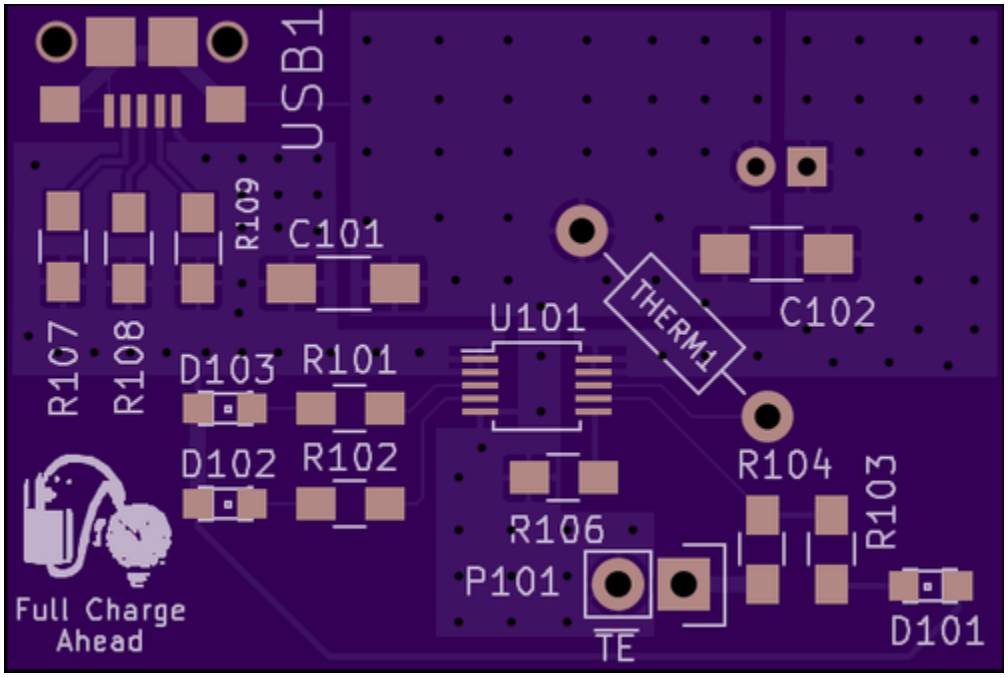
- Steve Mayze.
- Home Shared Projects Design Rules Pricing & Specs Support Projects Cart Profile Log out

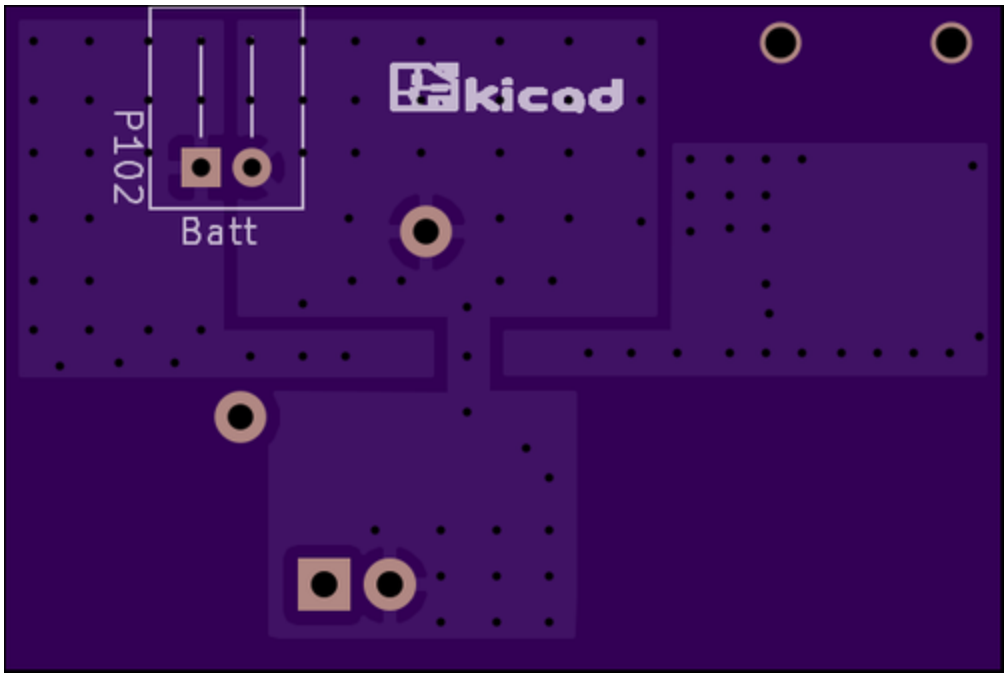
OSH Park

PCB Order - Verify your design



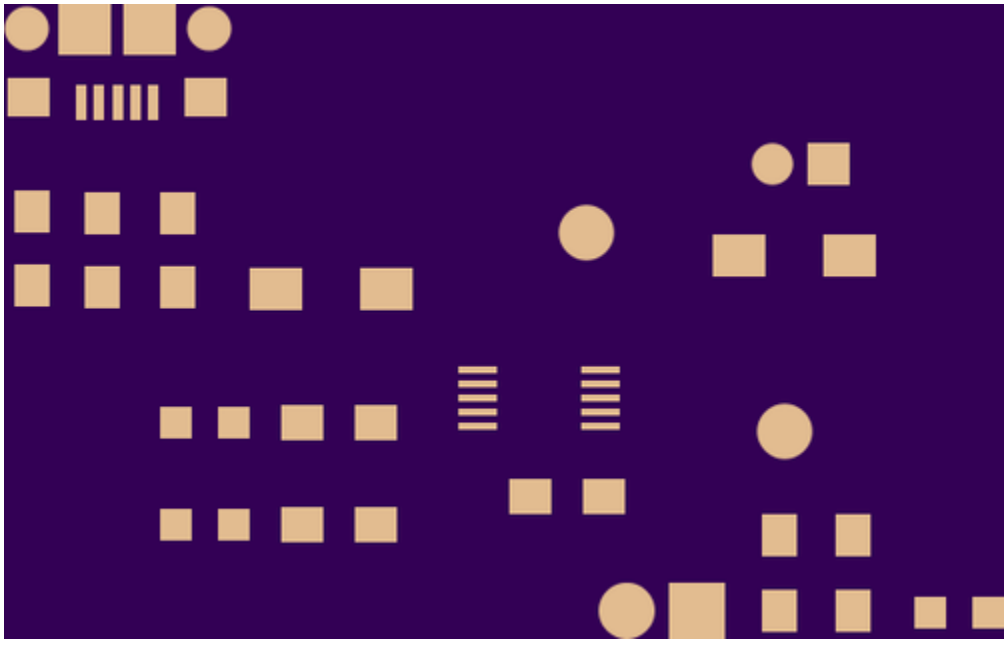
Board Top

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



Board Bottom

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



Rendered from "FullChargeAhead-F_Mask.gts"

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 "FullChargeAhead-B_Mask.gbs"

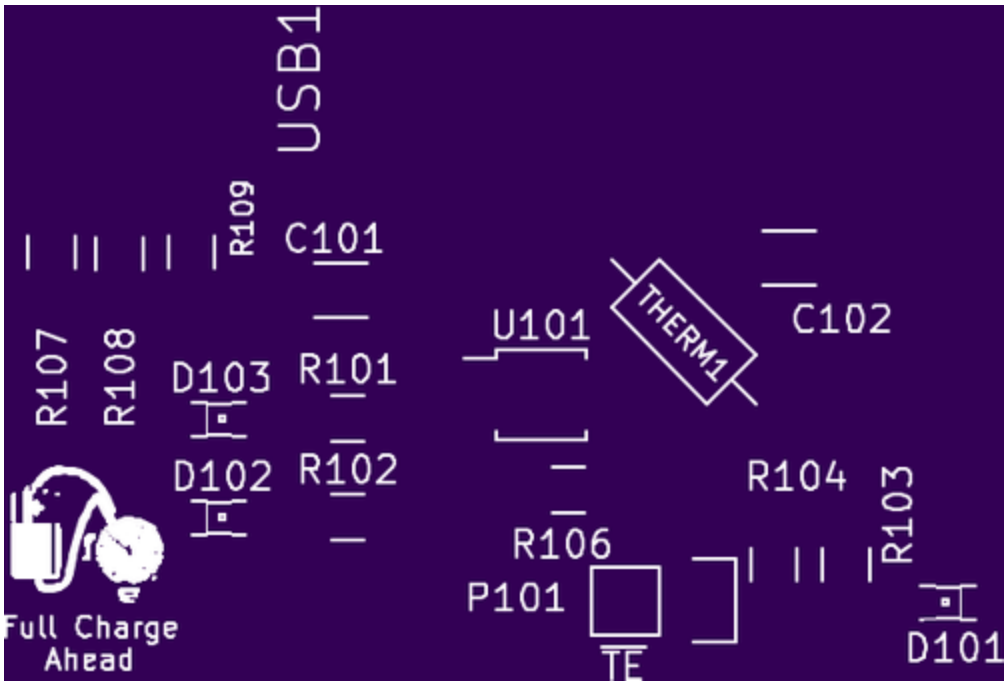
Bottom Solder Mask

- 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 "FullChargeAhead-Edge_Cuts.gbr"

Board Outline

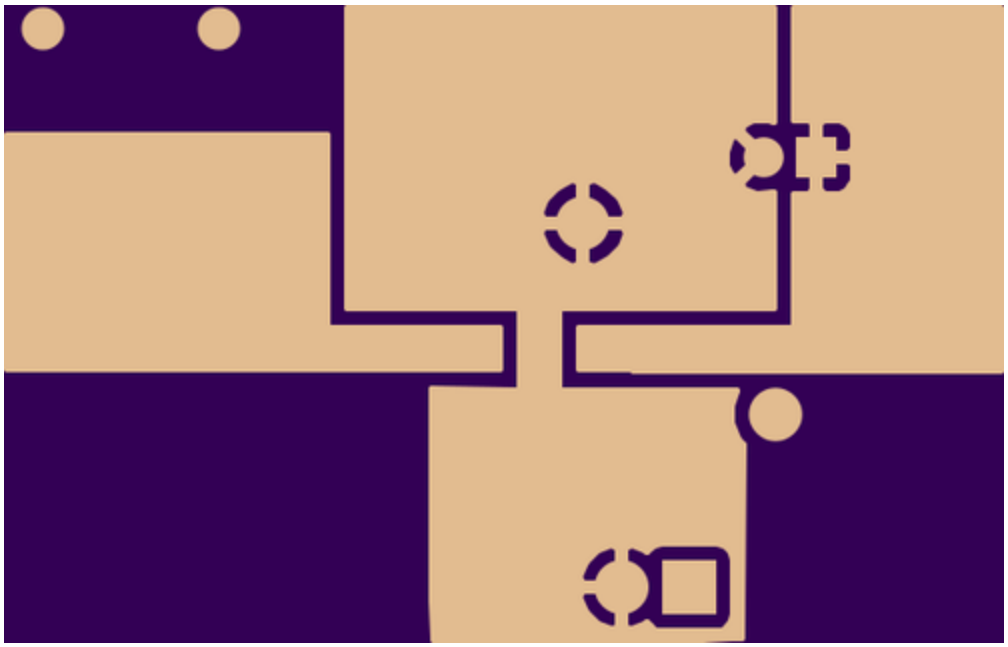
- The board outline needs to go all the way around the edge of the board such that it's "water tight" (no gaps).
- Non-rectangular board shapes are allowed, but you'll be billed for the smallest rectangle that would enclose your design. So a circle two inches in diameter would be billed at 4 square inches.
- 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 "FullChargeAhead-F_SilkS.gto"

Top Silk Screen

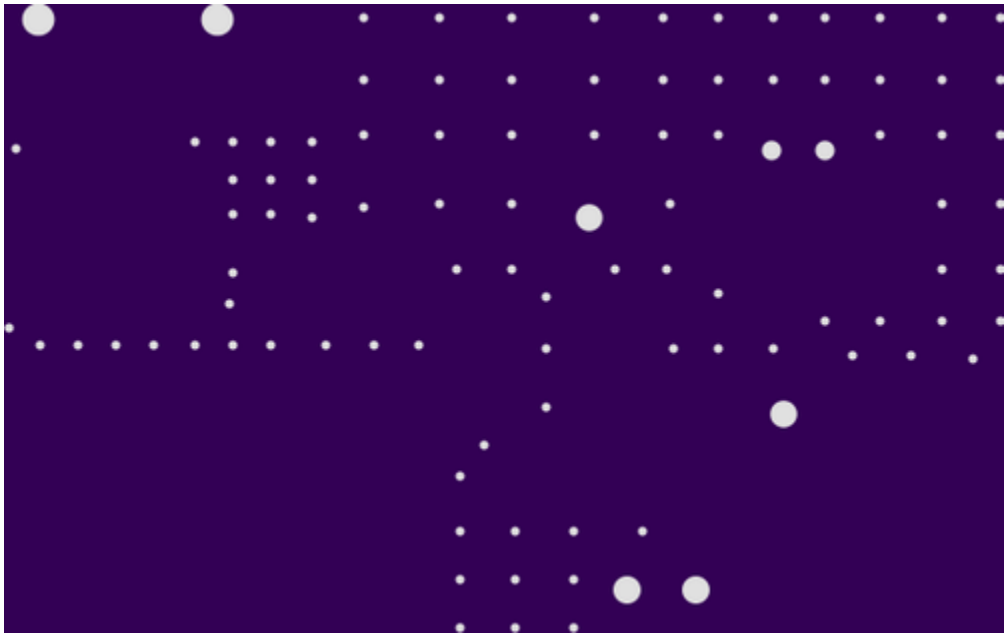
- The silkscreen is put on with what is basically a 200 dpi printer. Lines thinner than 5 mils will be fattened to 5 mils before printing.
- Try to keep your silkscreen inside the board outline. It's okay if it goes out of the board outline, but it will be trimmed with sometimes unpredictable results.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



Rendered from "FullChargeAhead-B_Cu.gbl"

Bottom Layer

- This is the bottom copper layer of your board.



Rendered from "FullChargeAhead.drl"

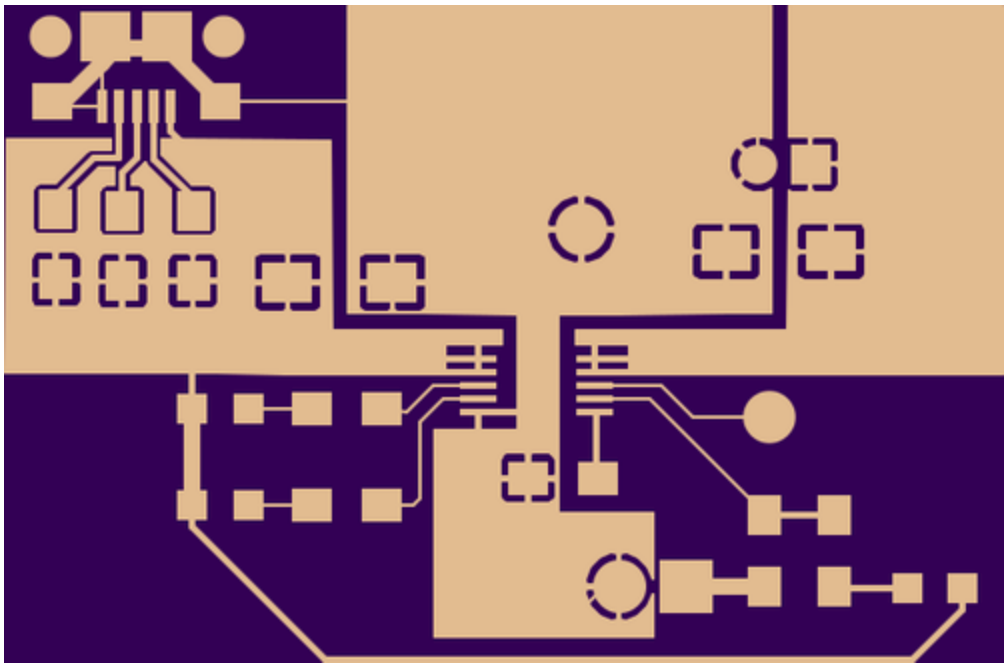
Drills

- 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.

Rendered from "FullChargeAhead-B_SilkS.gbo"

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 "FullChargeAhead-F_Cu.gtl"

Top Layer

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

Start Over ↺ | Approve → | Approve and Order →

Designed and developed by Resistor.