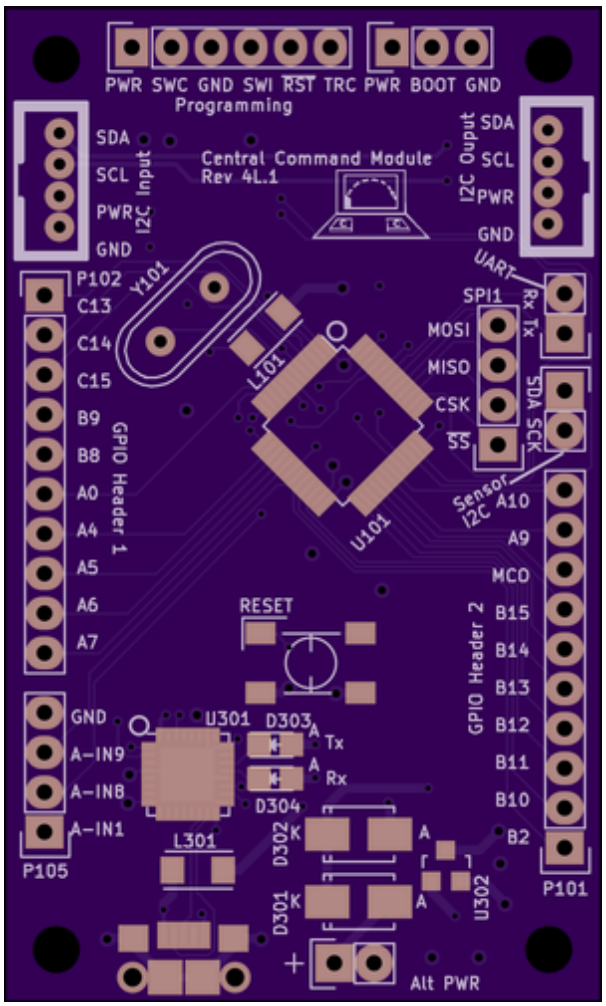
- Steve Mayze.
- Home Shared Projects Design Rules Pricing & Specs Support Projects Order History Cart Profile Log out

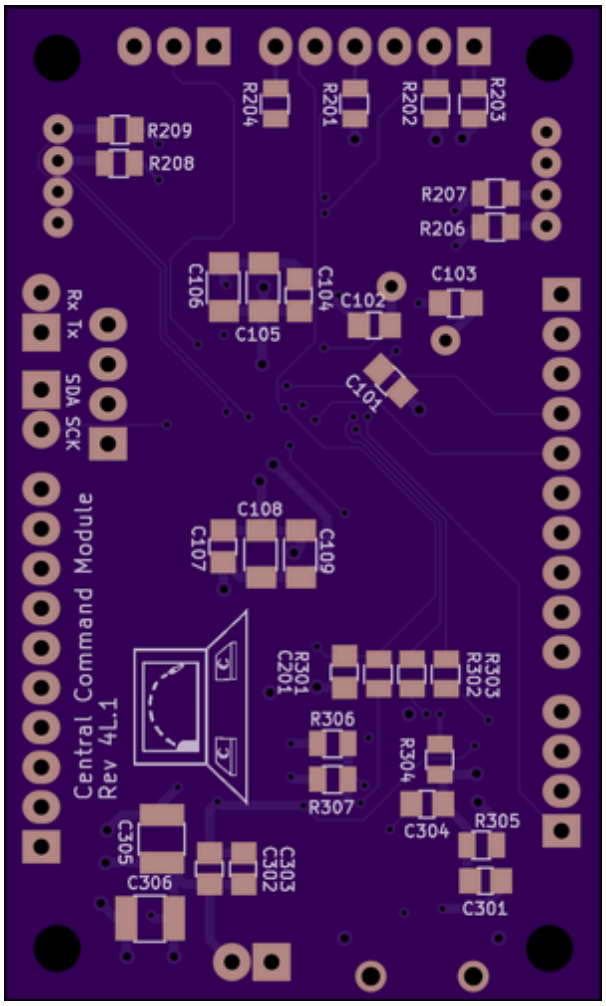
OSH Park

PCB Order - Verify your design



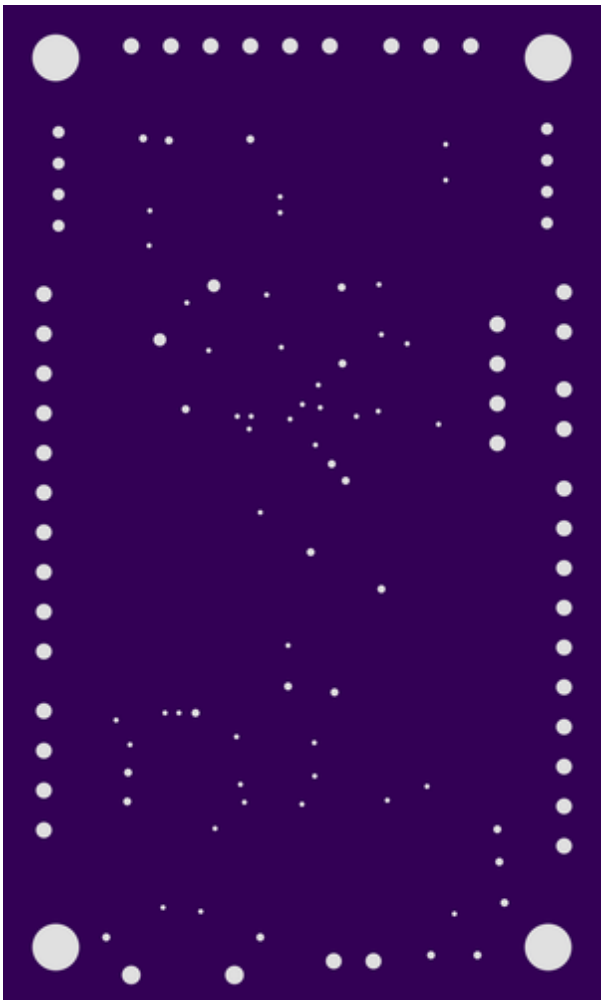
Board Top

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



Board Bottom

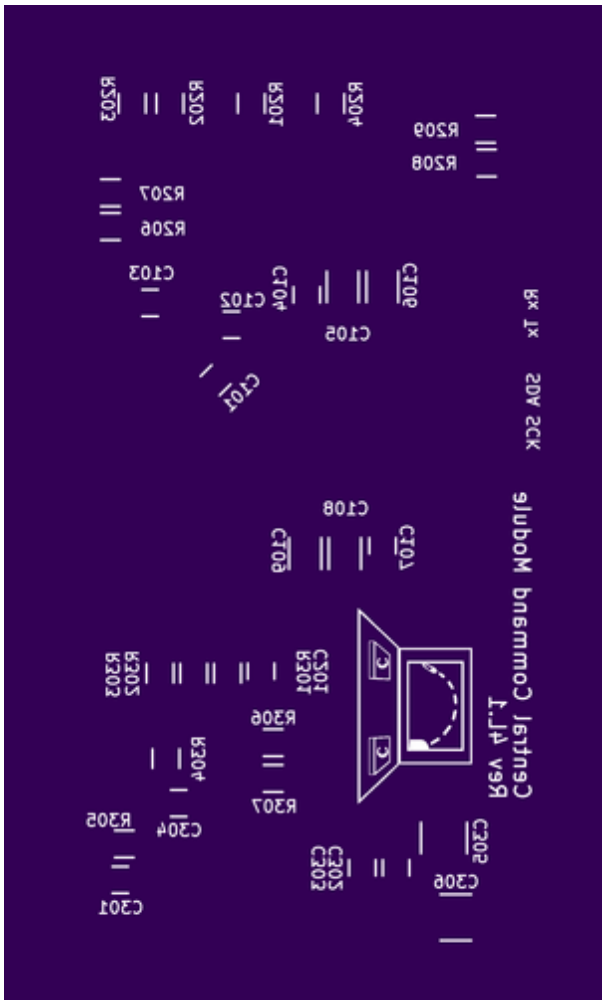
This is a render of what we think your board will look like after fabrication as viewed from the bottom.



Rendered from ""

Drills

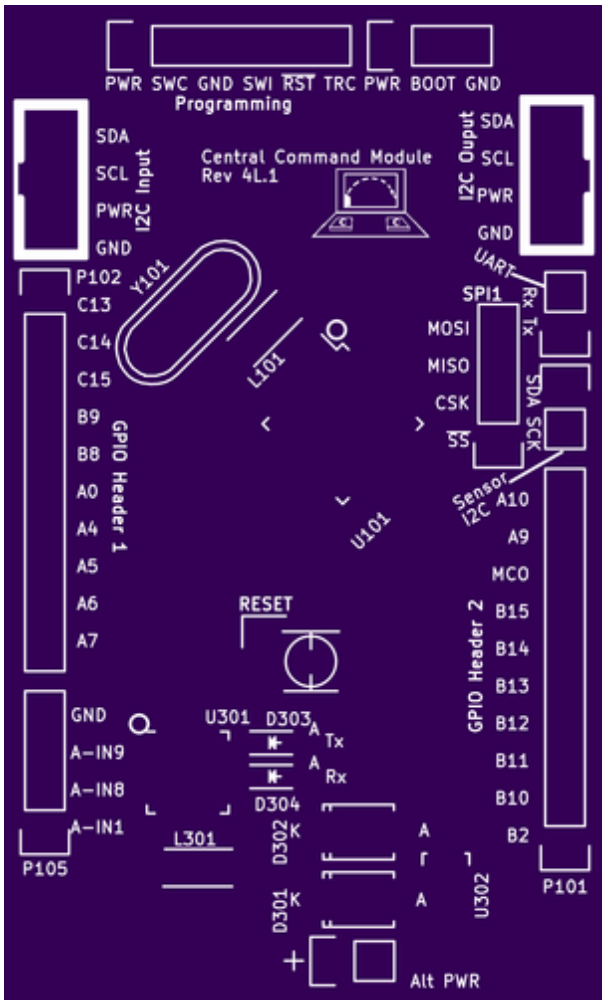
- Your drill file needs to be in text "NC Drills" or "Excellon" format, generated with "2:4" precision, and with "no zero suppression".
- Make sure the center of your drill hits are all inside the board outline. Anything outside of the board outline is automatically removed.
- Overlapping drill hits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



Rendered from ""

Bottom Silk Screen

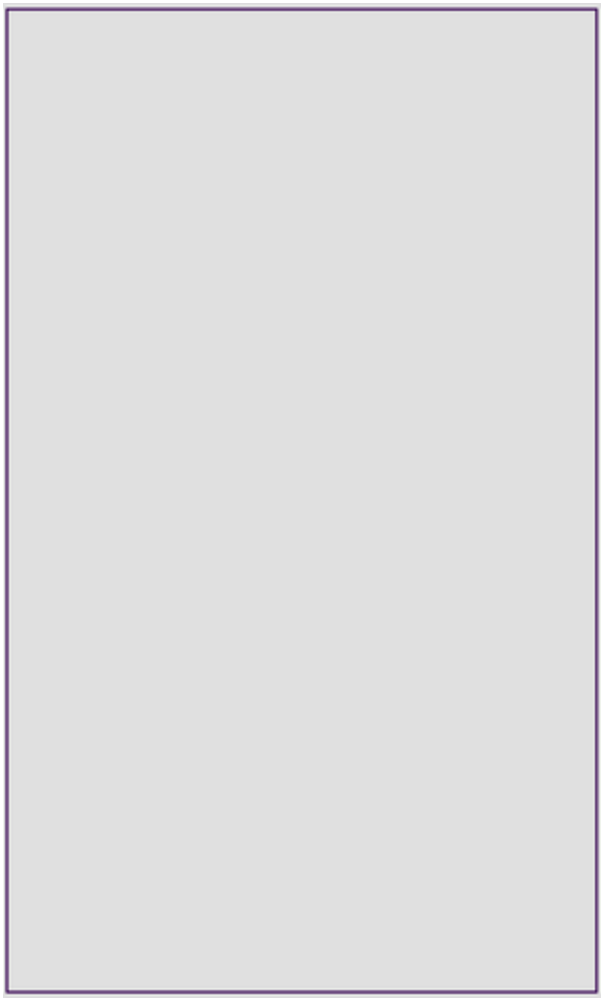
- The silkscreen is put on with what is basically a 200 dpi printer. Lines thinner than 5 mils will be fattened to 5 mils before printing.
- Try to keep your silkscreen inside the board outline. It's okay if it goes out of the board outline, but it will be trimmed with sometimes unpredictable results.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



Rendered from ""

Top Silk Screen

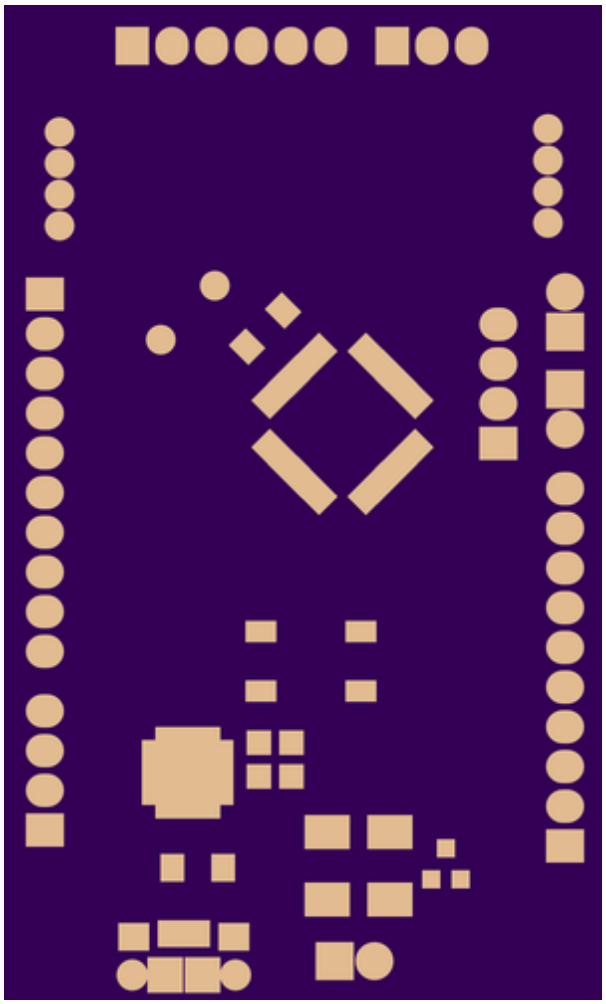
- The silkscreen is put on with what is basically a 200 dpi printer. Lines thinner than 5 mils will be fattened to 5 mils before printing.
- Try to keep your silkscreen inside the board outline. It's okay if it goes out of the board outline, but it will be trimmed with sometimes unpredictable results.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



Rendered from ""

Board Outline

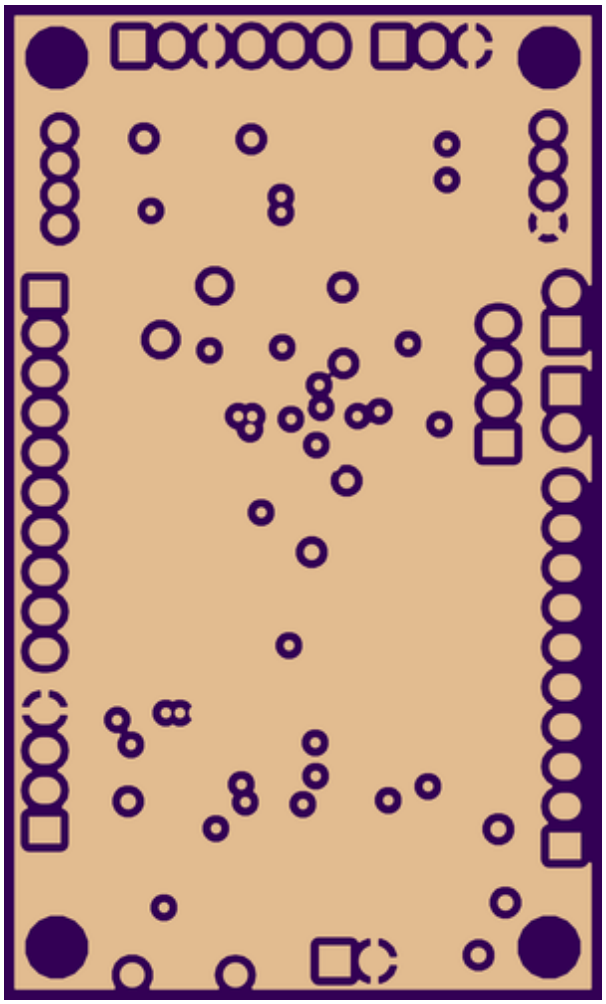
- The board outline needs to go all the way around the edge of the board such that it's "water tight" (no gaps).
- Non-rectangular board shapes are allowed, but you'll be billed for the smallest rectangle that would enclose your design. So a circle two inches in diameter would be billed at 4 square inches.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

Top Solder Mask

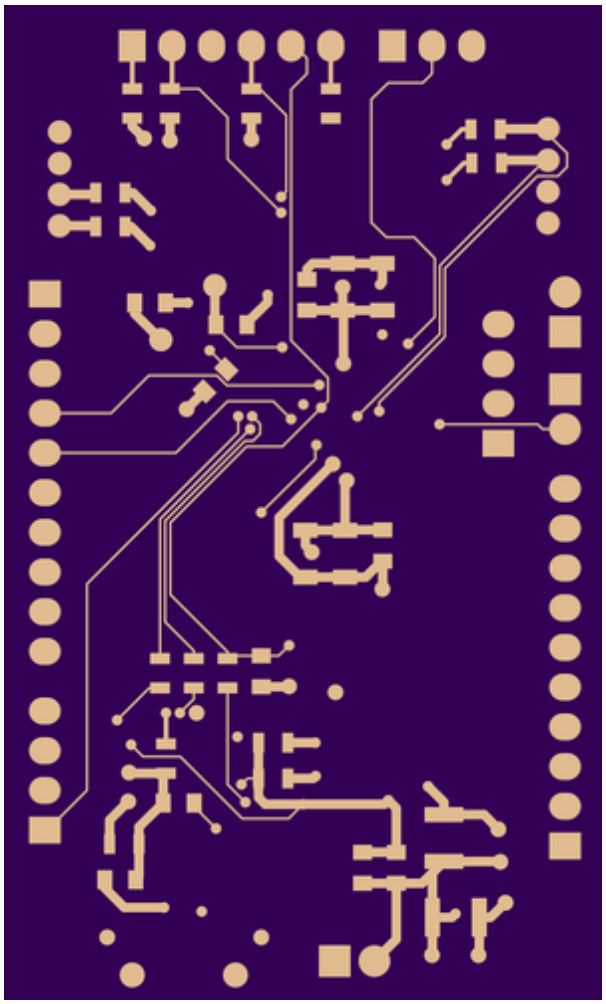
- Soldermask layers are "negative" layers. This layer really designates where there *shouldn't* be solder mask. If you draw on the soldermask layer ("tStop" and "bStop" in Eagle), those areas won't have soldermask.
- If you don't provide a soldermask layer here, this entire side of the board will be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



Rendered from ""

Internal Plane 1

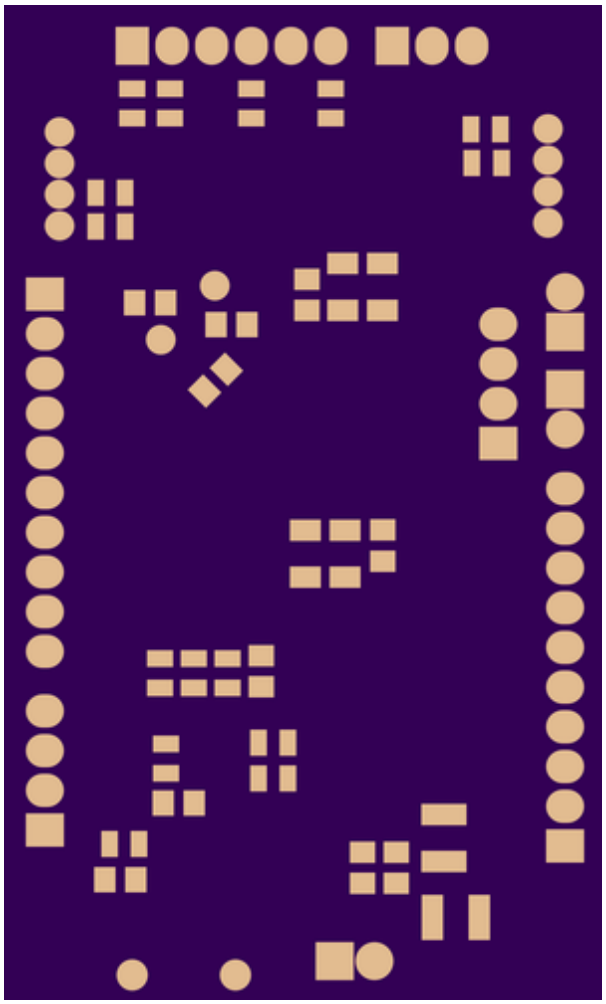
- Internal layers need to be generated as "positive". Lines on this layer represent copper, not absence of copper. Some CAD packages will generate them as "negative" if you define them as power planes. To work around it, make the layer a signal layer, then use a copper pour to define the power plane.



Rendered from ""

Bottom Layer

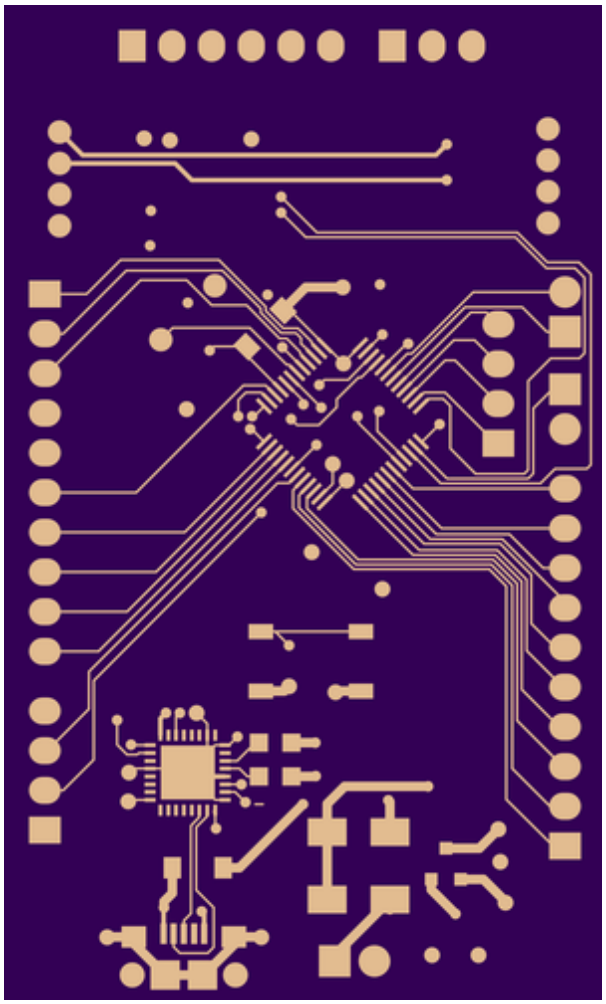
- This is the bottom copper layer of your board.



Rendered from ""

Bottom Solder Mask

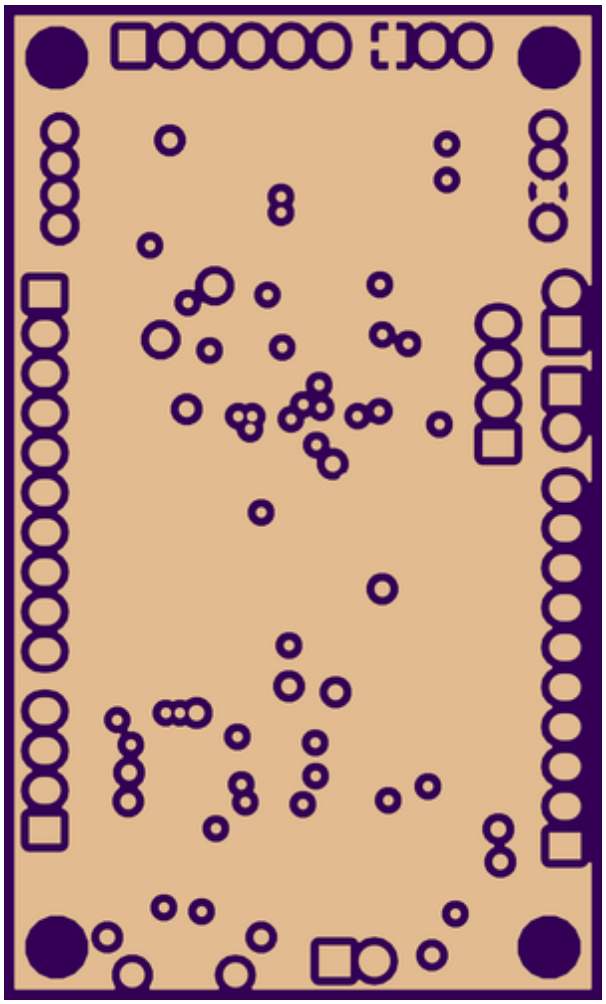
- Soldermask layers are "negative" layers. This layer really designates where there *shouldn't* be solder mask. If you draw on the soldermask layer ("tStop" and "bStop" in Eagle), those areas won't have soldermask.
- If you don't provide a soldermask layer here, this entire side of the board will be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



Rendered from ""

Top Layer

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



Rendered from ""

Internal Plane 2

- Internal layers need to be generated as "positive". Lines on this layer represent copper, not absence of copper. Some CAD packages will generate them as "negative" if you define them as power planes. To work around it, make the layer a signal layer, then use a copper pour to define the power plane.

Start Over ↺ | Approve → | Approve and Order →

Designed and developed by Resistor.