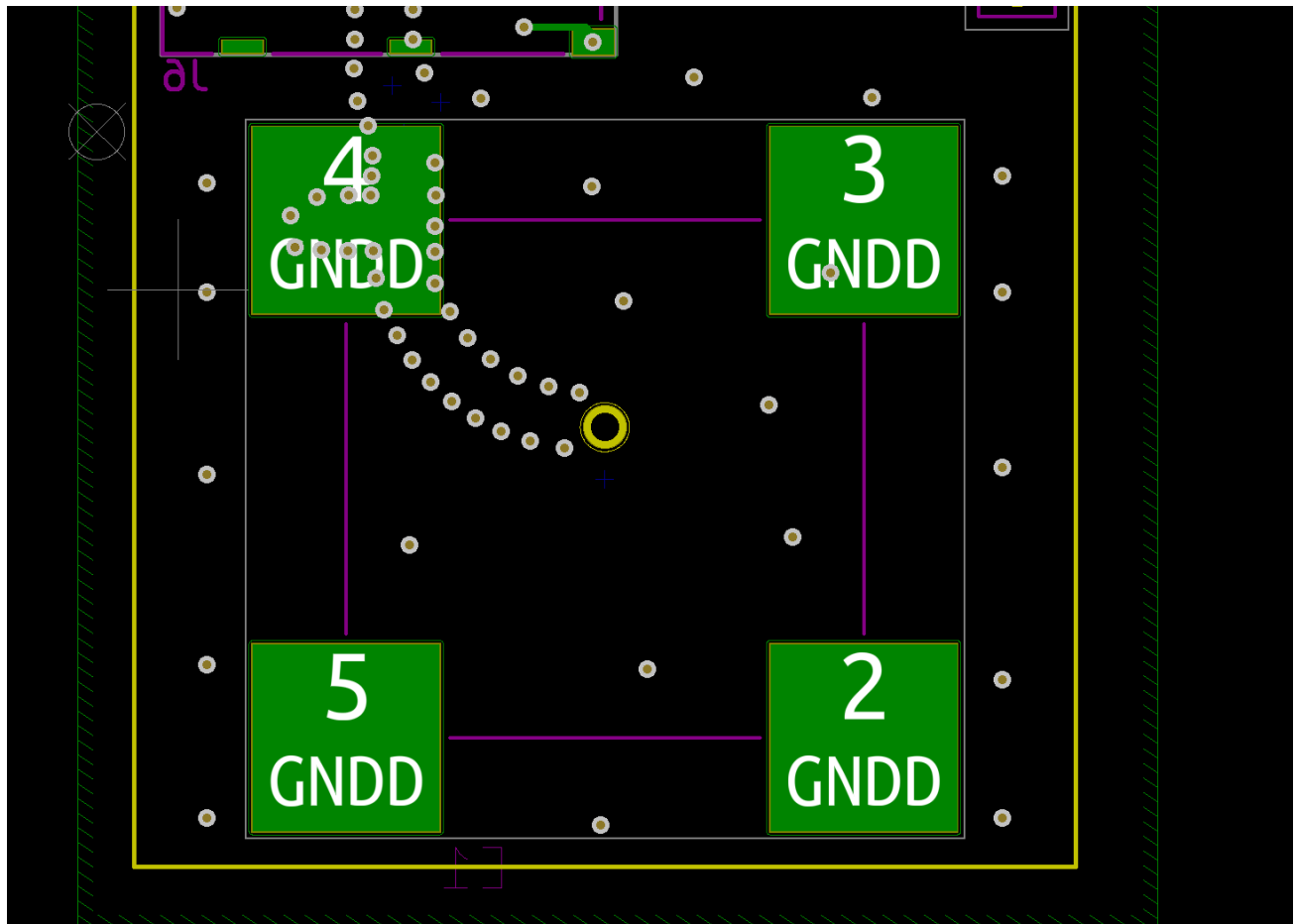


## GPS patch antenna pcb design

Asked 2 years ago Modified 2 years ago Viewed 575 times

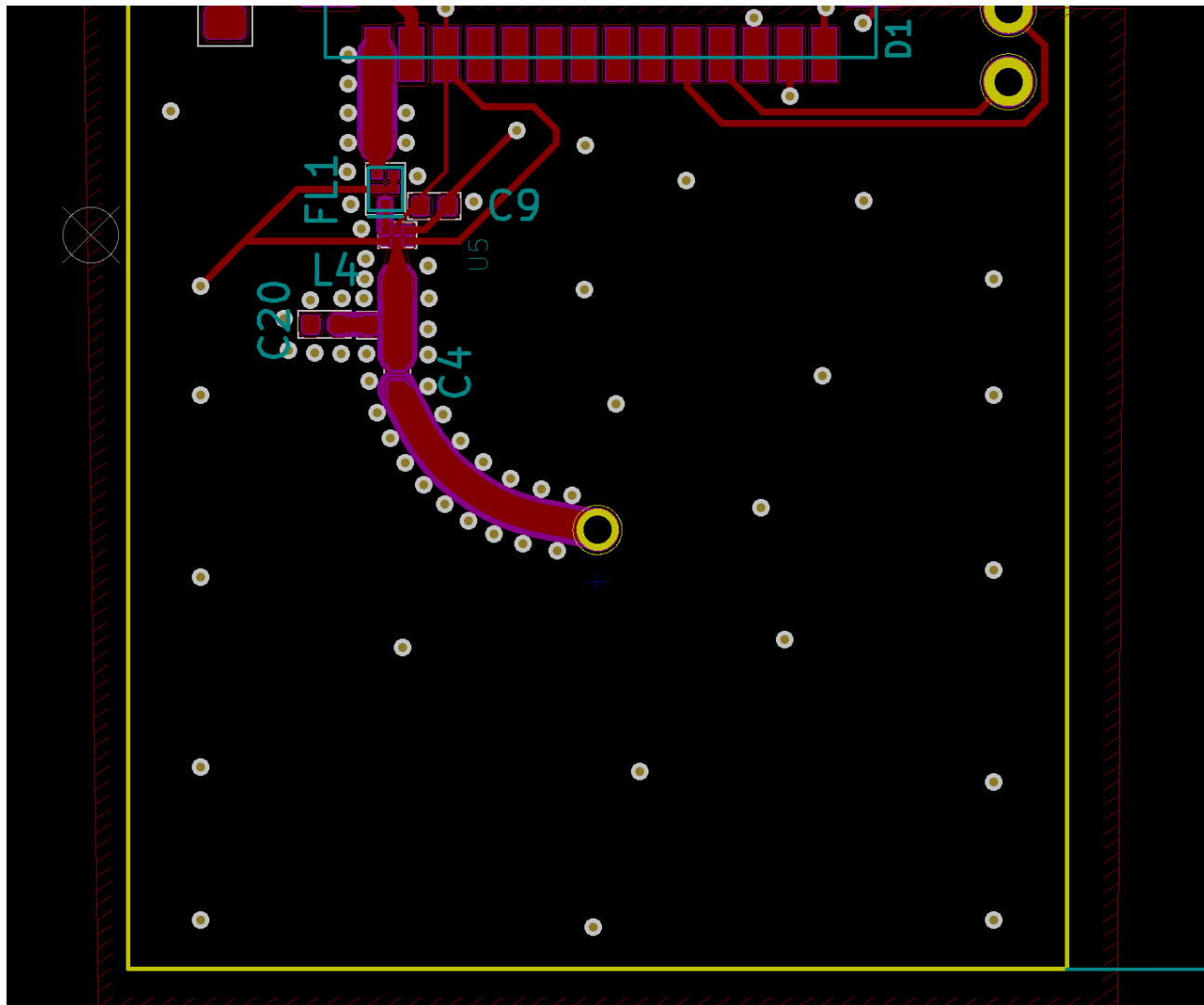
could you please review my layout?

0 It's the top side where the antenna is actually placed.



It's bottom side, here I've placed a matching network, LNA (U5) and SAW (FL1). The length of the trace between the antenna's output and IC input is about 15mm (without capacitors and LNA/Filter's size).

I used kicad's calculator to get trace width - 1mm (coplanar trace, dielectric height - 1.6). Both sides of the board are covered with a ground polygon.



It's a recommended layout from the antenna's datasheet. But I've used the matching network from the LNA's datasheet.

## Recommended Layout

The recommended printed circuit board (PCB) layout for the GNSSCP-TH18L1 is demonstrated by the AEK-GNSSCP-TH18L1 evaluation board shown in Figure 9. Contact Linx for availability of PCB layout design files.

The recommended layout includes a matching network, ground plane and PCB transmission line from the antenna to the matching network, and to the connector or radio circuitry. The connector used for the evaluation board is optional, the transmission line may be run directly to the radio if on the same PCB.

Linx recommends inclusion of at least a 3-element, surface mount pi matching network of two parallel capacitors, (C1, C2) and one serial inductor, (L1) in all designs (Figure 10). Surface mount components should be 0603 size. 0402 size components are also supported. The GNSSCP series antennas, as designed, do not require matching, but matching may improve end-product antenna performance depending on the effects of the enclosure, PCB and other electronic components. If no matching is necessary, the serial element may be populated with a zero-ohm resistor and no components in the two capacitor positions. This is the configuration of the Linx evaluation board as supplied. Linx believes in wireless made simple® and offers matching network design support.

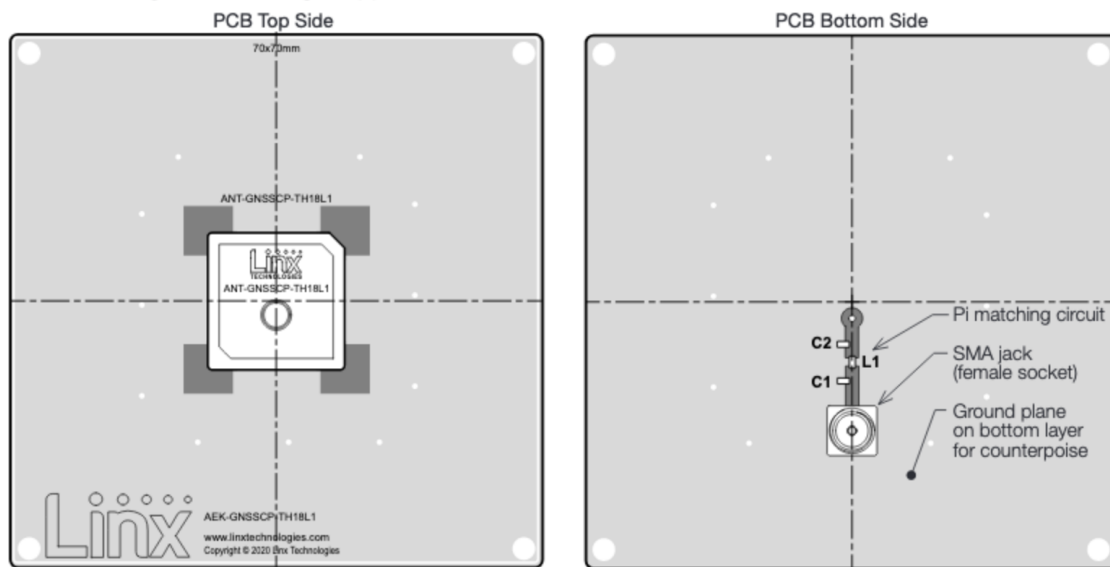
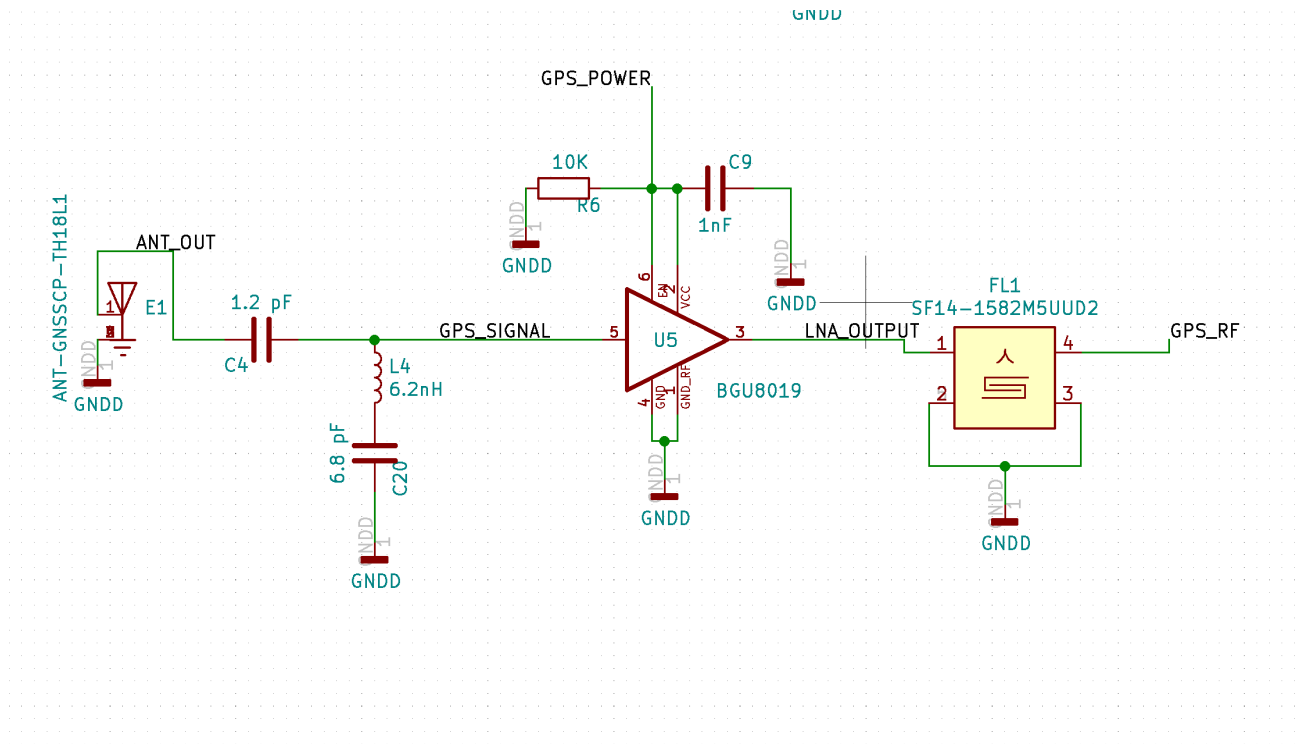


Figure 9. ANT-GNSSCP-TH18L1 Recommended Layout

Here is schematic



pcb rf antenna layout gps

Share Cite Follow

asked Nov 18, 2020 at 8:12



Denis Jelesniak

43 5

Where's the ground plane flood? – [Andy aka](#) Nov 18, 2020 at 9:05

Generally speaking you have a problem with the following : – [citizen](#) Nov 18, 2020 at 10:27

1. You've got some "long" traces for your LNA VCC etc that you should be able to reduce. If you're PCB is only two layer you may want to consider going to 4 layers with a layer-2 dedicated ground plane. The other inner layer could be a power plane that you can use to deliver the power supply to the semiconductors with vias. – [citizen](#) Nov 18, 2020 at 10:31

2. Your ground planes on top and bottom are missing and should have been added for clarity, but if one assumes they are there, it seems the via stitching is too close to the transmission line and should be furthered out for the micro-strip line – [citizen](#) Nov 18, 2020 at 10:33


Linx offers some general advice that you may easily obtain from them directly. You can send an email to Linx and they will offer some advice that will be more specific to their antennas etc. – [citizen](#) Nov 18, 2020 at 10:34

Thanks! Do you recommend to move LNA and SAW closer to an input pin? How close the via fence should be to the trace line? – [Denis Jelesniak](#) Nov 18, 2020 at 10:35

---

It really depends how many layers you have to work with. Yes you could move the LNA and SAW closer but it may not be enough as you have some of those other long traces close to the LNA etc. For micro-strip line, if you go to 4 layers anyway, its width will reduce as the ground plane will be more like 0.4mm thick (depends on stack-up chosen). Then you can do a bit of a google search and find various articles on this, but it depends on micro-strip width ... – [citizen](#) Nov 18, 2020 at 10:40

---

I use 2 layer board (to reduce the final cost). Could I call my trace a microstrip? It's just an ordinary trace with an expanded soldering mask Which tools could be used to check the prototype board? How could I check that my design is good enough? :- ) – [Denis Jelesniak](#) Nov 18, 2020 at 10:52 

---