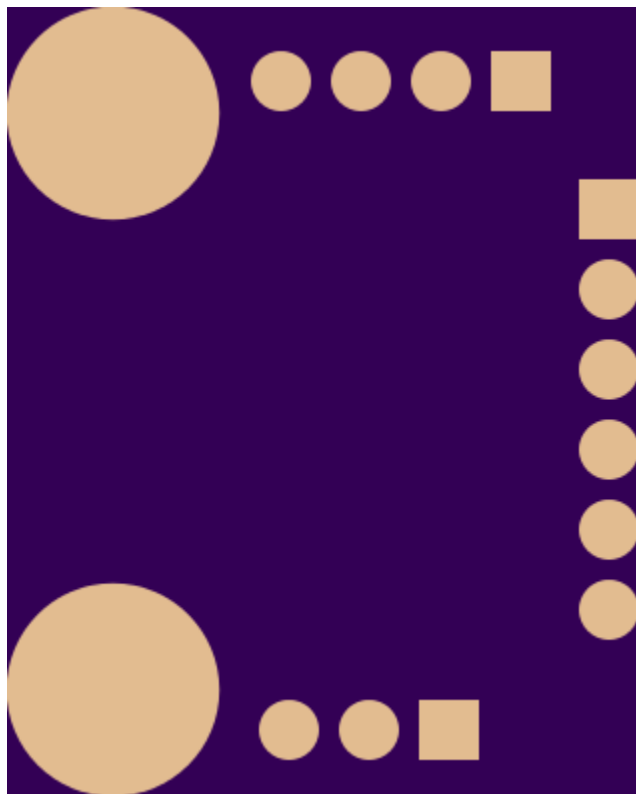
- This is the bottom copper layer of your board.



Rendered from "Bottom Silk Screen.ger"

## Bottom Silk Screen

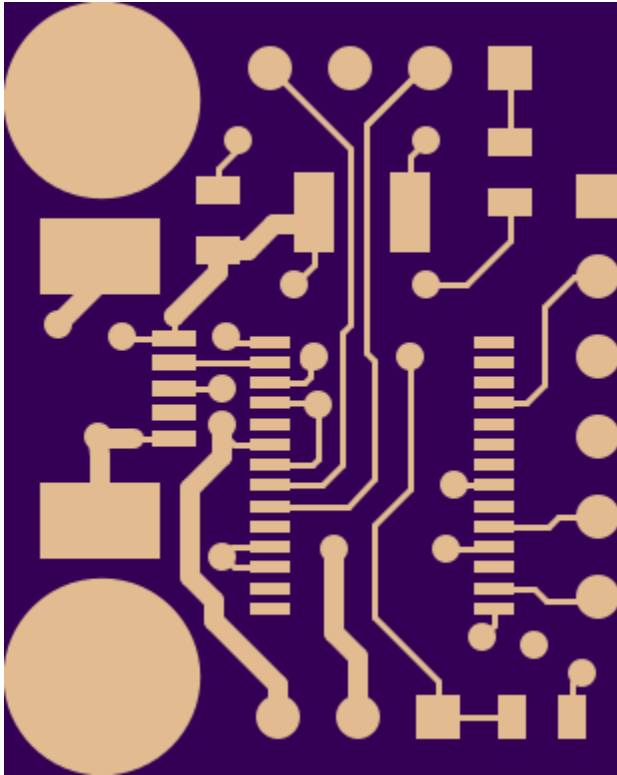
- The silkscreen is put on with what is basically a 200 dpi printer. Lines thinner than 5 mils will be fattened to 5 mils before printing.
- Try to keep your silkscreen inside the board outline. It's okay if it goes out of the board outline, but it will be trimmed with sometimes unpredictable results.
- The fab will automatically remove any silkscreen that crosses drilled holes or exposed metal.



Rendered from "Bottom Solder Mask.ger"

## Bottom Solder Mask

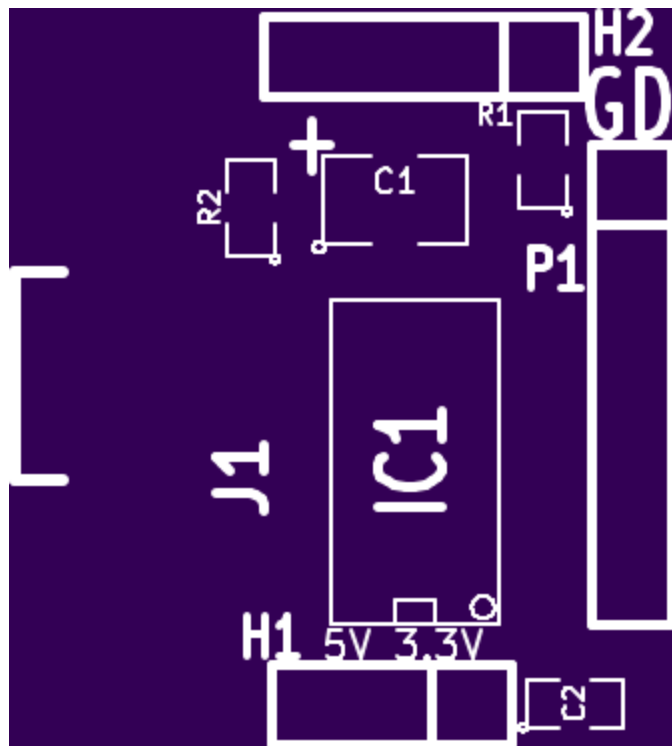
- Soldermask layers are "negative" layers. This layer really designates where there *shouldn't* be solder mask. If you draw on the soldermask layer ("tStop" and "bStop" in Eagle), those areas won't have soldermask.
- If you don't provide a soldermask layer here, this entire side of the board will be coated in soldermask. You probably don't want this.



Rendered from "Top Layer.ger"

## Top Layer

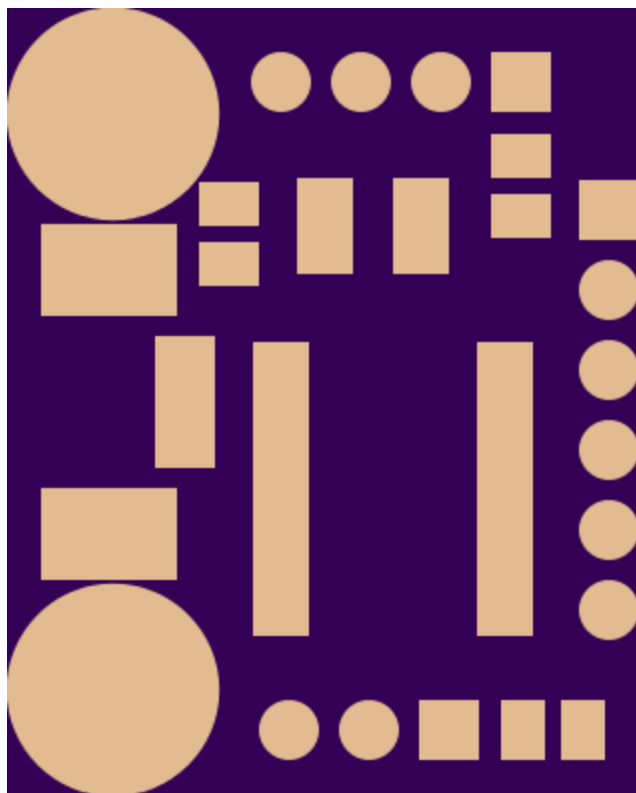
- This is the top layer of copper on your PCB.



Rendered from "Top Silk Screen.ger"

## Top Silk Screen

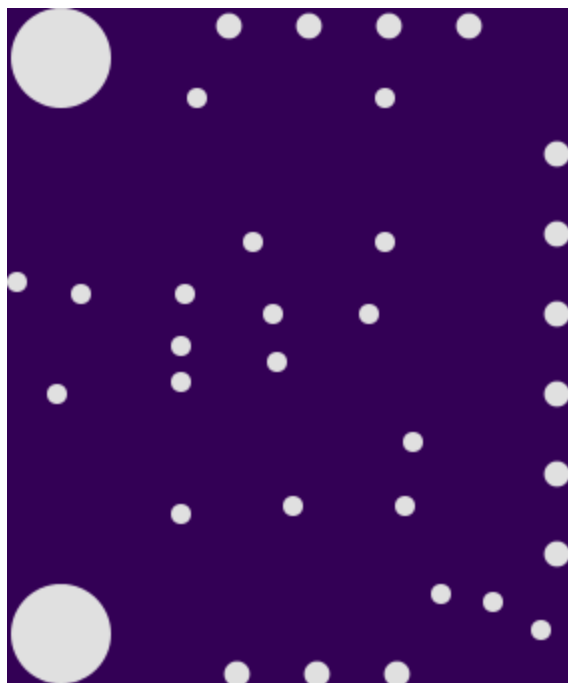
- The silkscreen is put on with what is basically a 200 dpi printer. Lines thinner than 5 mils will be fattened to 5 mils before printing.
- Try to keep your silkscreen inside the board outline. It's okay if it goes out of the board outline, but it will be trimmed with sometimes unpredictable results.
- The fab will automatically remove any silkscreen that crosses drilled holes or exposed metal.



Rendered from "Top Solder Mask.ger"

## Top Solder Mask

- Soldermask layers are "negative" layers. This layer really designates where there *shouldn't* be solder mask. If you draw on the soldermask layer ("tStop" and "bStop" in Eagle), those areas won't have soldermask.
- If you don't provide a soldermask layer here, this entire side of the board will be coated in soldermask. You probably don't want this.



Rendered from "Drills.xln"

## Drills

- Your drill file needs to be in text "NC Drills" or "Excellon" format, generated with "2:4" precision, and with "no zero suppression".
- Make sure the center of your drill hits are all inside the board outline. Anything outside of the board outline is automatically removed.
- Overlapping drill hits aren't allowed.
- Minimum drill size is 13 mils. Maximum is 360 mils.
- Plated slots aren't supported.

Start Over ↶

Approve and Continue →



Designed and developed by [Sociable Limited](#).