RF PCB ROUTING

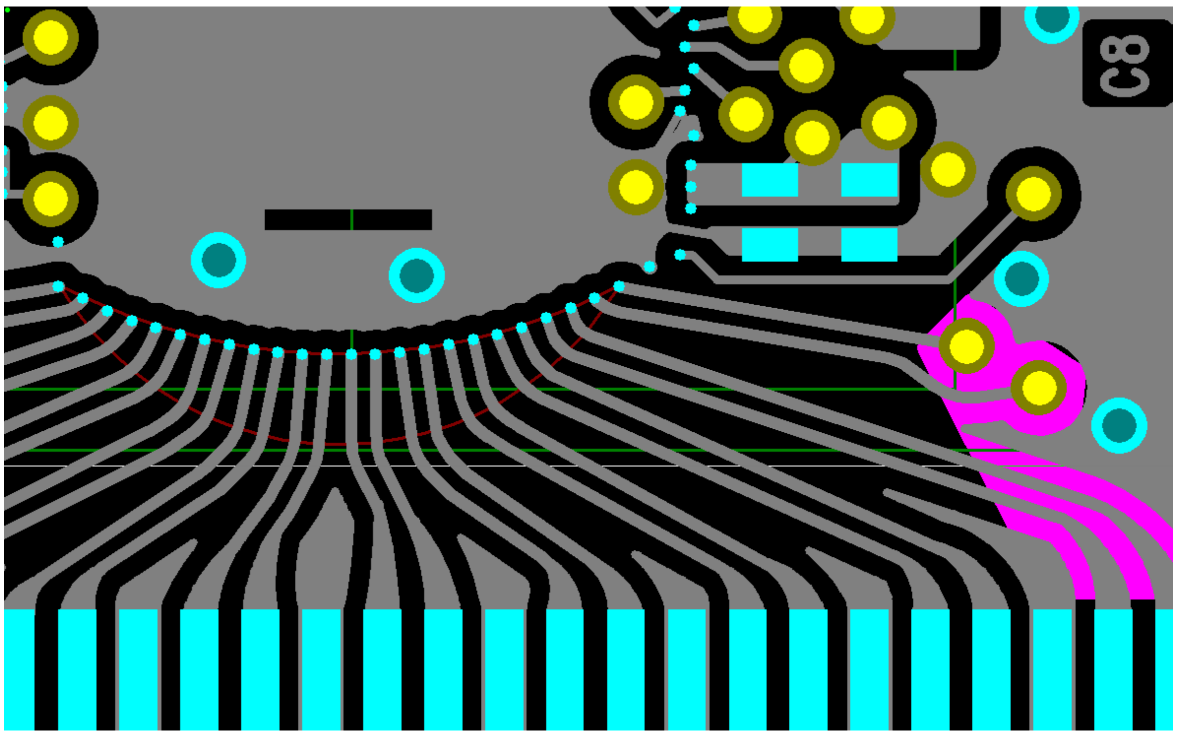
"A battle between high frequency considerations and manufacturability."

RTP Chapter of the IPC

July 18,2012

Introduction:
What is RF PCB routing?

Even "digital" PCBs tend to be RF PCBs nowadays .



Radio Frequency (RF) PCB routing is nothing more than "regular" routing, with an emphasis shift to signal integrity and isolation .

This presentation, intended for the PCB designer, seeks to illuminate some of the common pitfalls inherent in the layout of typical RF circuitry.

Agenda

1. Brief review of the following topics:

    - Impedance
    - Transmission line types in PCB's
    - GND
    - Isolation
    - Signal loss

2. A sample RF Downconverter PCB with a number of errors will be critiqued, and corrected.

3. Miscellaneous tips and tricks to improve layout productivity while maintaining signal integrity.

IMPEDANCE

A very brief review of signal reflection, transmission lines

A gross oversimplification of impedance

Impedance of a PCB trace is driven by the following PCB parameters.

- Substrate Material ϵ_r. (There are laminates tailored for RF PCBs)
- Layer Stackup and pad/trace geometry, which affects these electromagnetic parameters:
  - Inductance per unit length

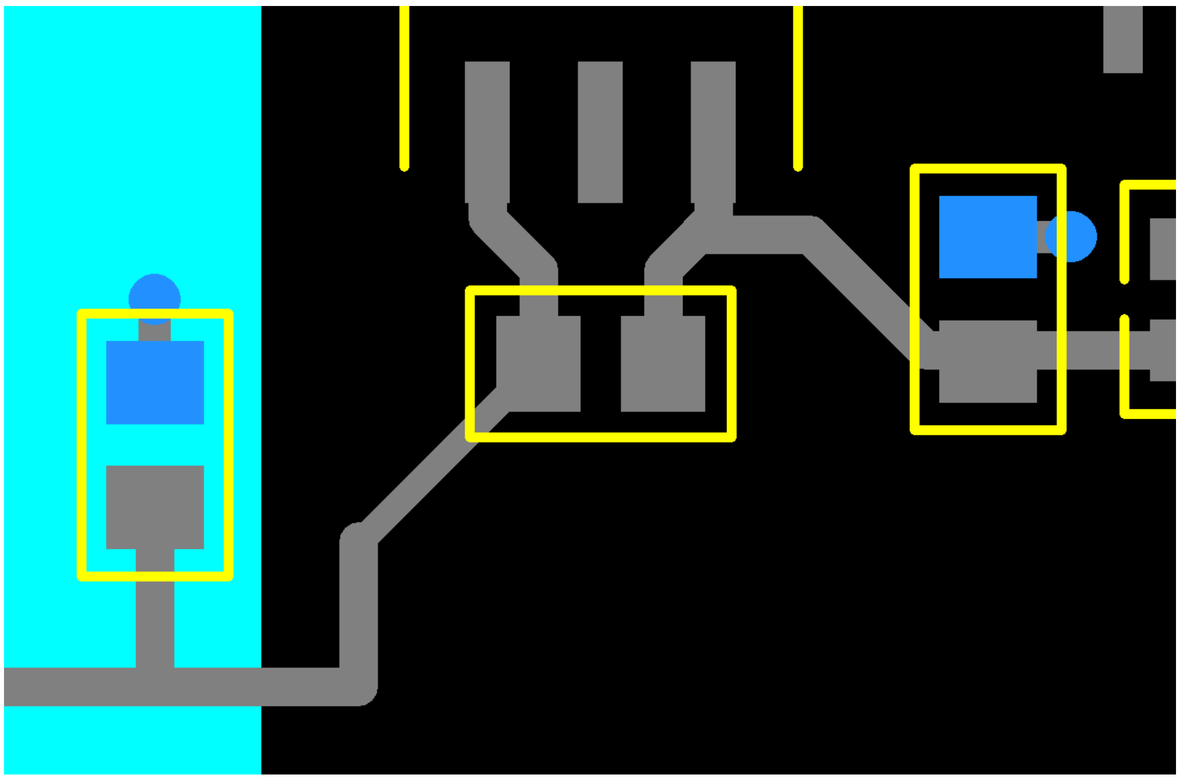
    Impedance varies with inductance
  - Capacitance per unit length

    impedance varies inversely with capacitance

Several good field solvers can calculate trace impedance reasonably

Impedance discontinuities

Impedance problems in a PCB layout

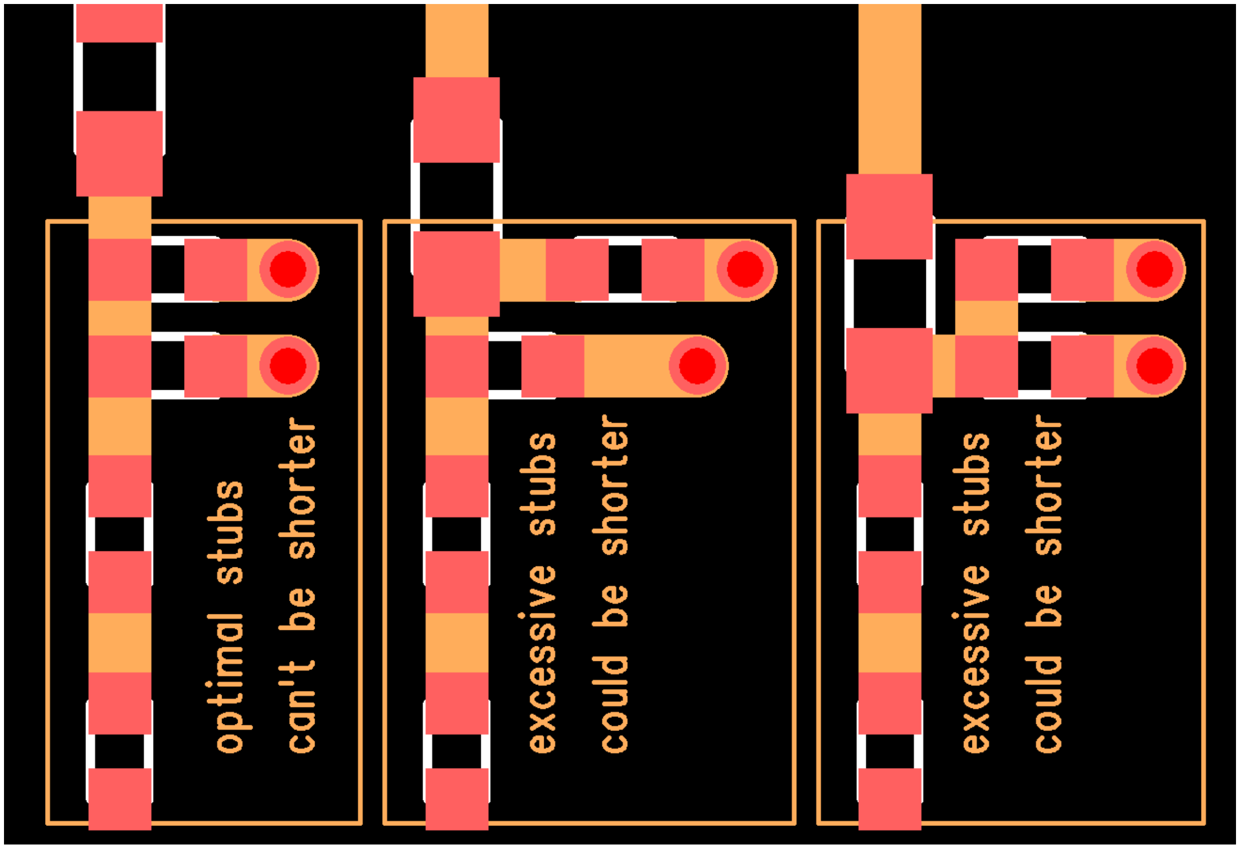


Any abrupt change in trace geometry, or its relation to PCB stackup, can cause an impedance discontinuity.

Stubs, plane breaks, missing return current paths, harsh trace angles, changes in trace width, all affect impedance, and impair signal flow by causing reflections.

Example stubs, shunt parts

Shunt components, with stub traces.

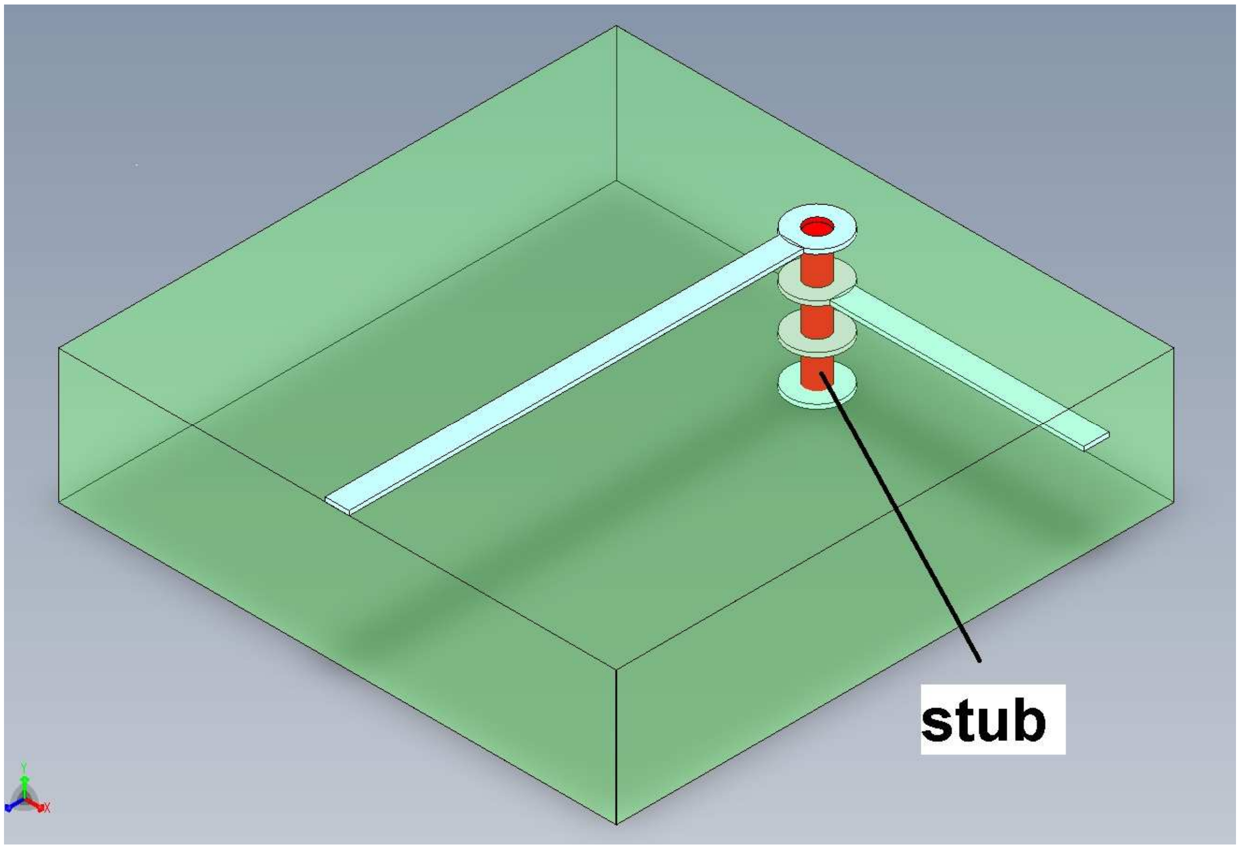


Stubs are inevitable. Some are easy to spot, some hard. They inevitably add some parasitic inductance and/or parasitic capacitance.

Parasitics ruin filters. Keeping stubs short mitigates this.

Example , via stubs

Via stub

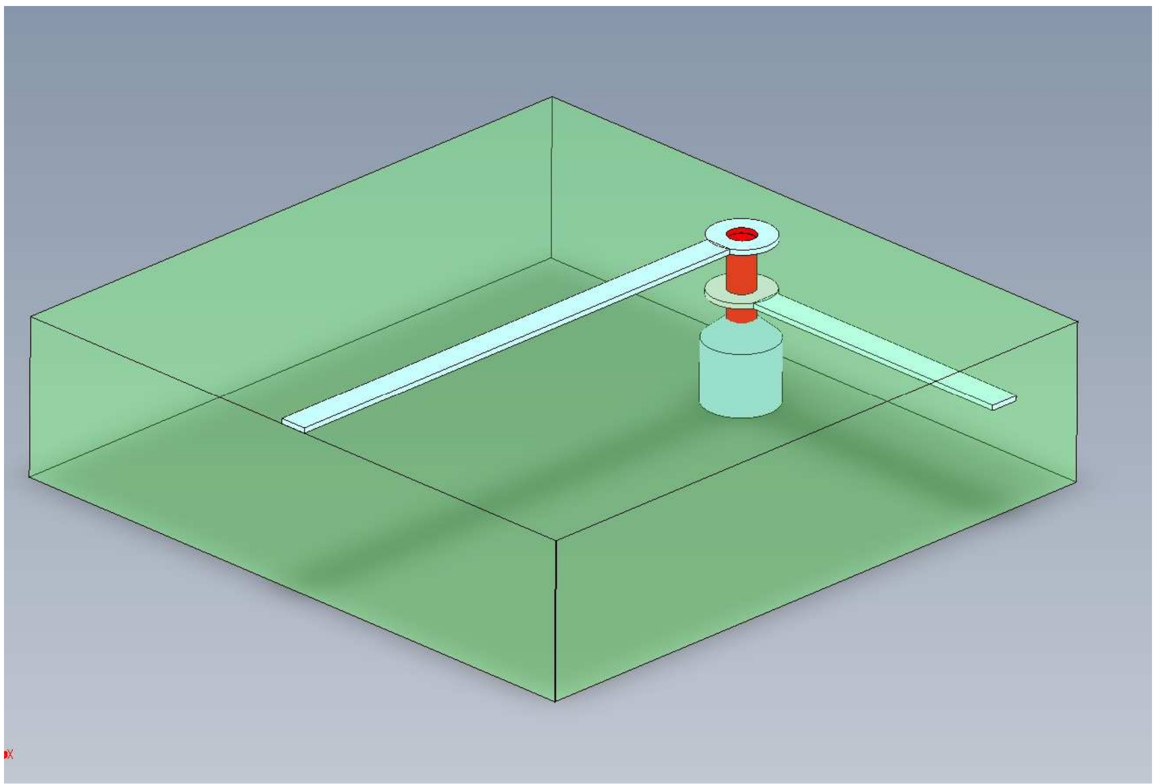


This is a particularly sneaky type of stub that's easy to overlook working in 2D.

At higher frequencies, stubs like this become increasingly relevant.

One cure -Backdrilled Vias

Backdrilled via



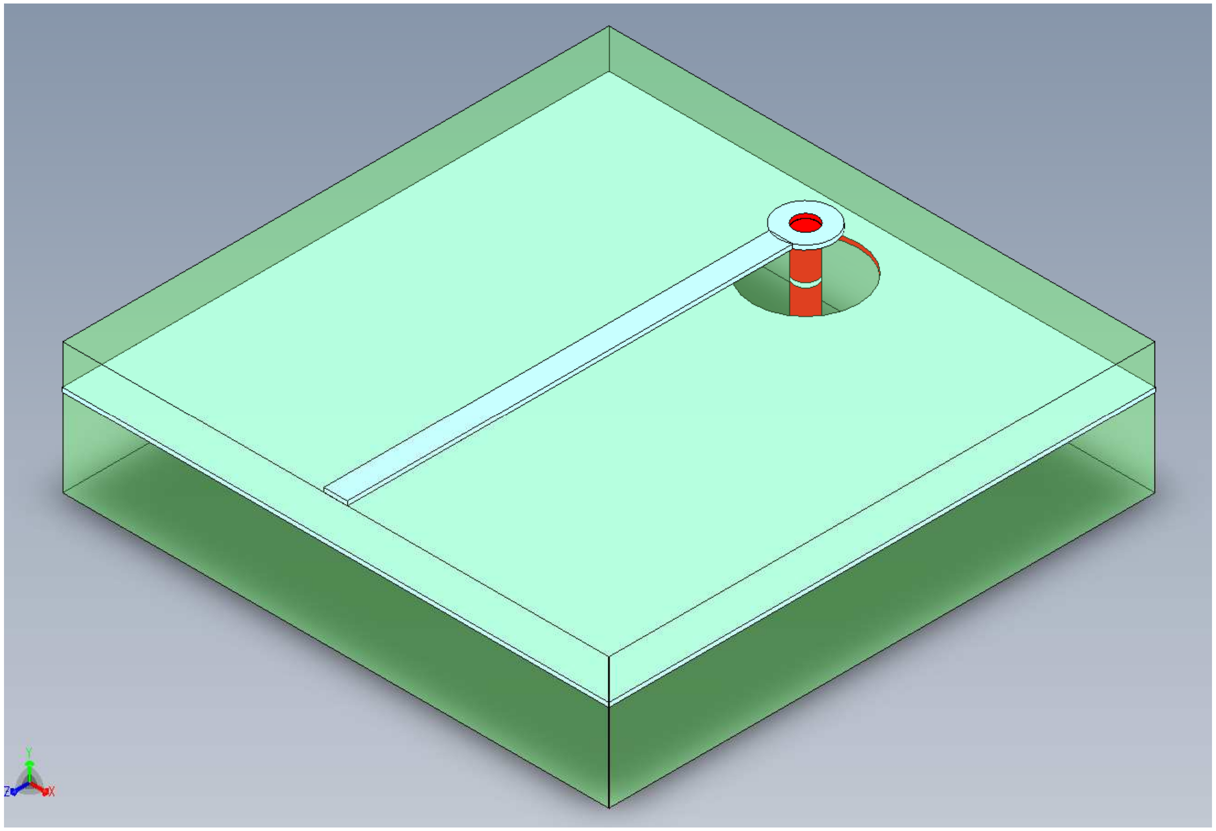
Backdrilling vias does remove the via stub, but comes at a cost in money and risk.

TRANSMISSION LINES

"With RF, if it don't look good, it don't work good."

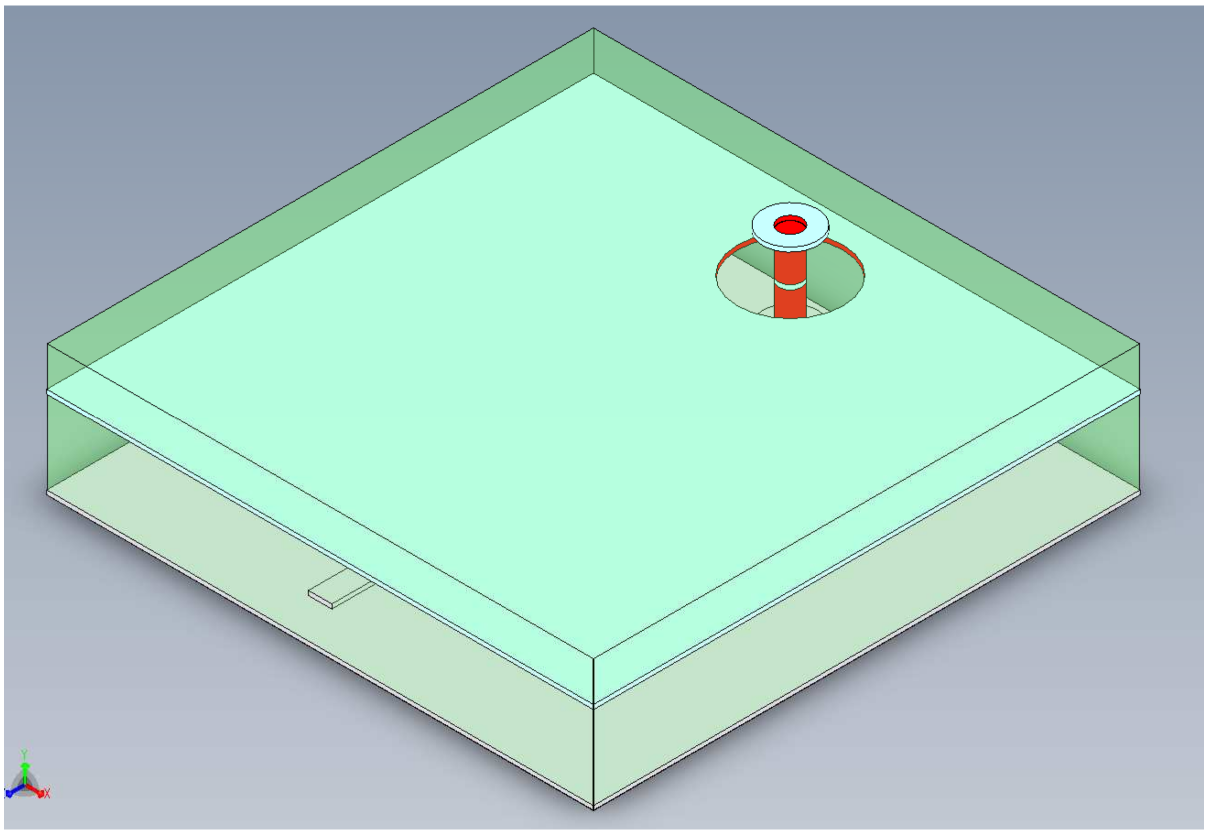
Anonymous

Microstrip



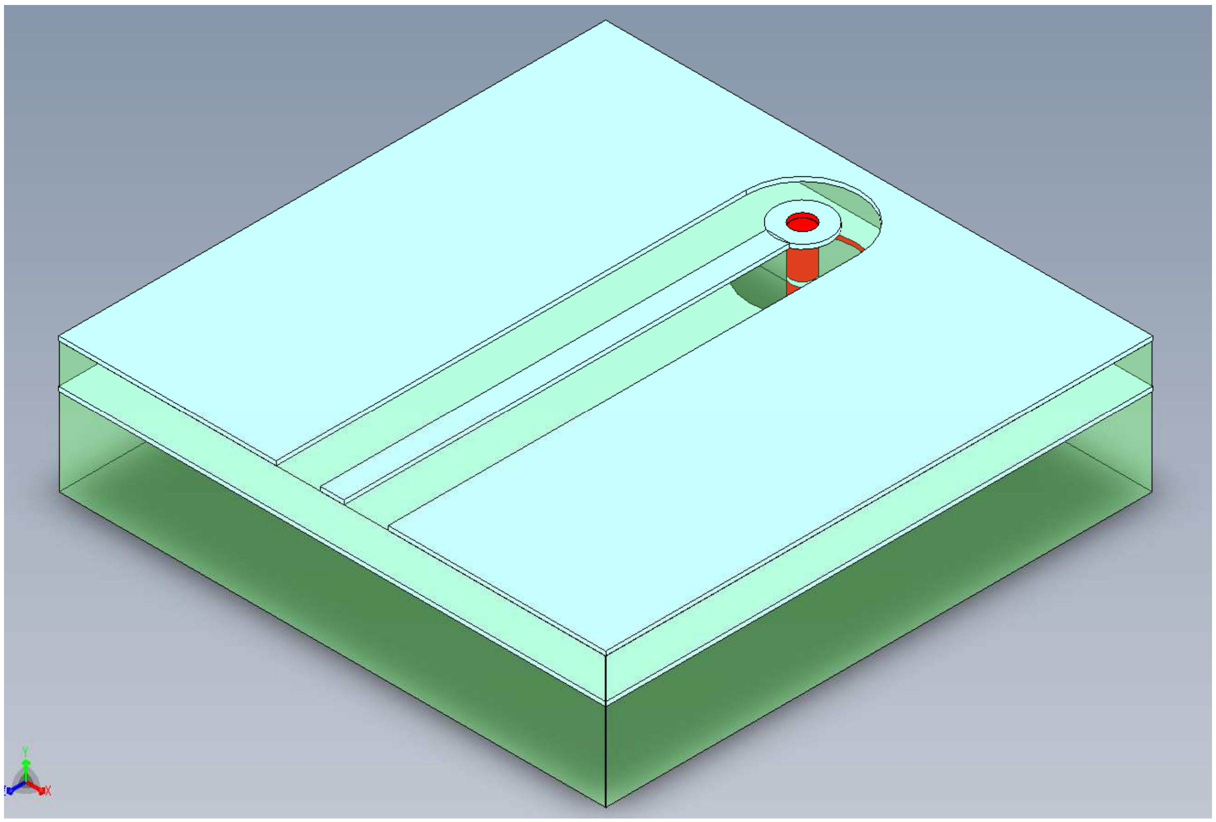
Trace referenced to a GND plane on one nearby adjacent layer.

Stripline



Trace referenced to two GND planes on two nearby adjacent layers.

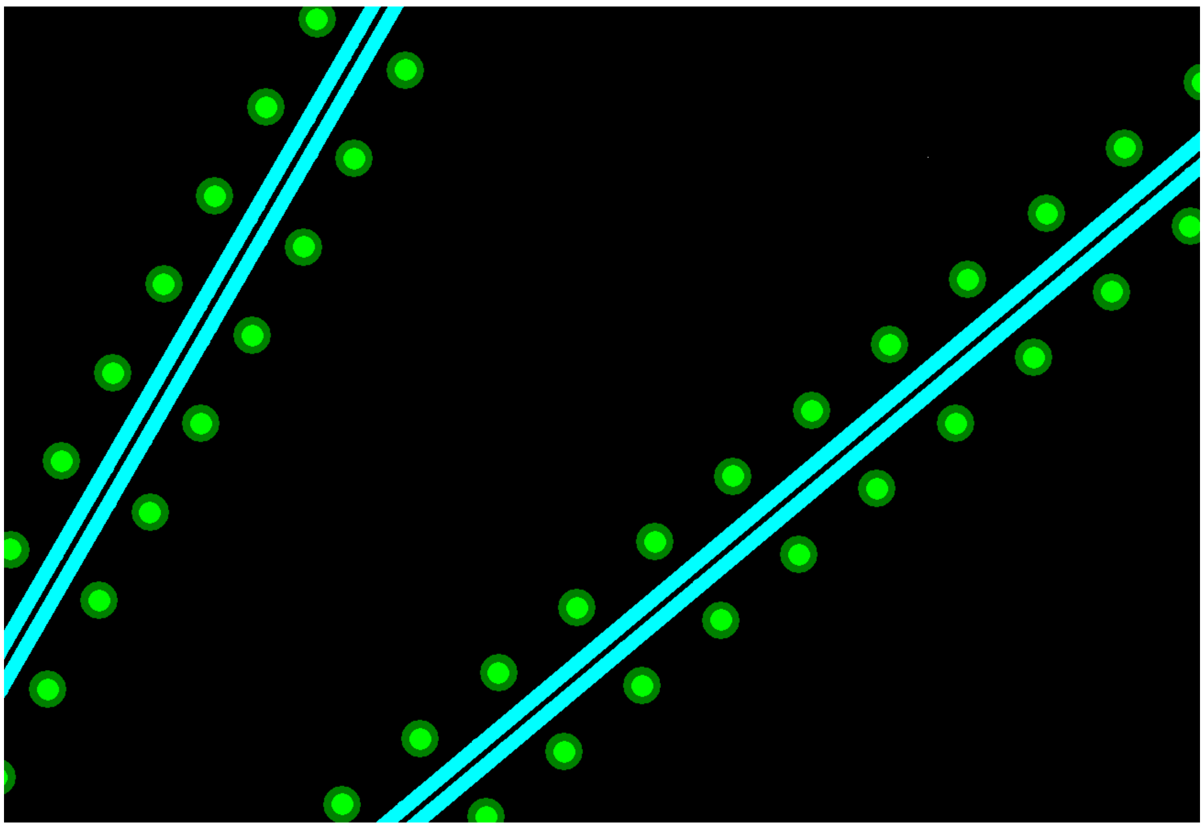
Coplanar waveguide with GND



Trace referenced to a GND pour on an adjacent plane and on the same layer.

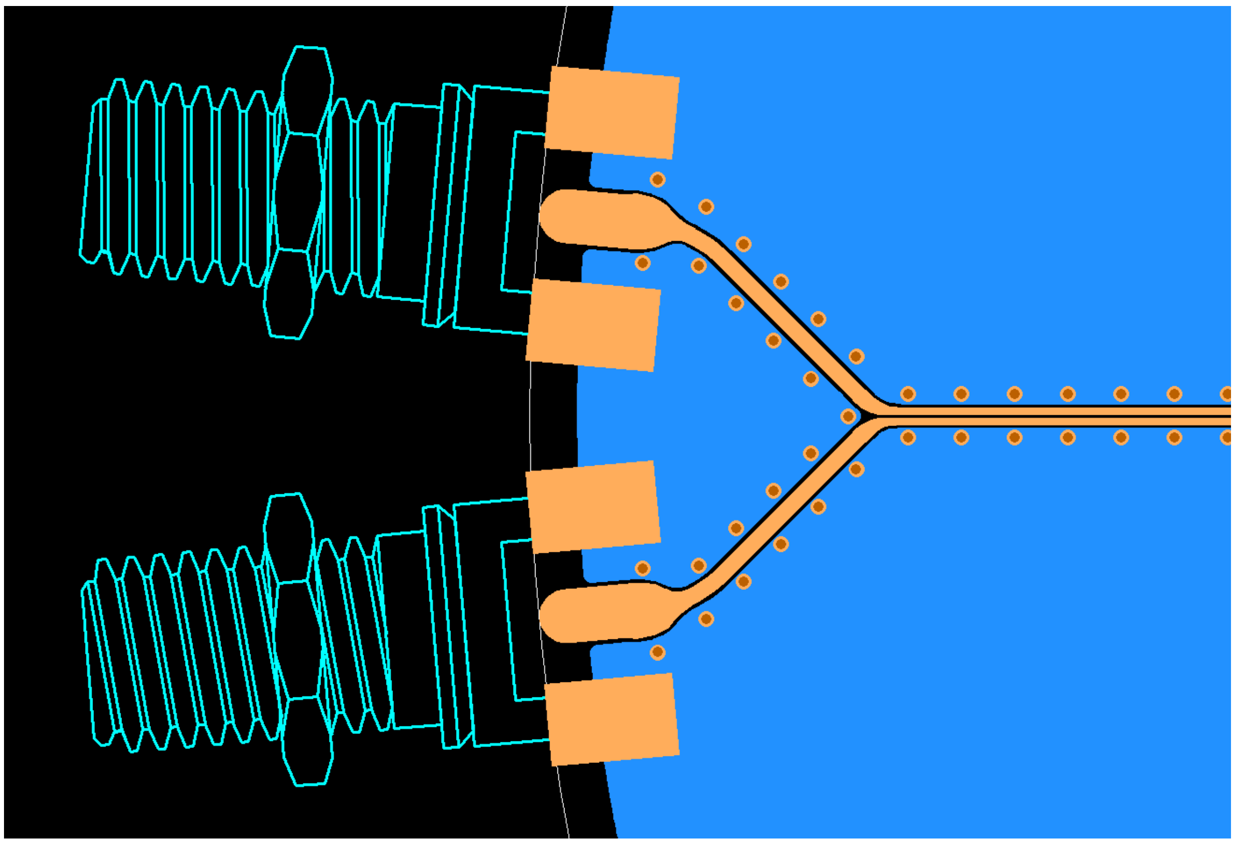
Differential coplanar waveguide with GND

100 ohm differential coplanar waveguide, GND not shown



Coplanar waveguide with GND

50 ohm single ended to 100 ohm differential waveguide

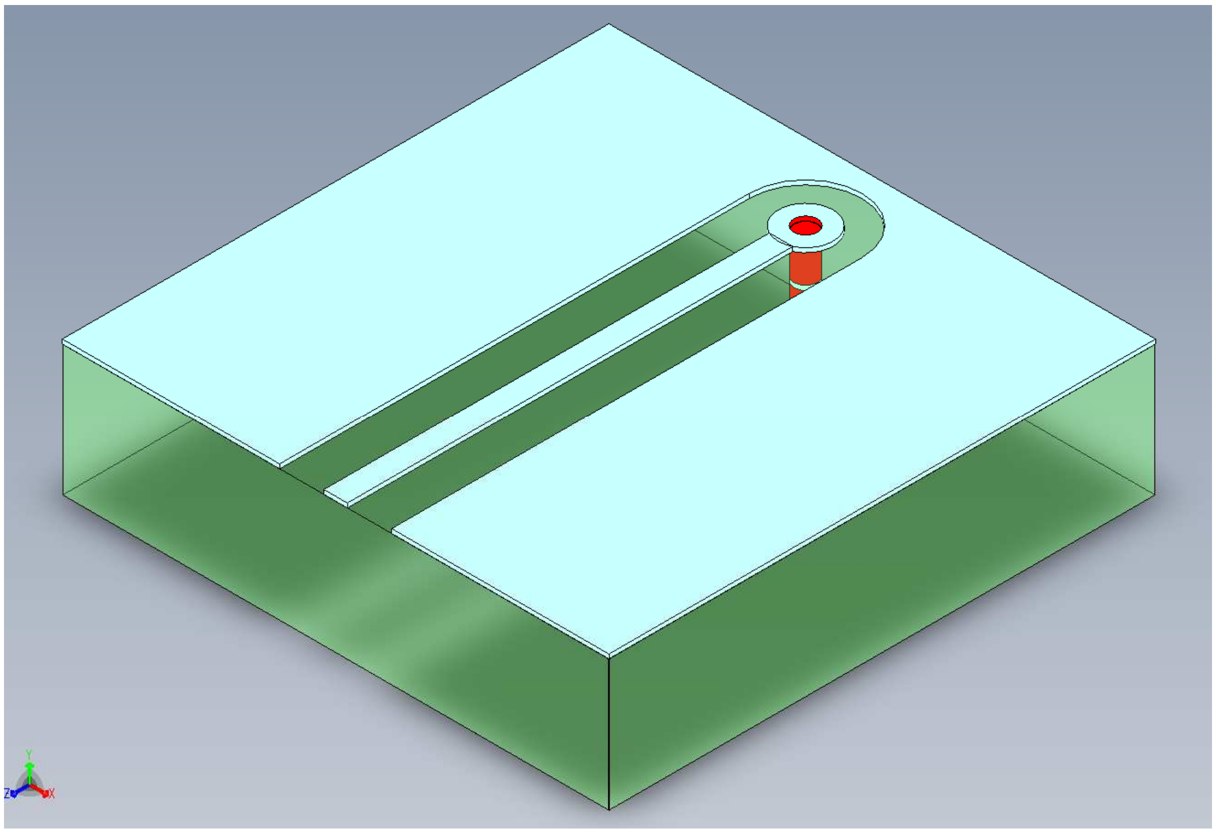


Single-ended to differential transition.

Note the intentional width changes to deliberately alter the trace impedance at the transition.

Coplanar waveguides, cont.

Coplanar waveguide referenced to a single GND plane on same layer



Trace referenced to a GND plane on same layer.

Very uncommon.

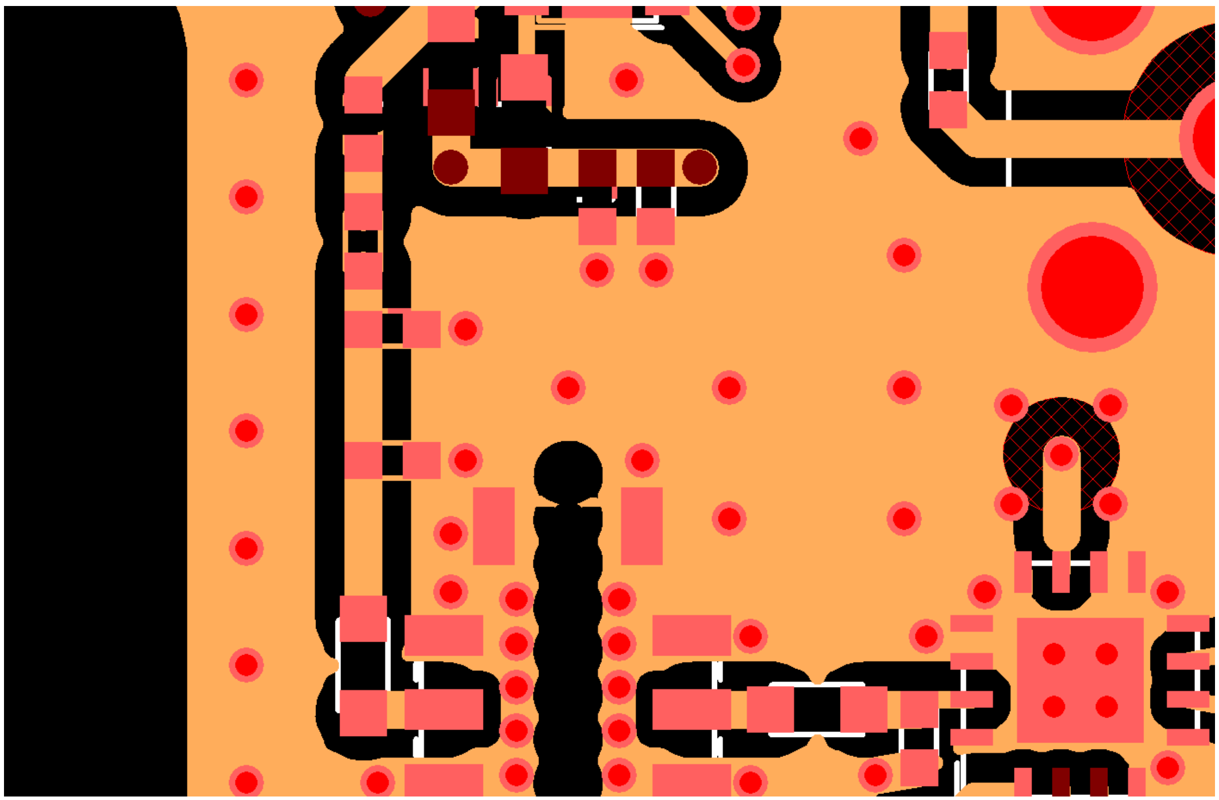
GND

"You can never have too many GND vias…"

- Anonymous

GND

Typical RF signal chain in a surface layer GND pour



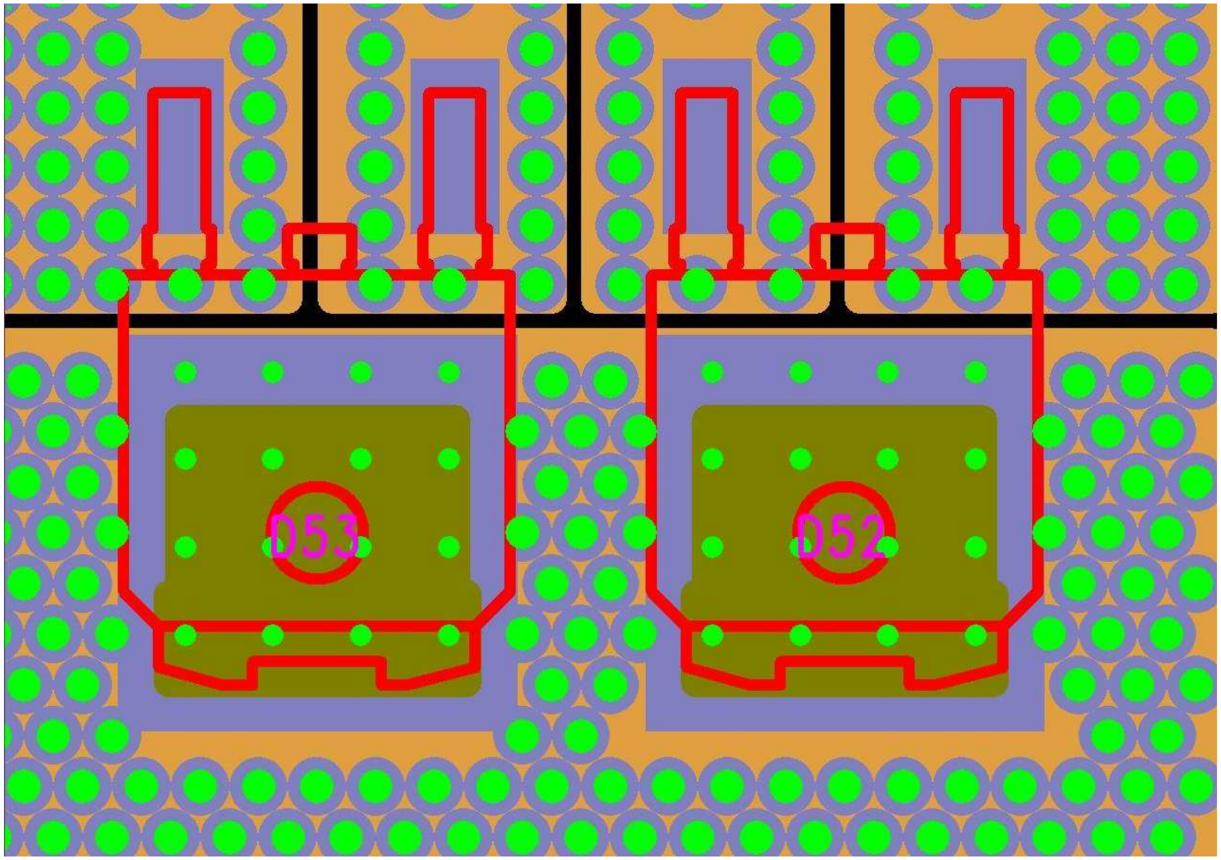
We think of GND as a special reference net, but in practice, it's just another wide trace, with all the usual trace parasitics.

RF PCB's frequently have multiple GND layers.

It's considered good practice to place a number of GND vias in the area around RF circuitry.

GND Vias

"You can never have too many GND vias."



Well, maybe this is too many GND vias…

Seriously though, RF power amplifier PCB's sometimes have arrangements of GND vias very much like that shown here.

RF ISOLATION

"Isolation is always important, and always problematic"

- Anonymous

RF shields

Simple stamped shield cover

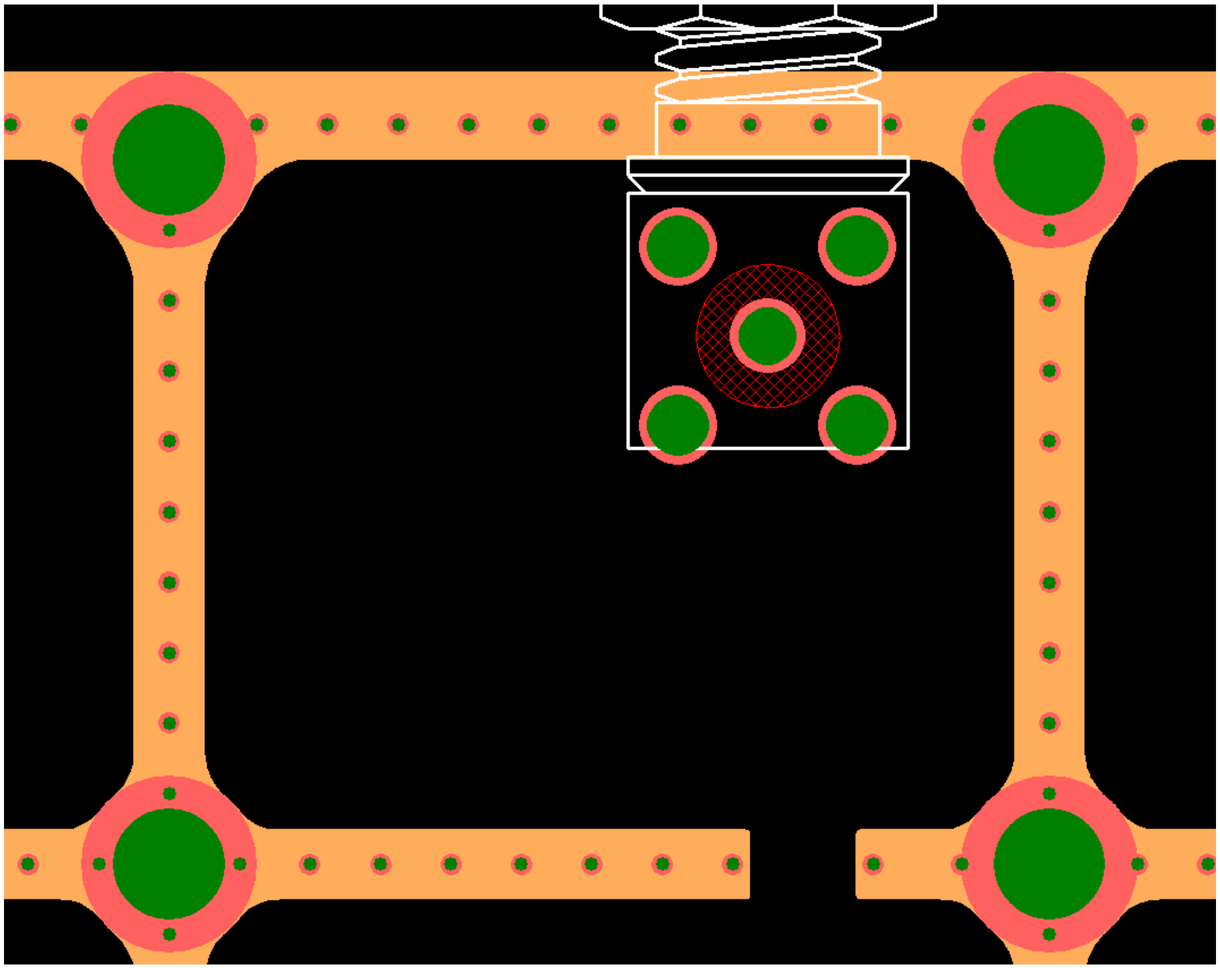
PCB mounted shields come in several types.

Intimate connection between the shield and the PCB is important to minimize leakage.

Close proximity between shields and nearby parts impacts impedance

RF shields, cont.

Typical layout of the footprint of a milled shield

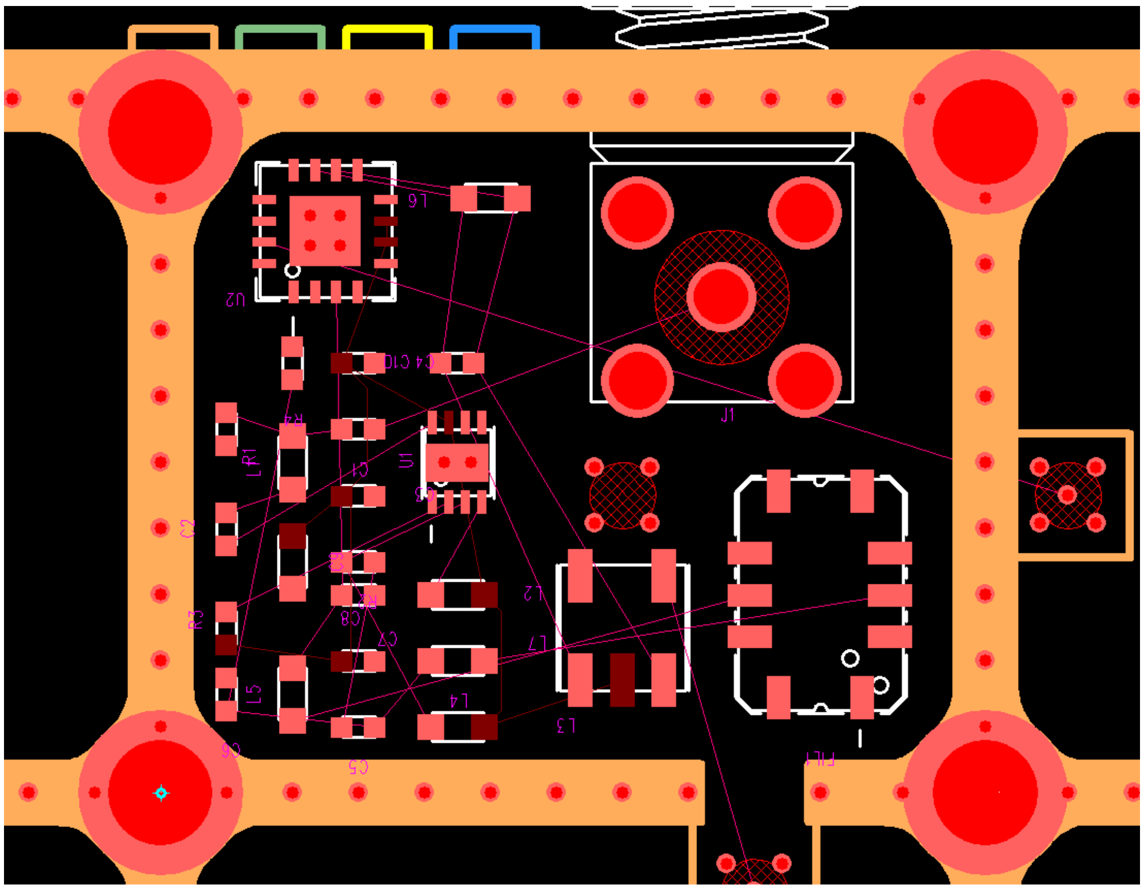


RF PCB's are frequently contained in milled aluminum enclosures with integrated milled shields.

Watch that soldermask!

Isolation between circuits within a compartment

Typical import of parts in their shielded compartment.



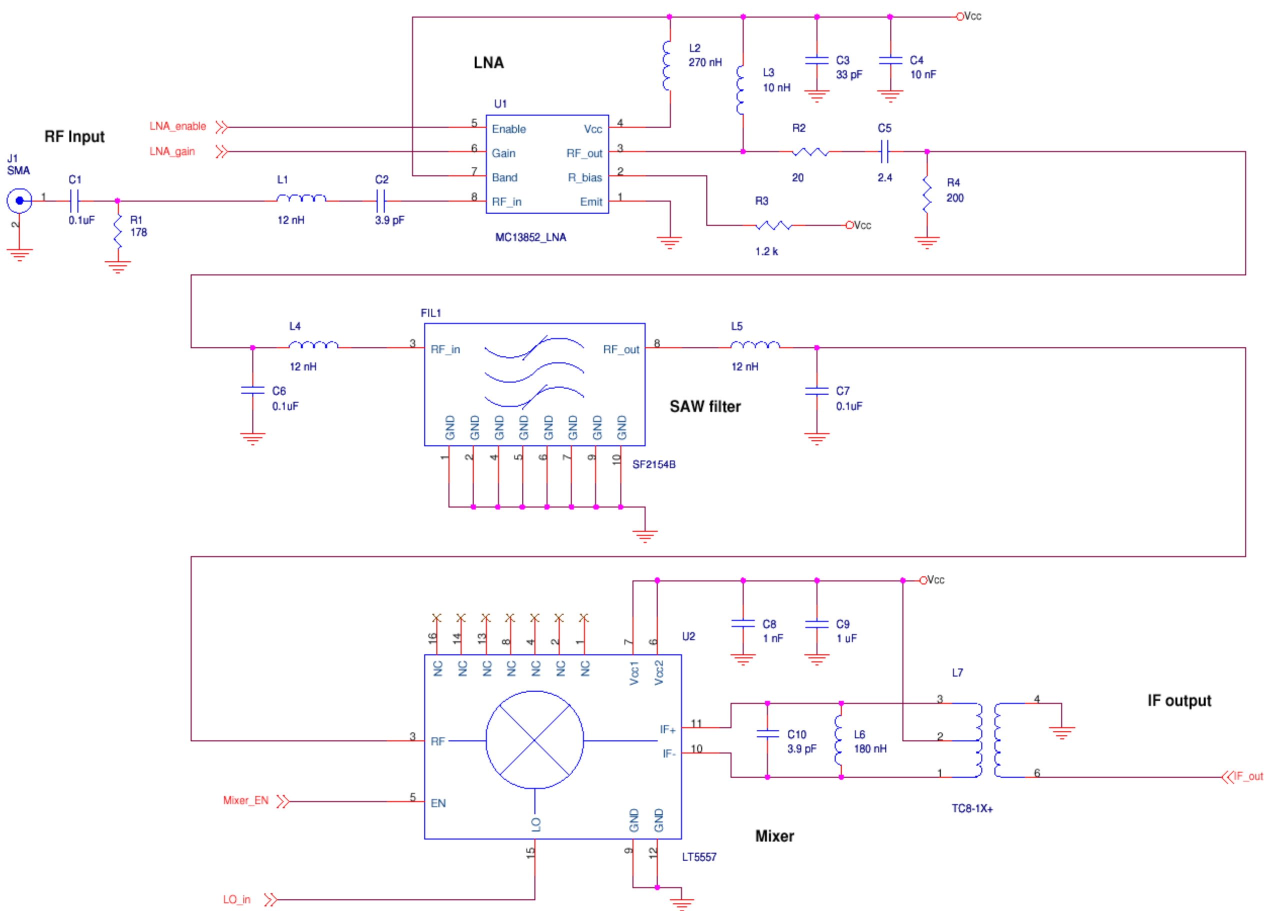
RF Circuits are frequently crammed into small compartments making it a juggling act to maintain distance between adjacent circuits.

Coupling of inputs and outputs of LNA's, coupling between circuit blocks, and the circuit as a whole is profoundly impacted either positively or negatively, by the layout.

DOWNCONVERTER DEMO PCB

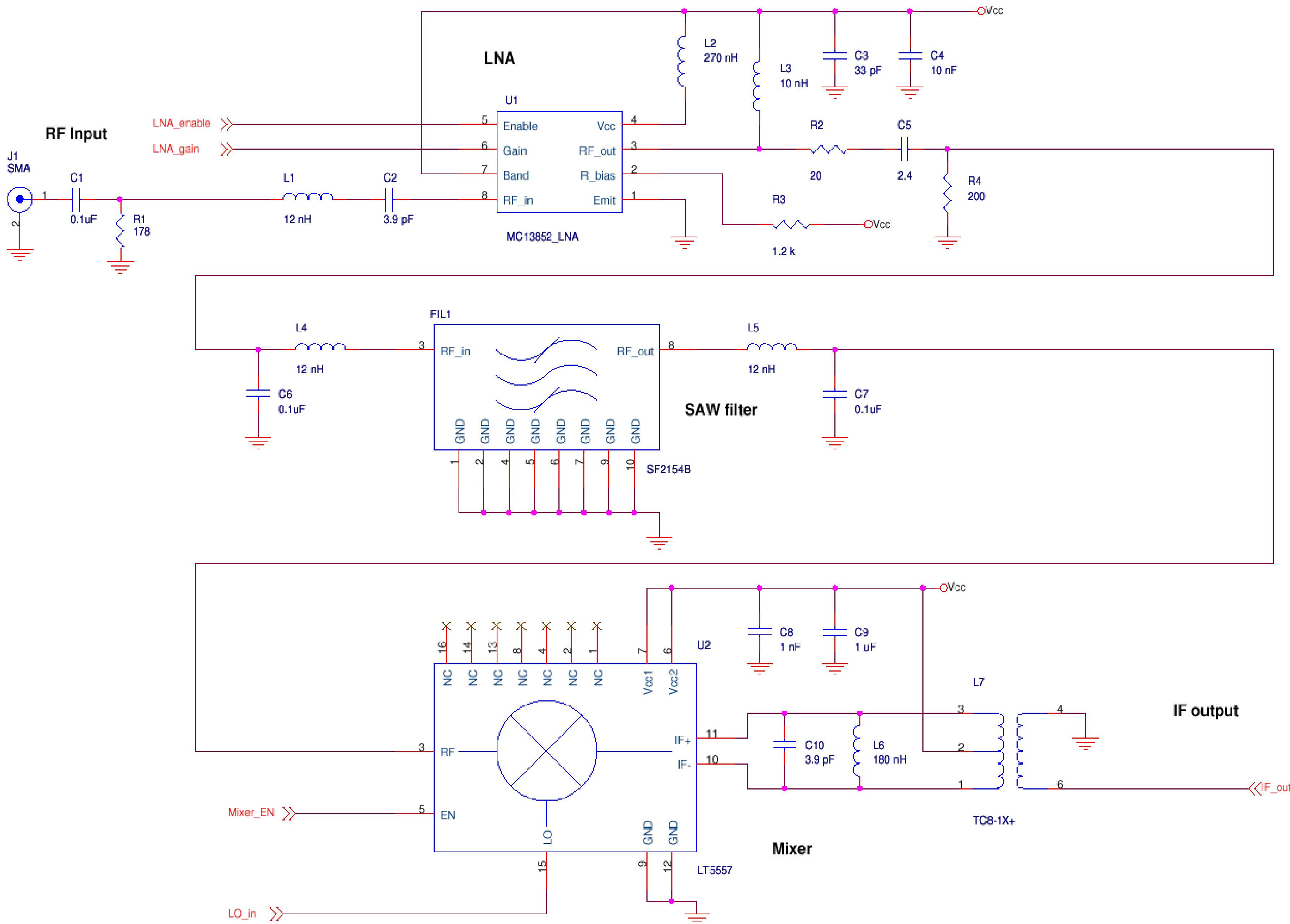
A step by step walk through the signal chain of a relatively small PCB containing a relatively large number of errors.

Demo Downconverter Schematic



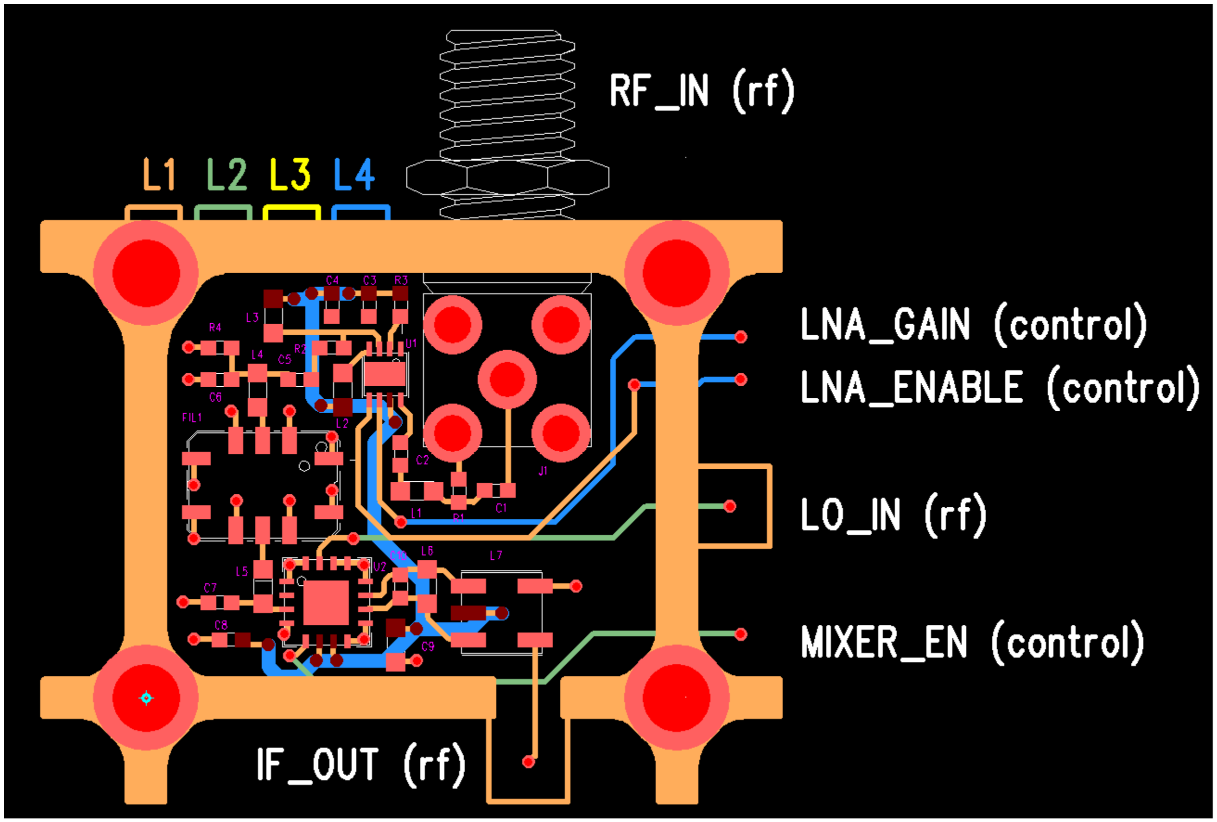
This demo RF circuit comprises an input connector, matching network, an LNA, SAW filter, and a mixer.

This is not a real design, just a routing demonstration tool.


LNA
RF Input
J1
SMA
C1
0.1uF
R1
178
L1
12 nH
C2
3.9 pF
LNA_enable
LNA_gain
U1
Enable
Gain
Band
RF_in
Vcc
RF_out
R_bias
Emit
MC13852_LNA
L2
270 nH
L3
10 nH
C3
33 pF
C4
10 nF
Vcc
R2
20
C5
2.4
R3
1.2 k
R4
200
FIL1
L4
12 nH
C6
0.1uF
RF_in
RF_out
GND
SF2154B
L5
12 nH
C7
0.1uF
SAW filter
U2
NC
Vcc1
Vcc2
C8
1 nF
C9
1 uF
L7
IF output
RF
IF+
IF-
EN
LO
GND
C10
3.9 pF
L6
180 nH
IF_out
TC8-1X+
Mixer_EN
Mixer
LT5557
LO_in

Demo Downconverter PCB Layout

Demo Downconverter PCB layout, with botched RF routing



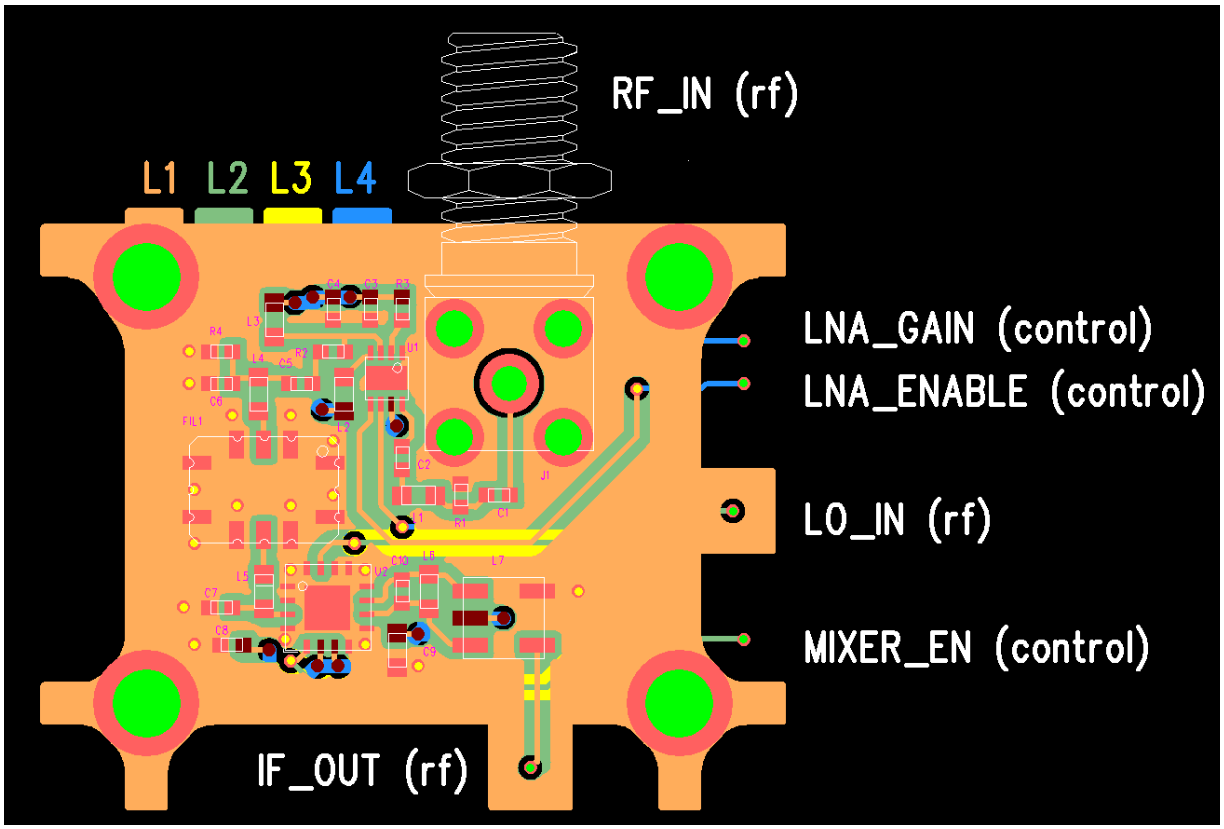
This demo PCB attempts to abide by a typical collection of RF layout mechanical constraints.

The circuit is meant to fit within the shielded area shown.

A handful of control lines and RF signals enter and exit the compartment.

Demo Downconverter PCB Layout, with errors

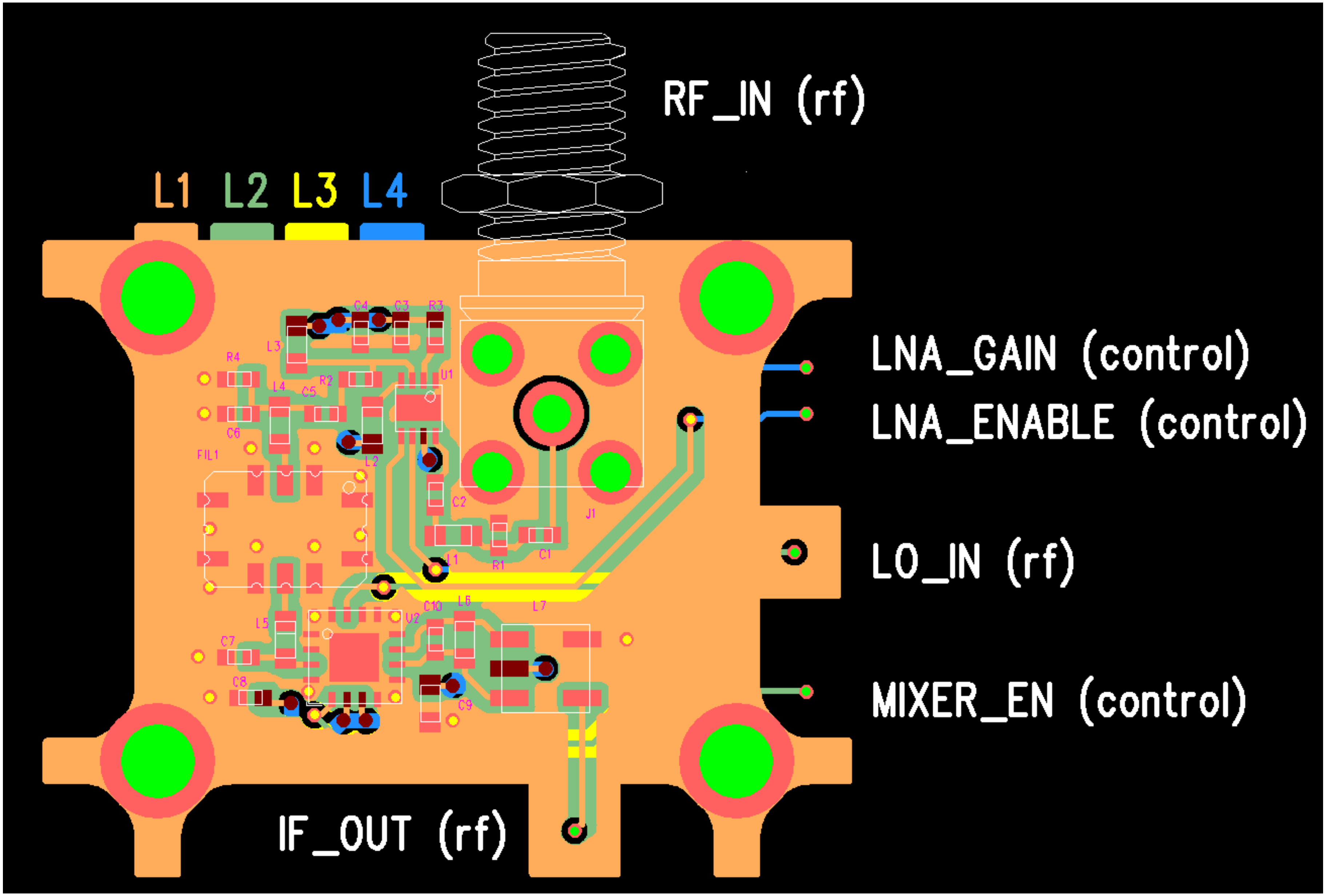
Demo Downconverter PCB layout, pours "on"



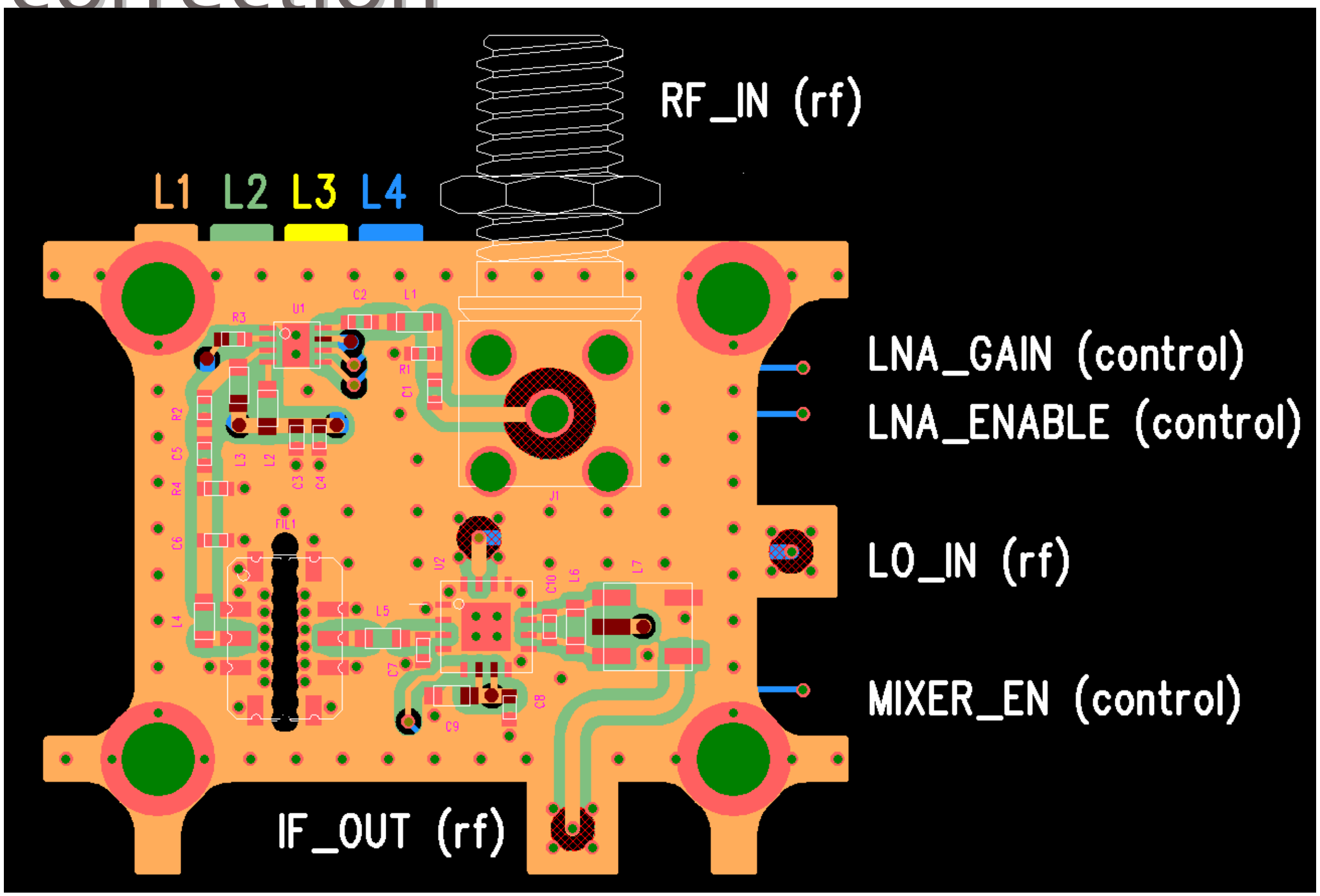
The demo PCB has 4 layers, with GND pours on L1,L2,L3, and L4. They are rendered "on" in this image

The IF_OUT line is considered the output, and is required to exit the shield through the mousehole opening.

The layout as shown contains, by design, a large number of common RF PCB routing errors.

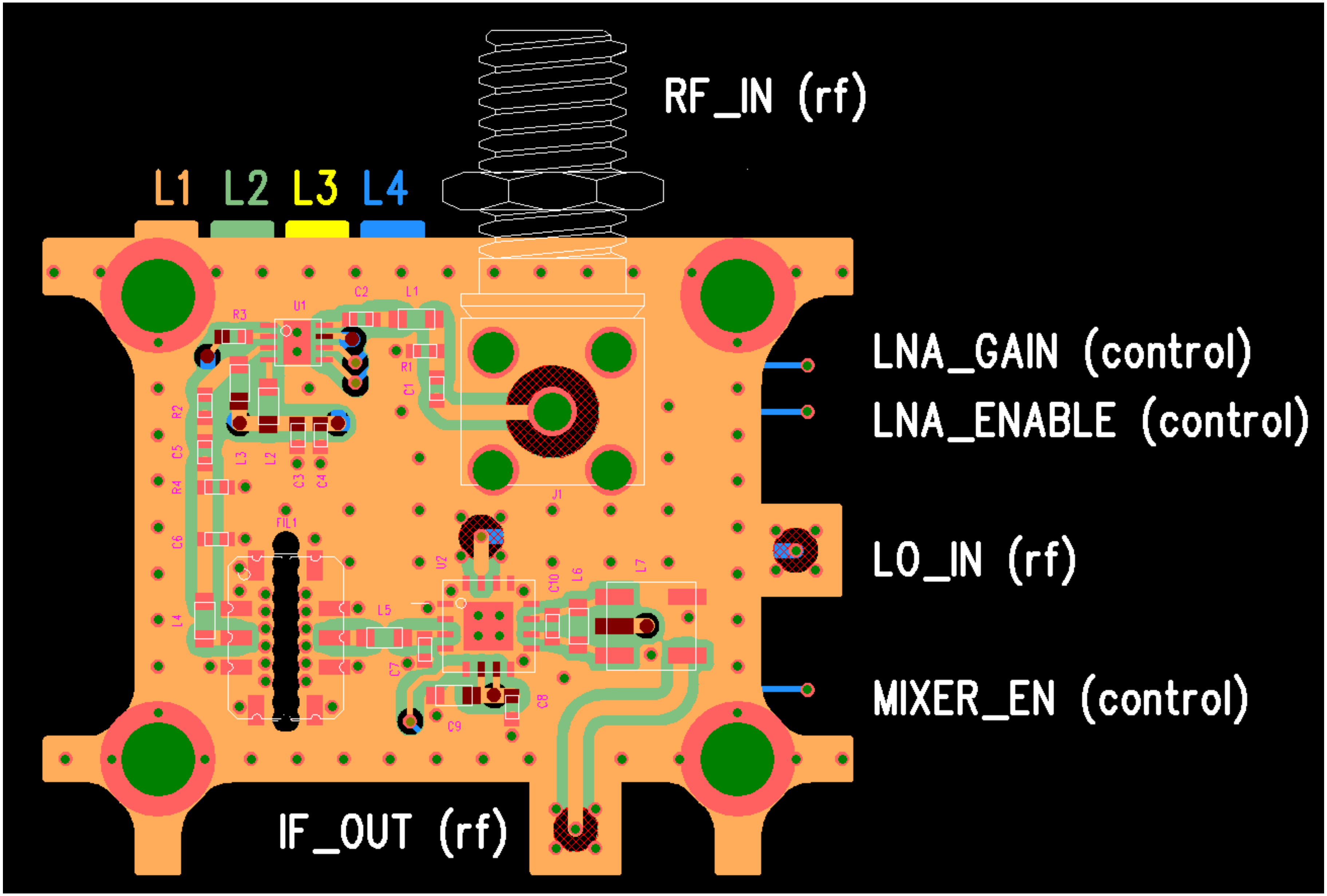

RF_IN (rf)
L1 L2 L3 L4
LNA_GAIN (control)
LNA_ENABLE (control)
LO_IN (rf)
MIXER_EN (control)
IF_OUT (rf)

Demo Downconverter PCB Layout, after error correction

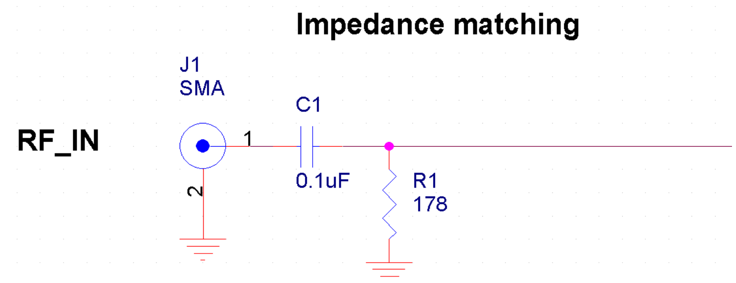


Here's a sneak preview of the corrected layout.

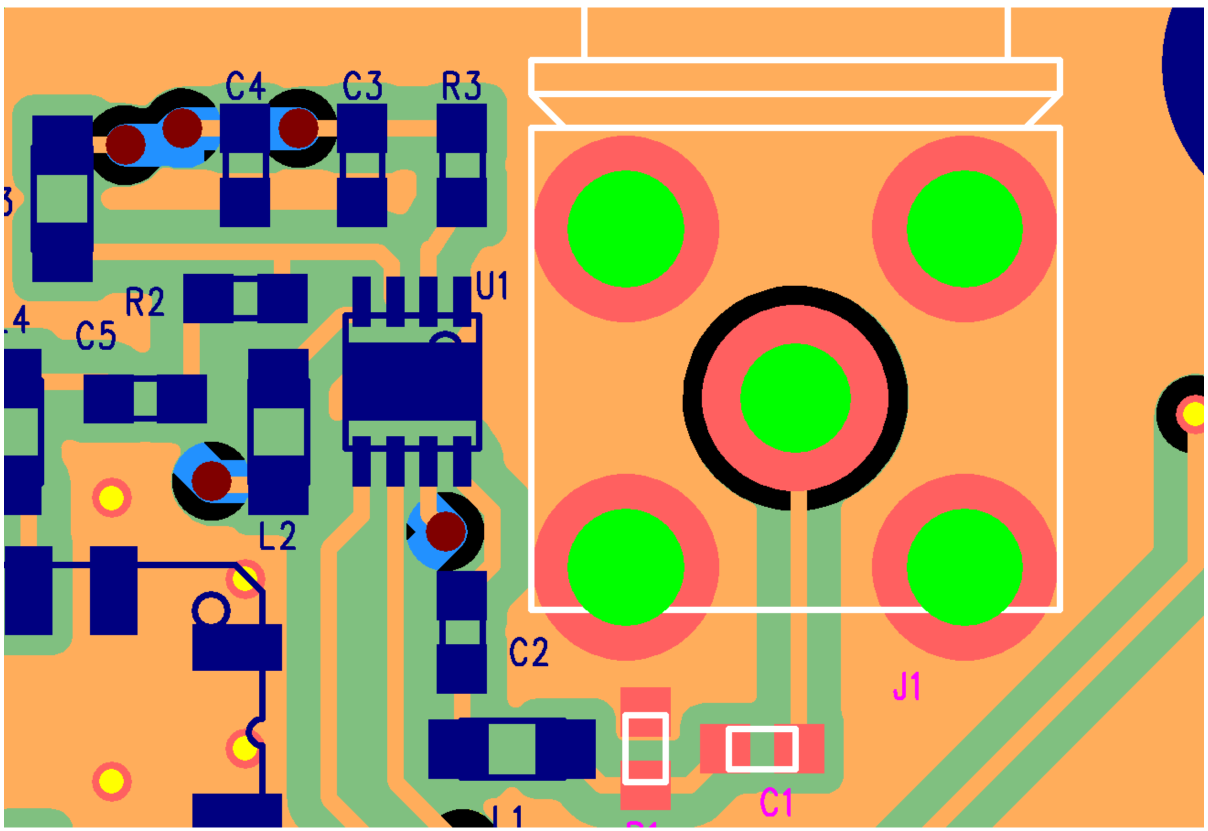
Next is a more detailed look at each section.


RF_IN (rf)
L1 L2 L3 L4
LNA_GAIN (control)
LNA_ENABLE (control)
LO_IN (rf)
MIXER_EN (control)
IF_OUT (rf)

Input connector and matching network

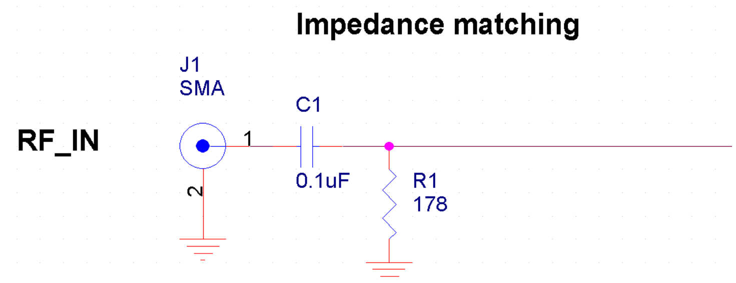


Input and impedance matching network with routing errors

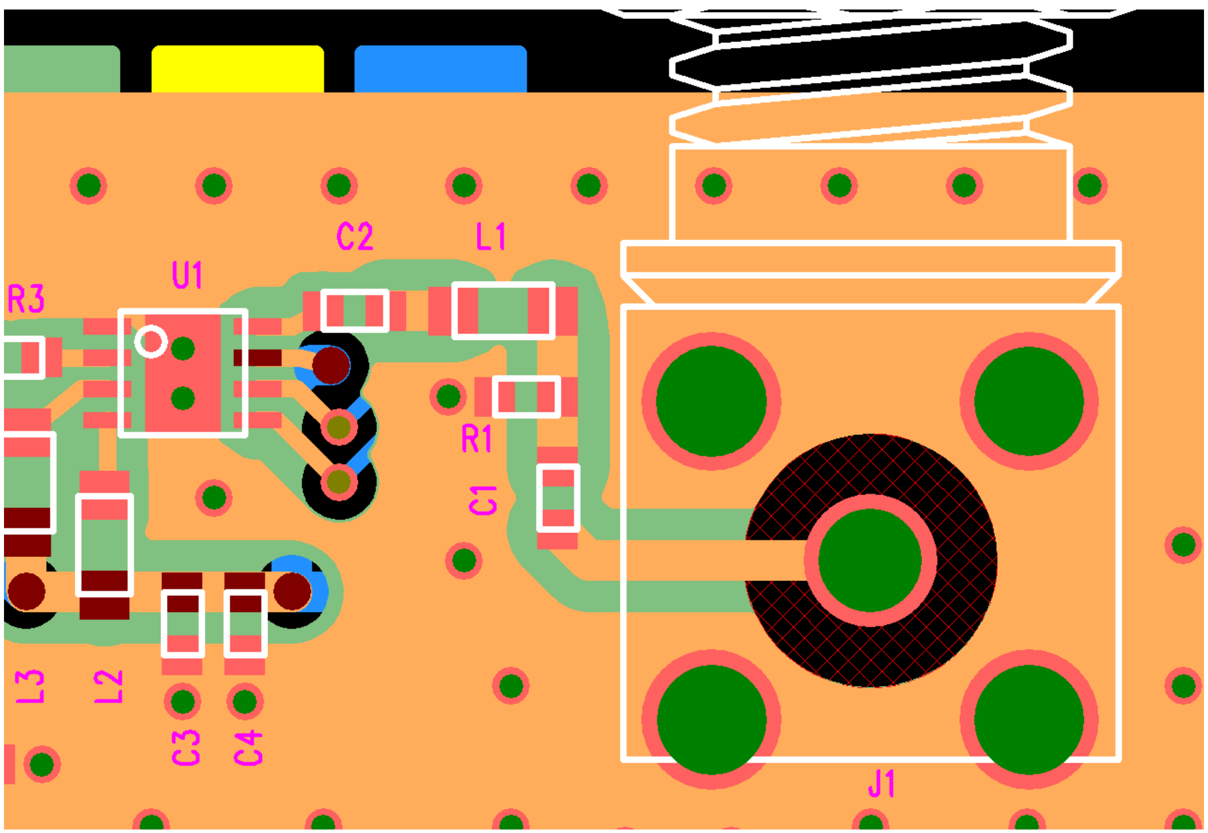


There are at least 4 layout errors at the input connector J1 alone! Maybe you spot even more.

Input connector and matching network



Corrected RF_IN input routing

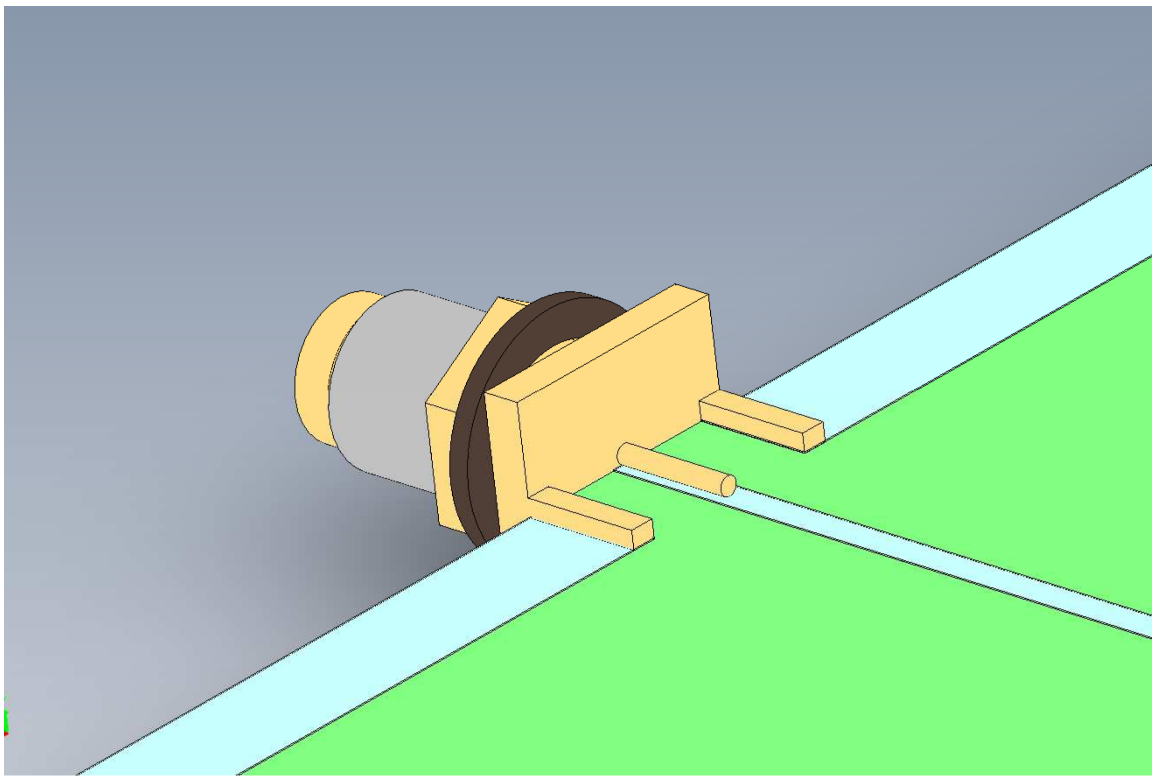


- Trace width is more appropriate.
- Pad size is smaller
- Ground is cleared from center conductor.

An edge mounted connector would be better – no stub.

Edge mounted SMA connectors

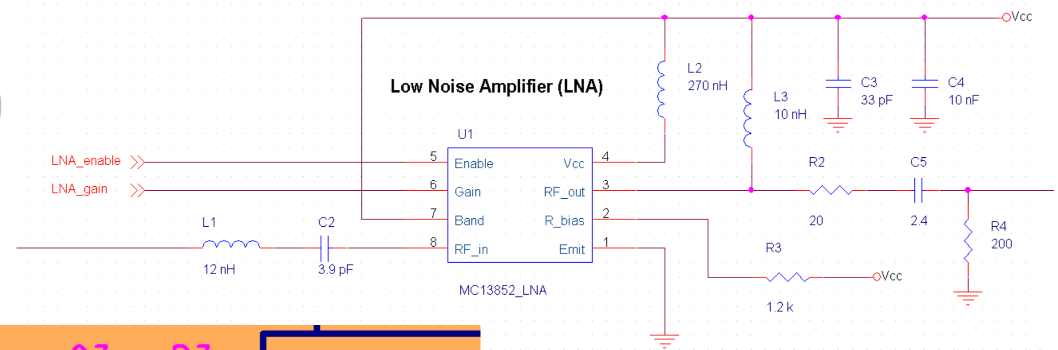
Edgemounted SMA connector

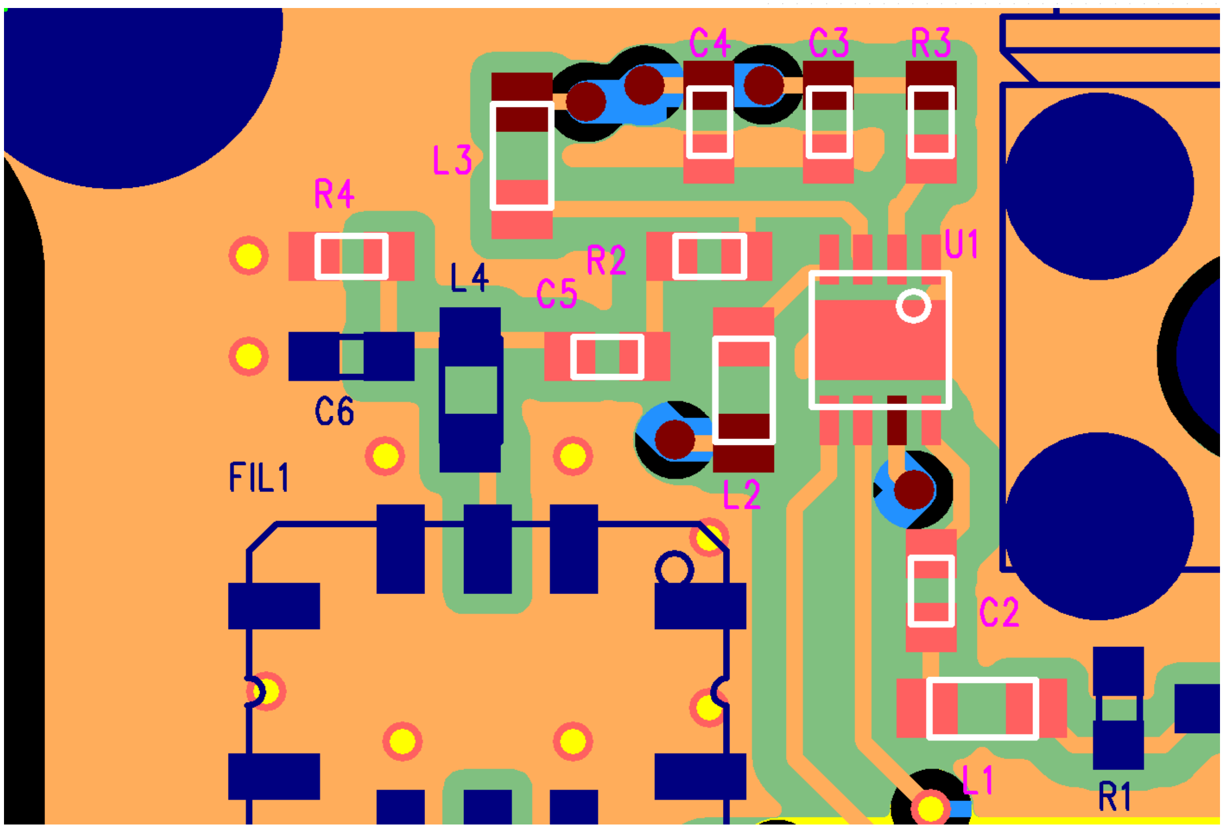


Edge mounted, or right angle surface mounted connectors nicely mitigate stubs caused by through-hole connectors.

Low Noise Amplifier (LNA)

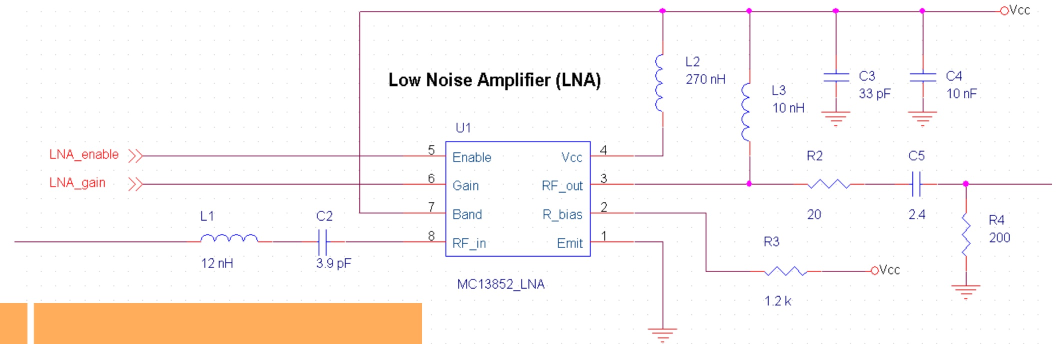
LNA circuit Schematic and Layout



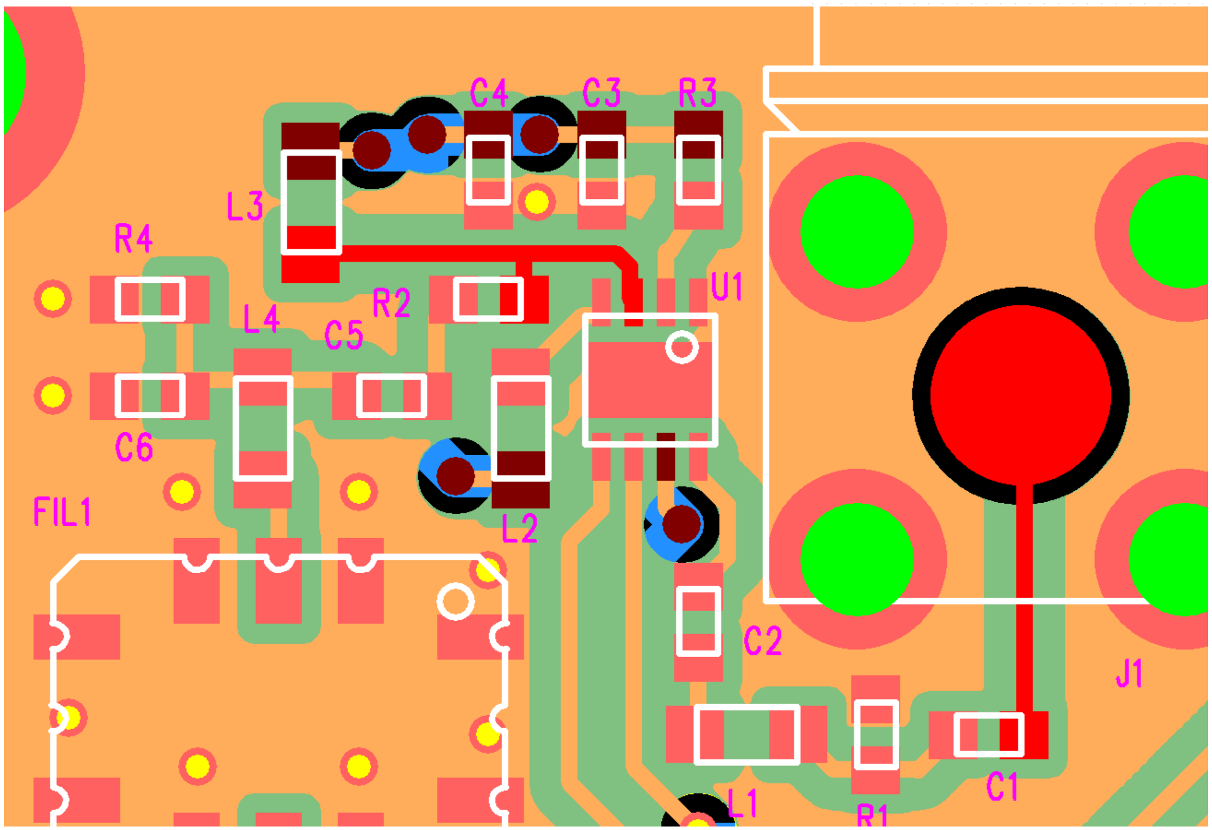


Can you spot the placement and routing errors? There are many.

LNA I/O Isolation



RF_IN and output of LNA are highlighted

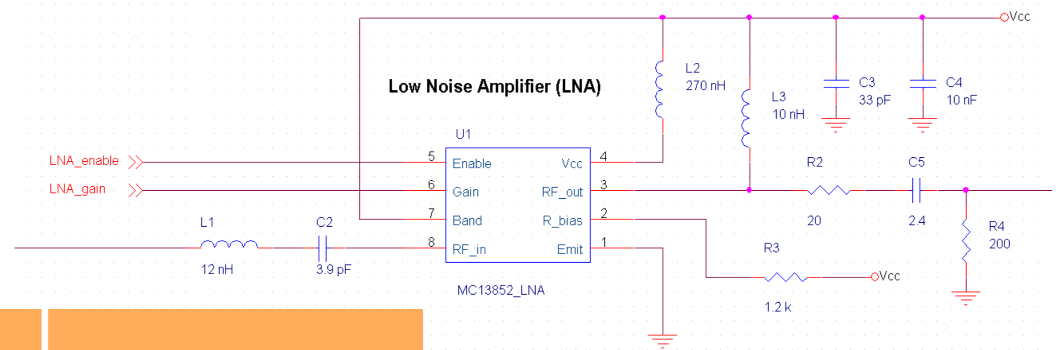


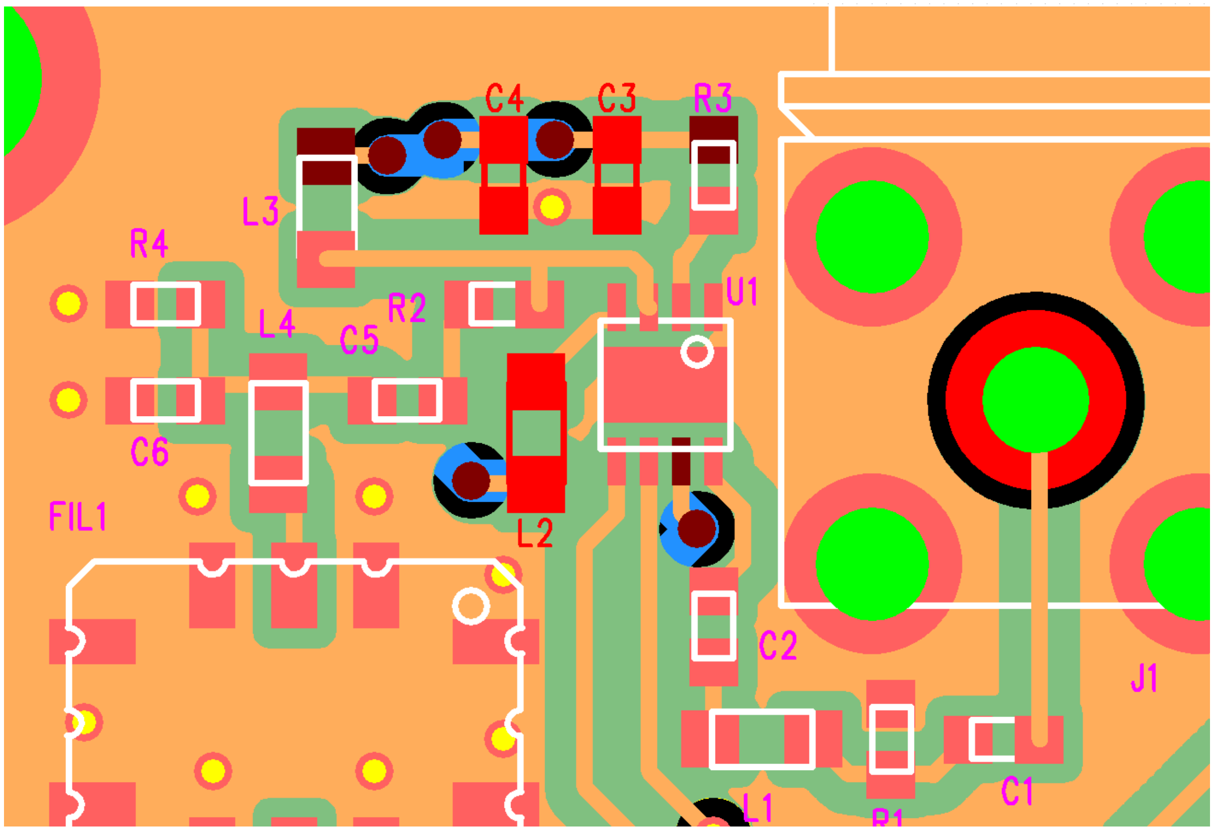
Gain blocks with insufficient isolation between input and output, tend to oscillate

The question isn't "Are they far enough apart?", it's "How can they be moved even further apart?"

LNA bypassing

L2, C3 and C4 are highlighted



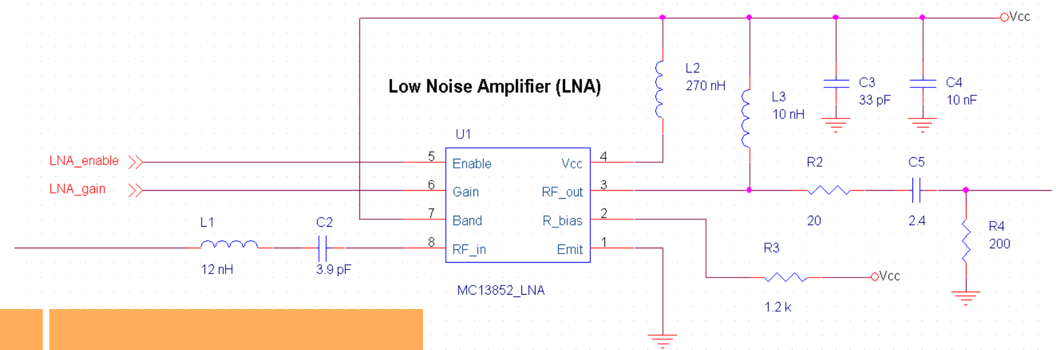


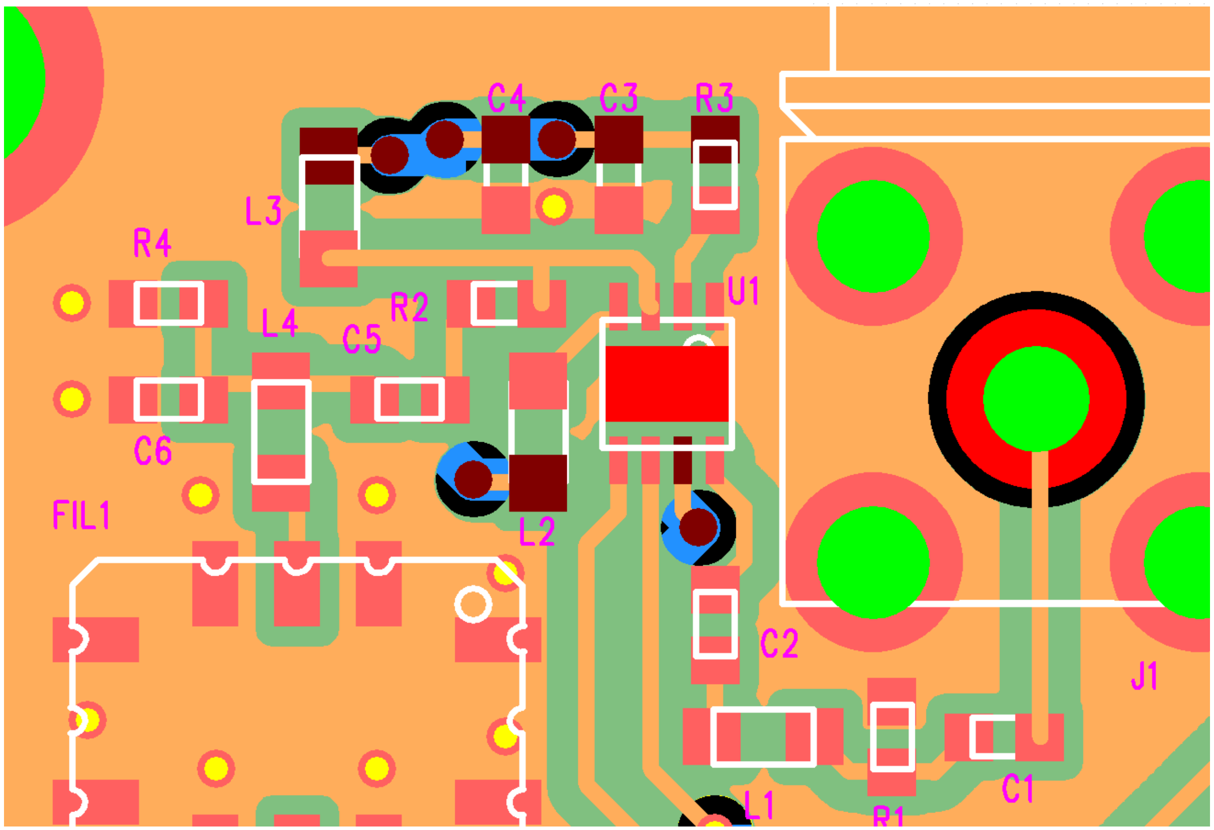
Closer to supply pin is better, for bypass capacitors.

Smallest value bypass capacitor (33pF, here) always goes closest to the supply pin.

LNA grounding

GND pad of the LNA U1 is highlighted



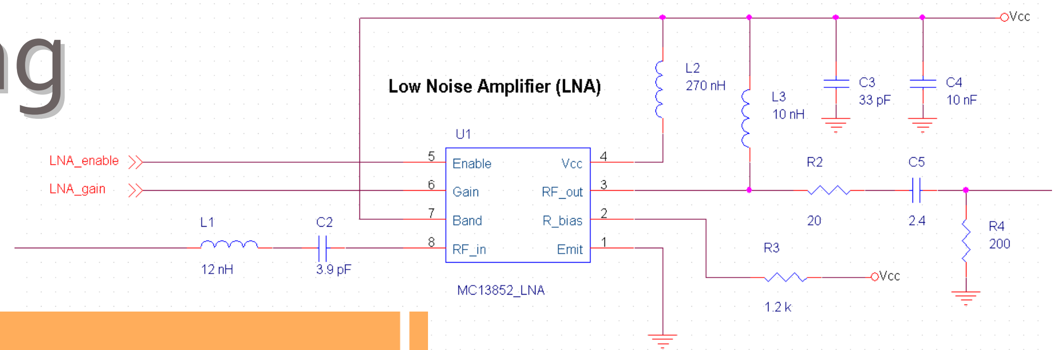


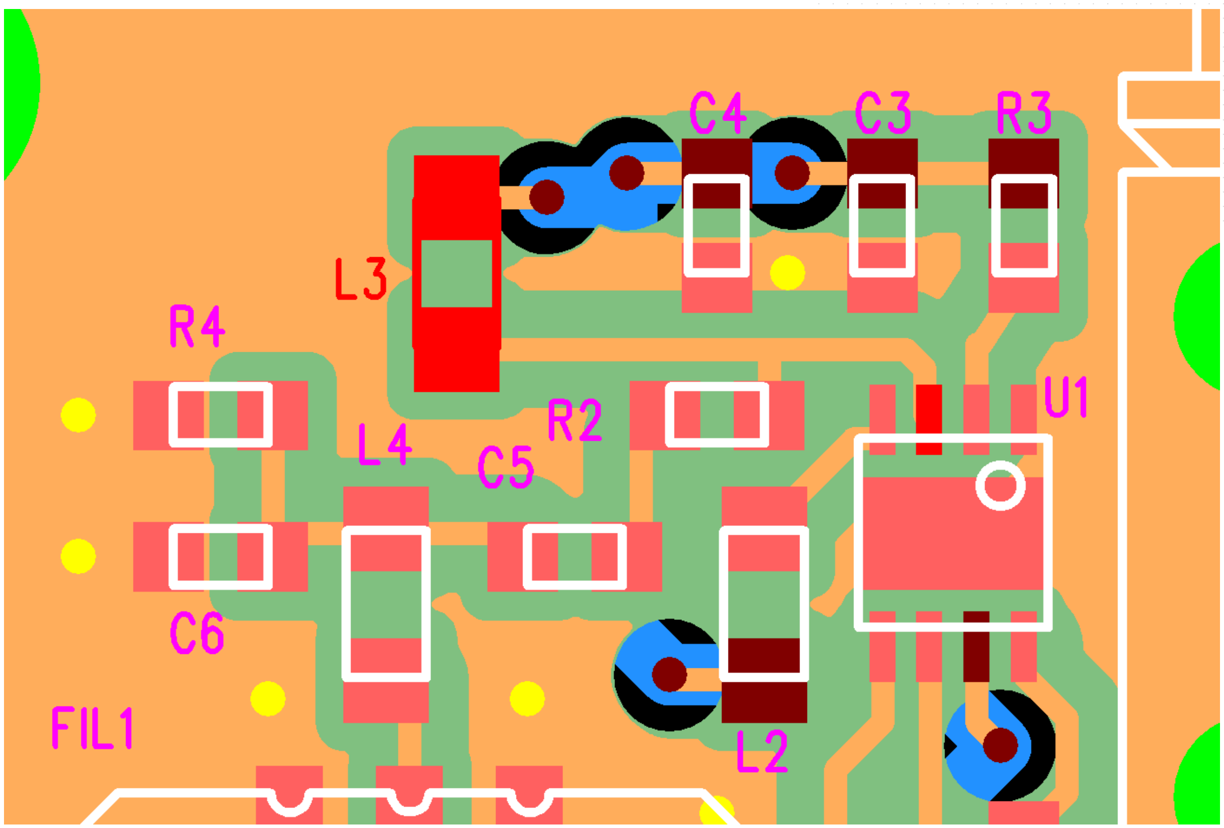
Zero probably isn't enough GND vias. The datasheet showed 2.

RF ICs with GND slugs require local vias directly to the plane(s).

LNA output biasing

LNA output bias inductor is highlighted.

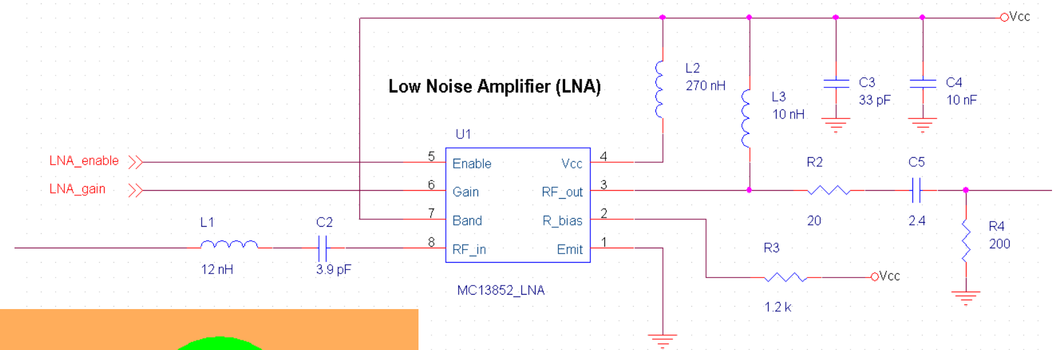




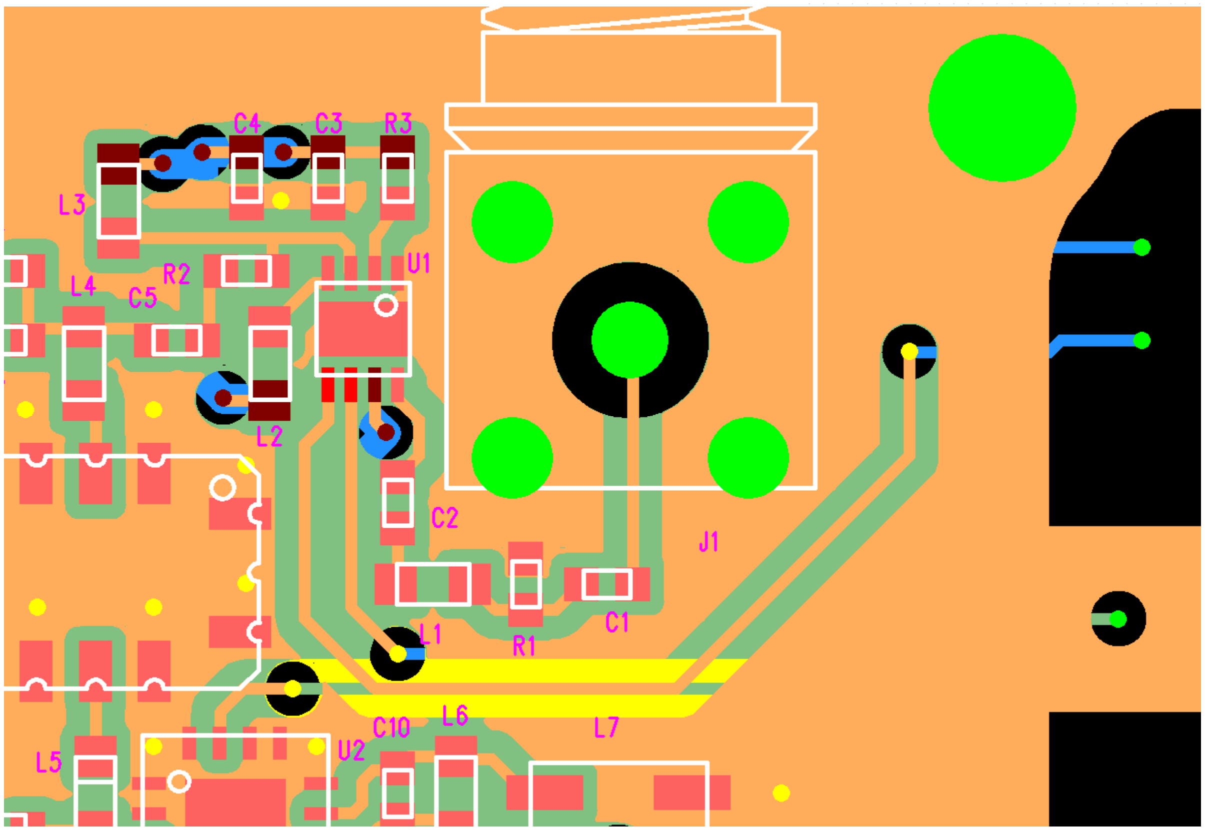
RF amplifier outputs require external biasing.

This bias component is very far from its output pin U1, pin 3.

LNA control lines

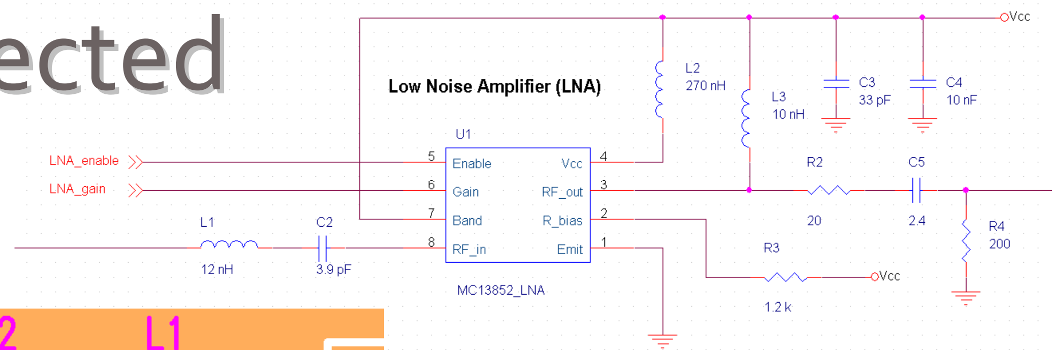


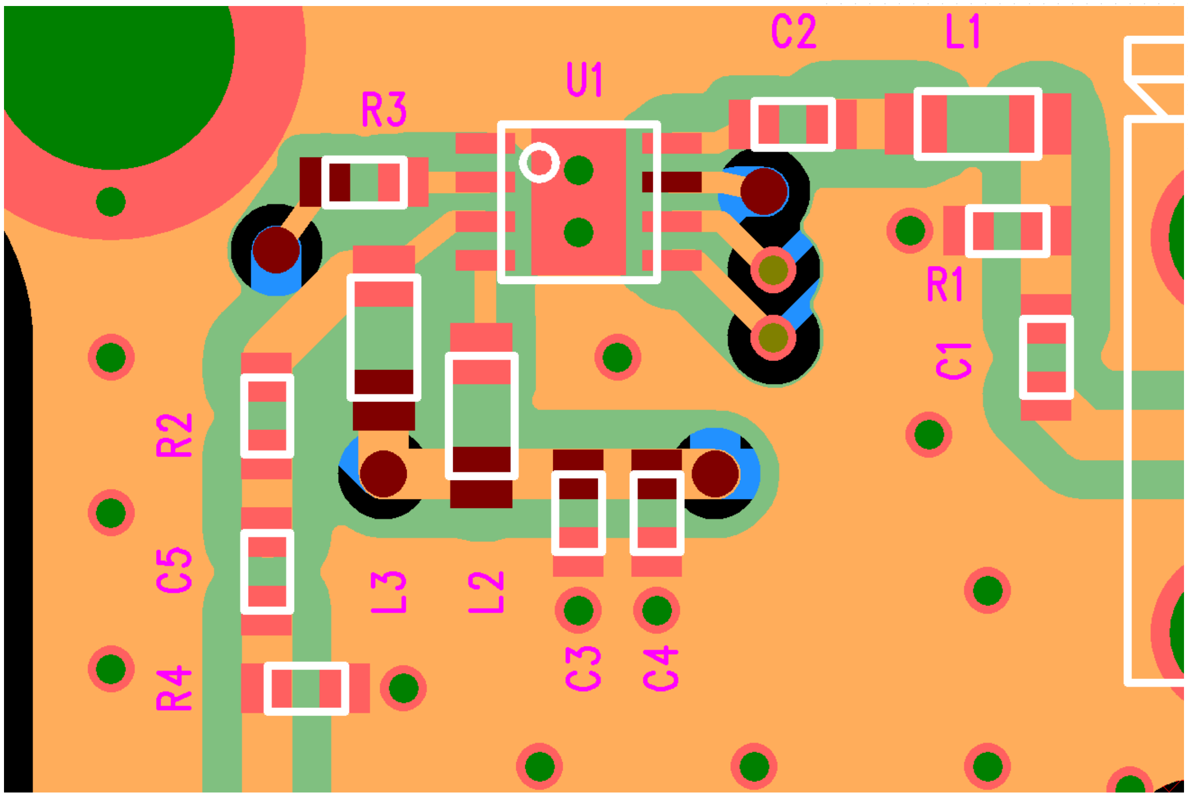
Pins 5 and 6 of the LNA u1 are highlighted



The overlap of the control line with LO_IN is particularly problematic. It forms an unintentional path for the escape of the LO signal from within the compartment.

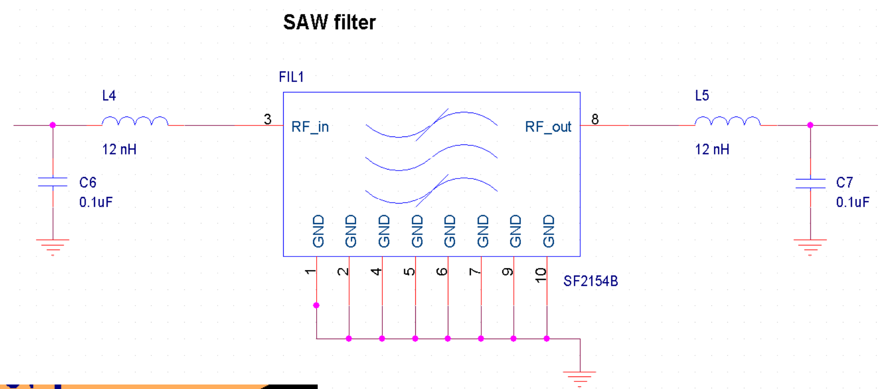
LNA routing, corrected



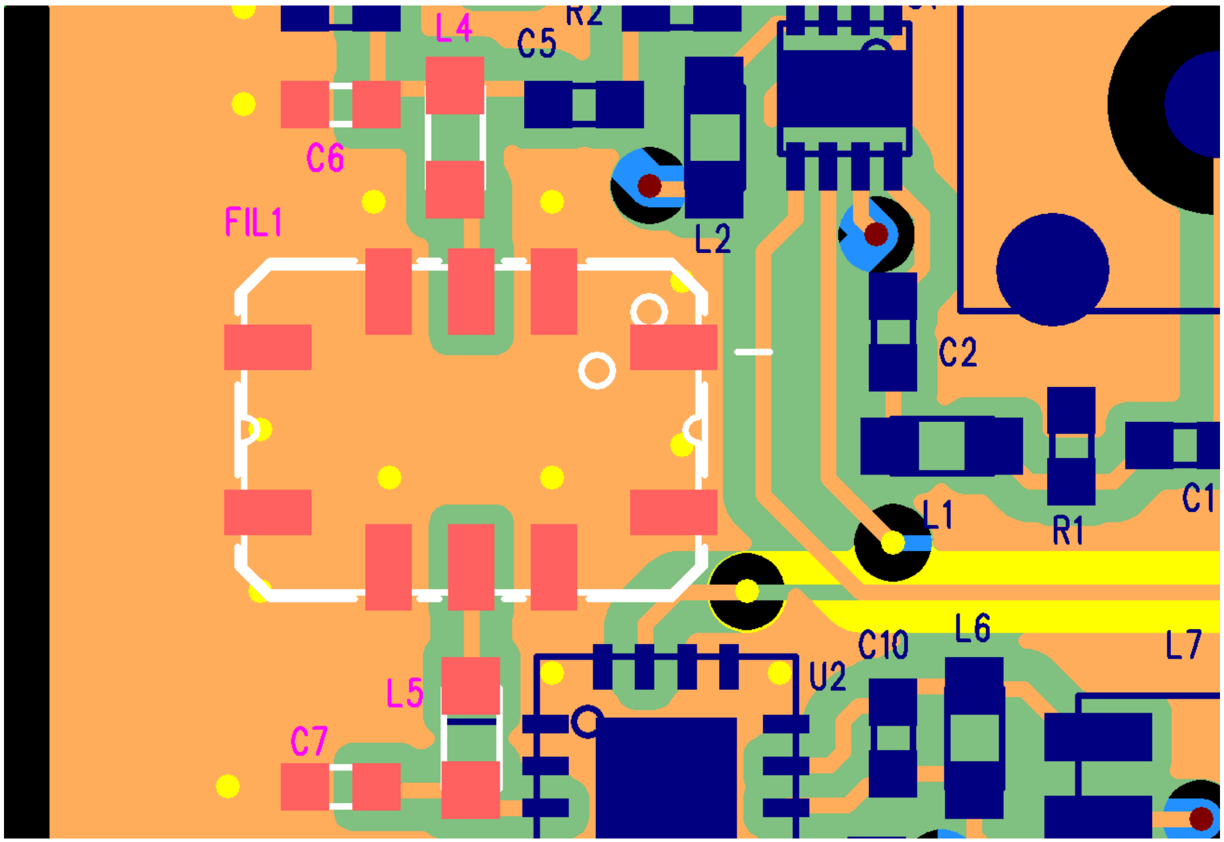


- Input is further from output
- Bypass capacitors are closer
- Ground slug is grounded with local vias
- Output bias part is at output
- Control lines avoid sensitive RF lines

SAW filter layout



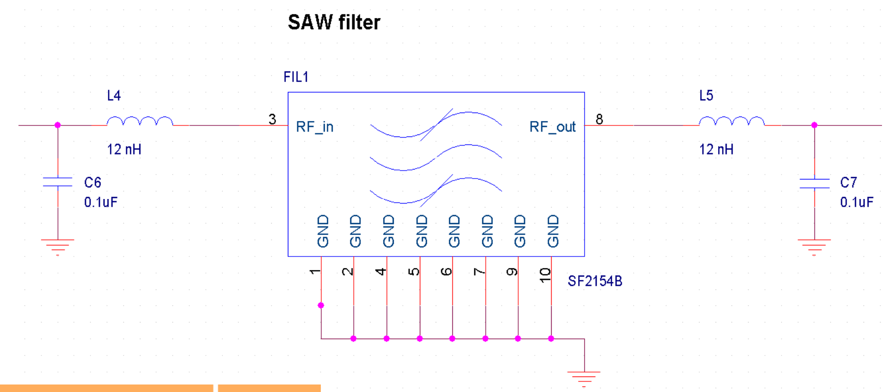
SAW filter Schematic and Layout sections

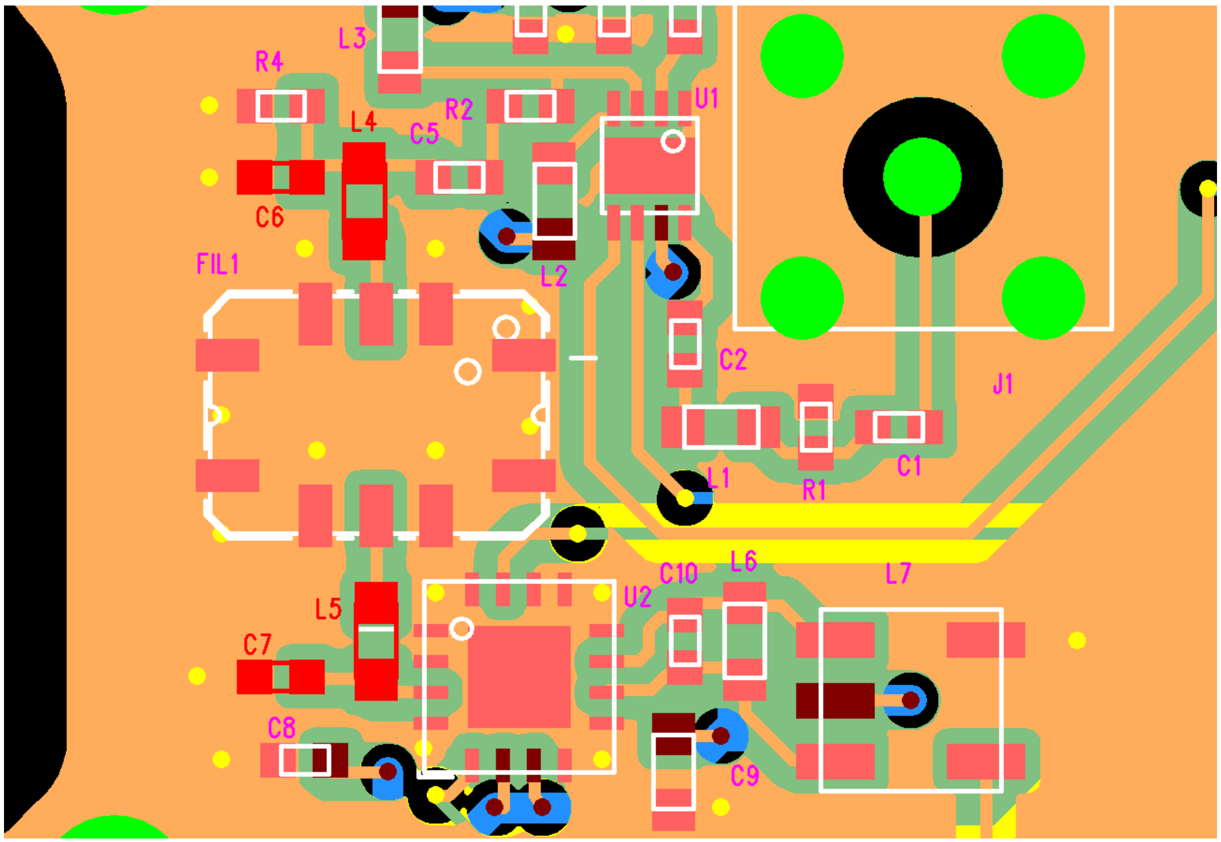


Isolation of input and output is important to improve filtering

SAW filter Z-matching components

L4, C6, L5, C7 are highlighted



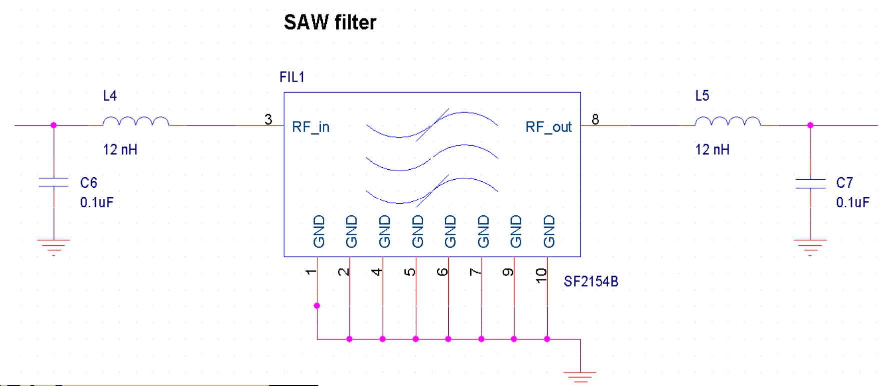


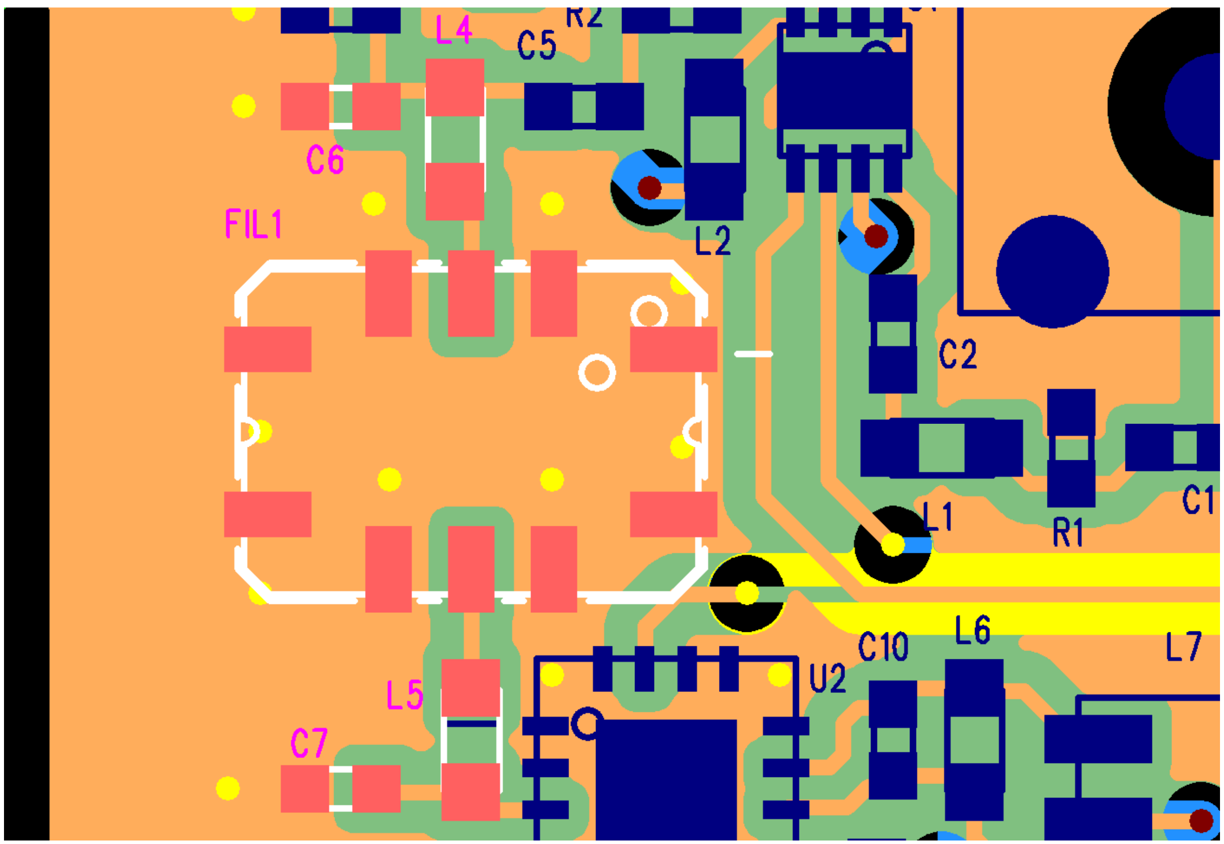
Both c6 and c7 form nasty stubs.

The two inductors should be at a right angle to each other to reduce magnetic coupling.

SAW filter plane slotting

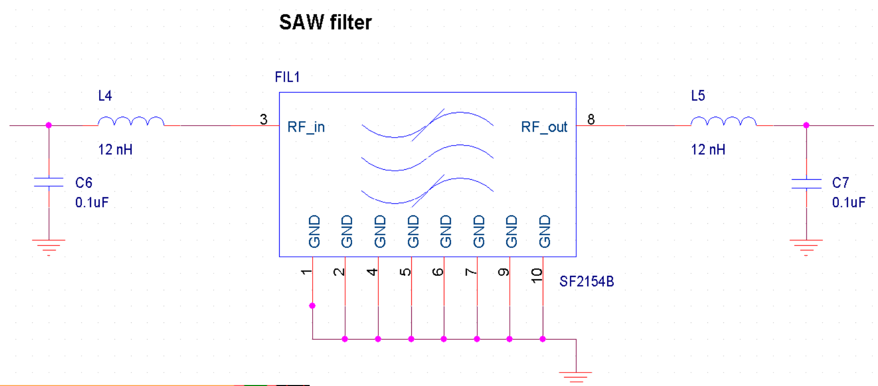
SAW filter missing GND slot



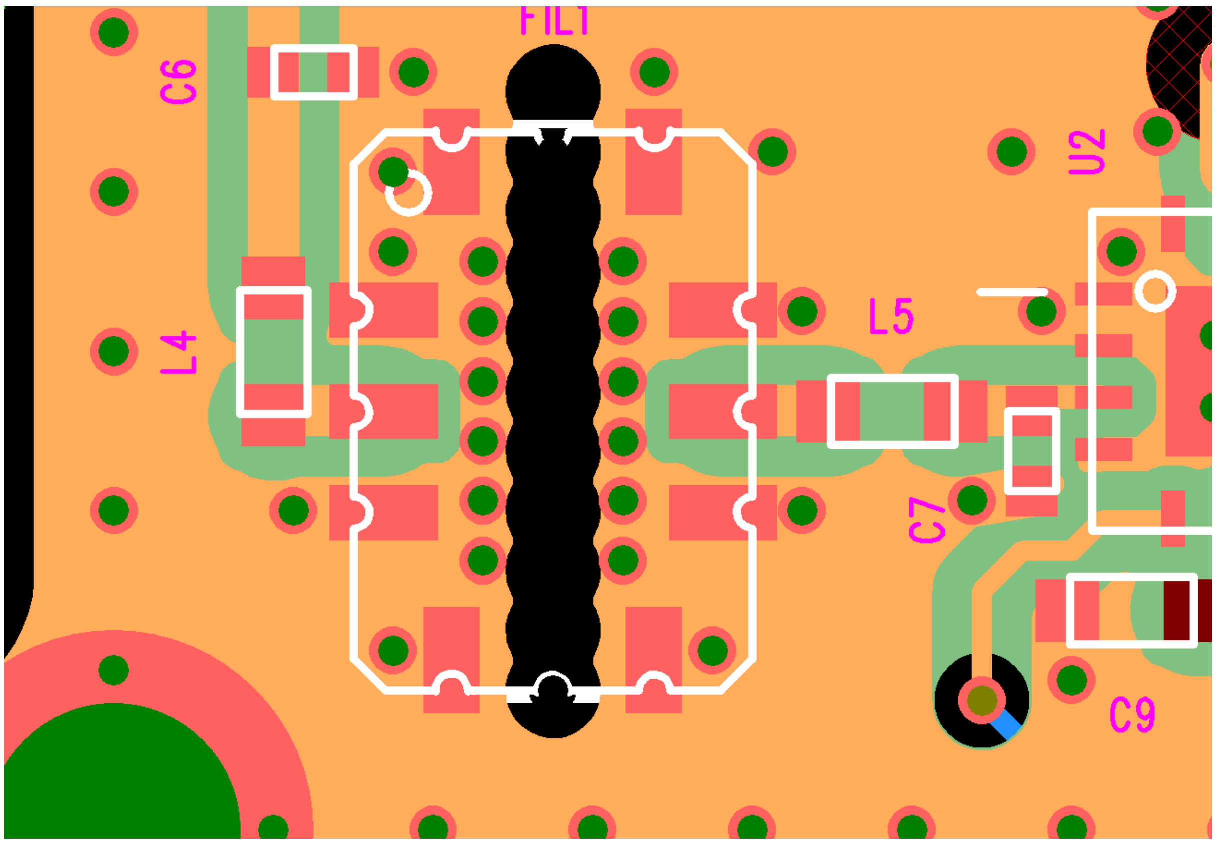


SAW filter layouts sometime have slots, or rows of vias to prevent RF coupling of the input and output. This layout lacks this.

SAW filter routing, corrected

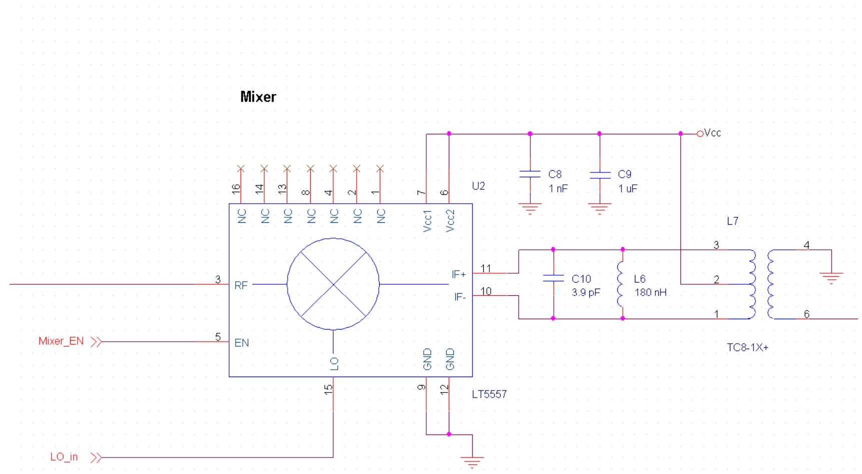


SAW filter layout with GND slotting

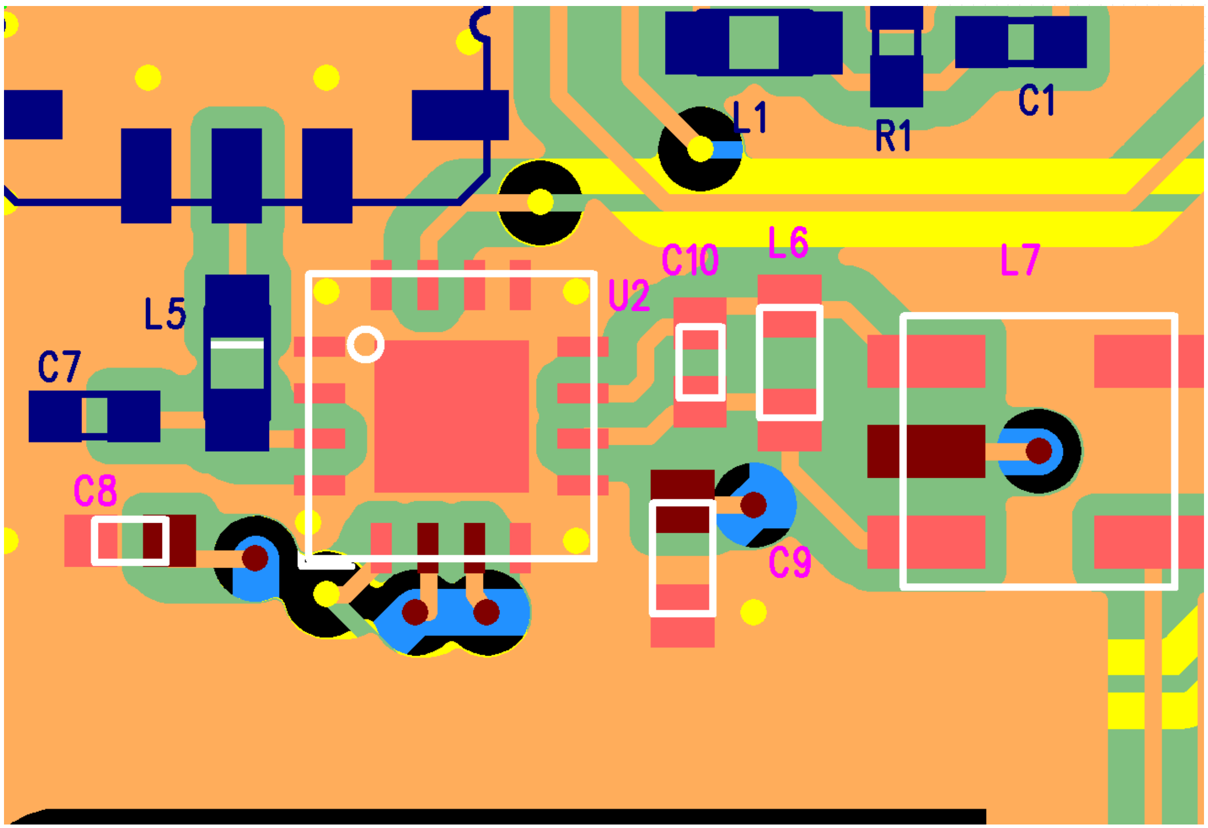


- Stubs have been minimized,
- Inductors in the matching circuits are at right angles
- Input and outputs are better isolated

Mixer Layout



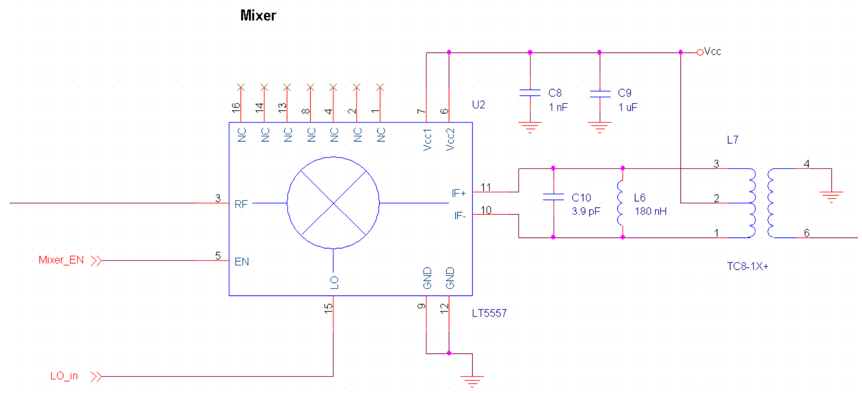
Mixer Schematic and Layout sections



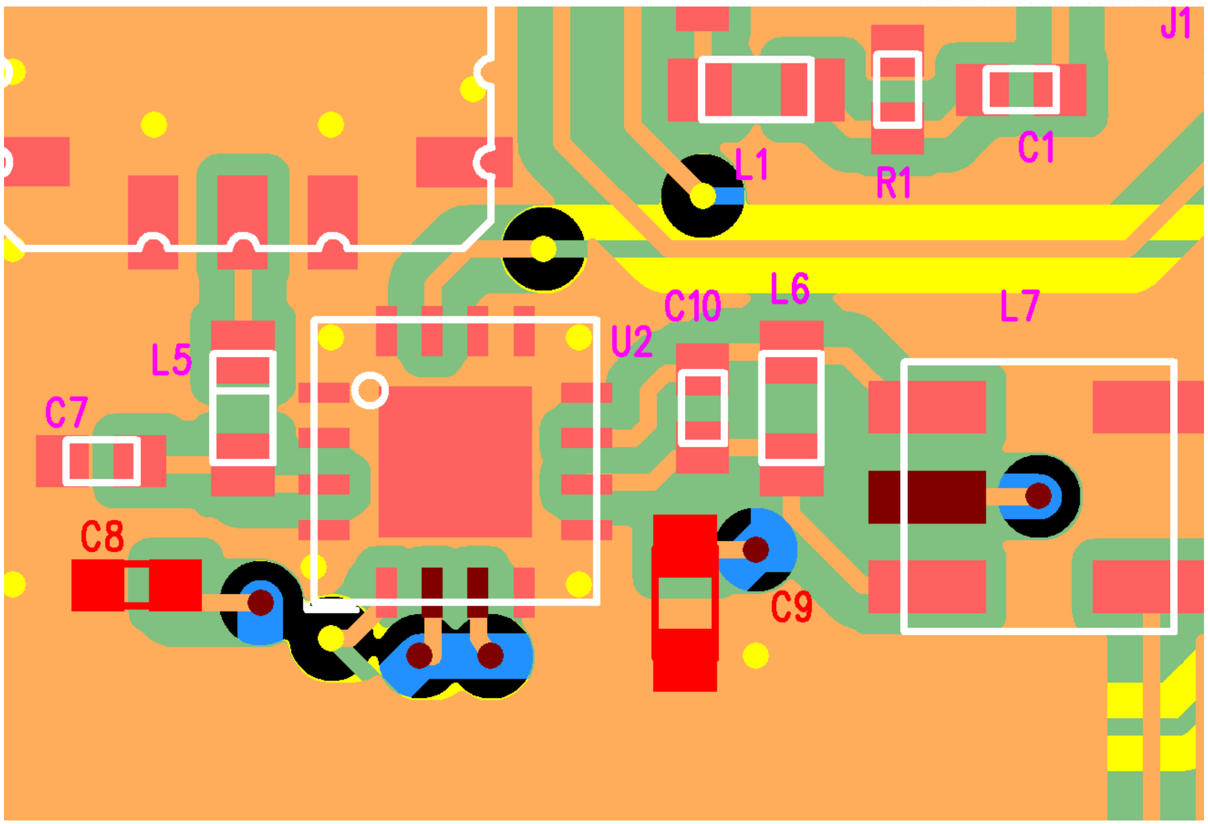
The mixer combines two input frequencies to create a third frequency plus many undesired byproducts.

This mixer has a differential output, which is converted to single-ended by balun L7.

Mixer IC Bypassing

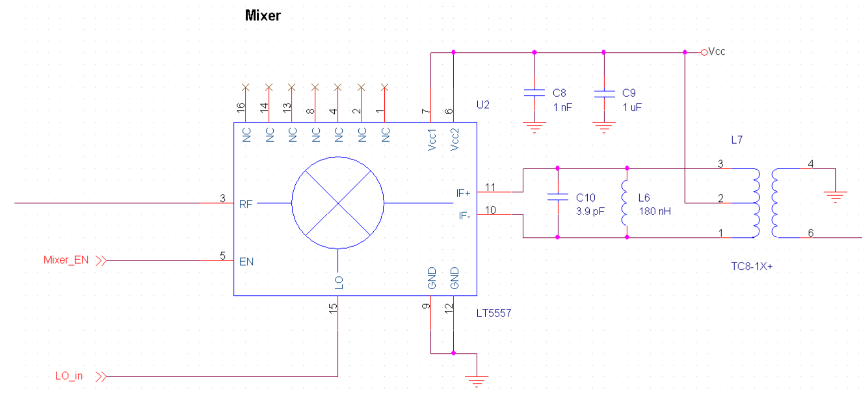


Mixer bypassing capacitors C8 and C9

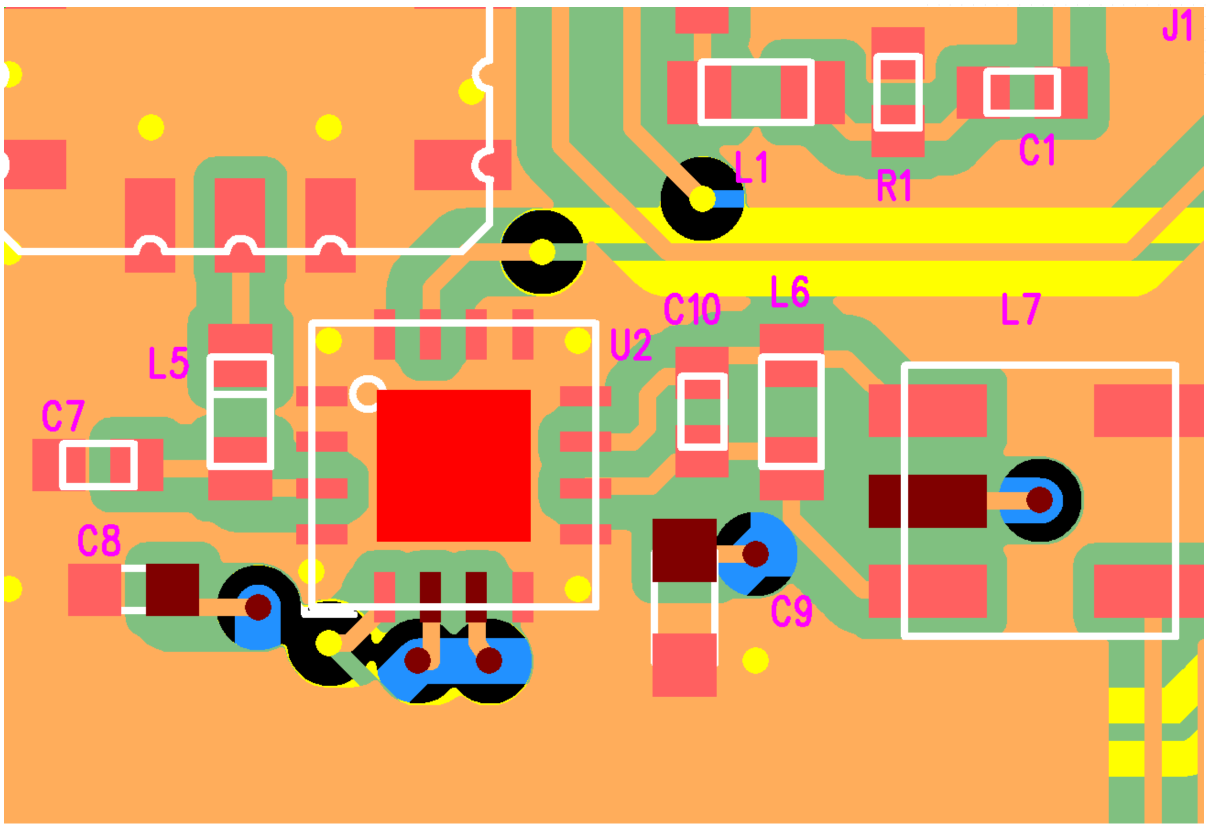


Bypass capacitors should be closer to the supply pins to reduce inductance.

Mixer IC grounding

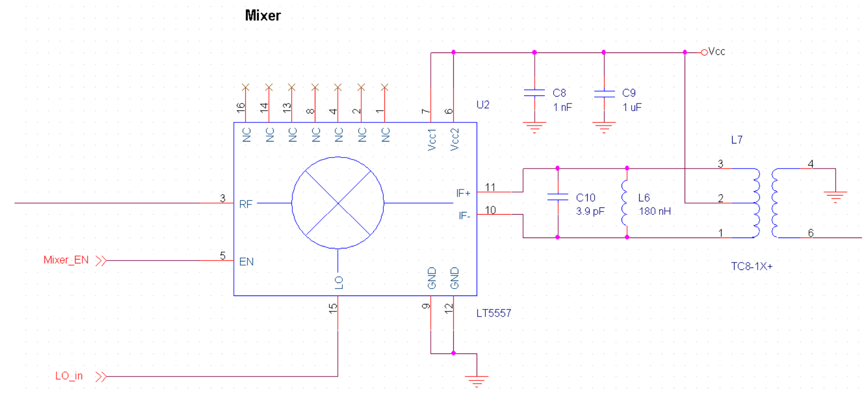


Mixer ground slug is highlighted

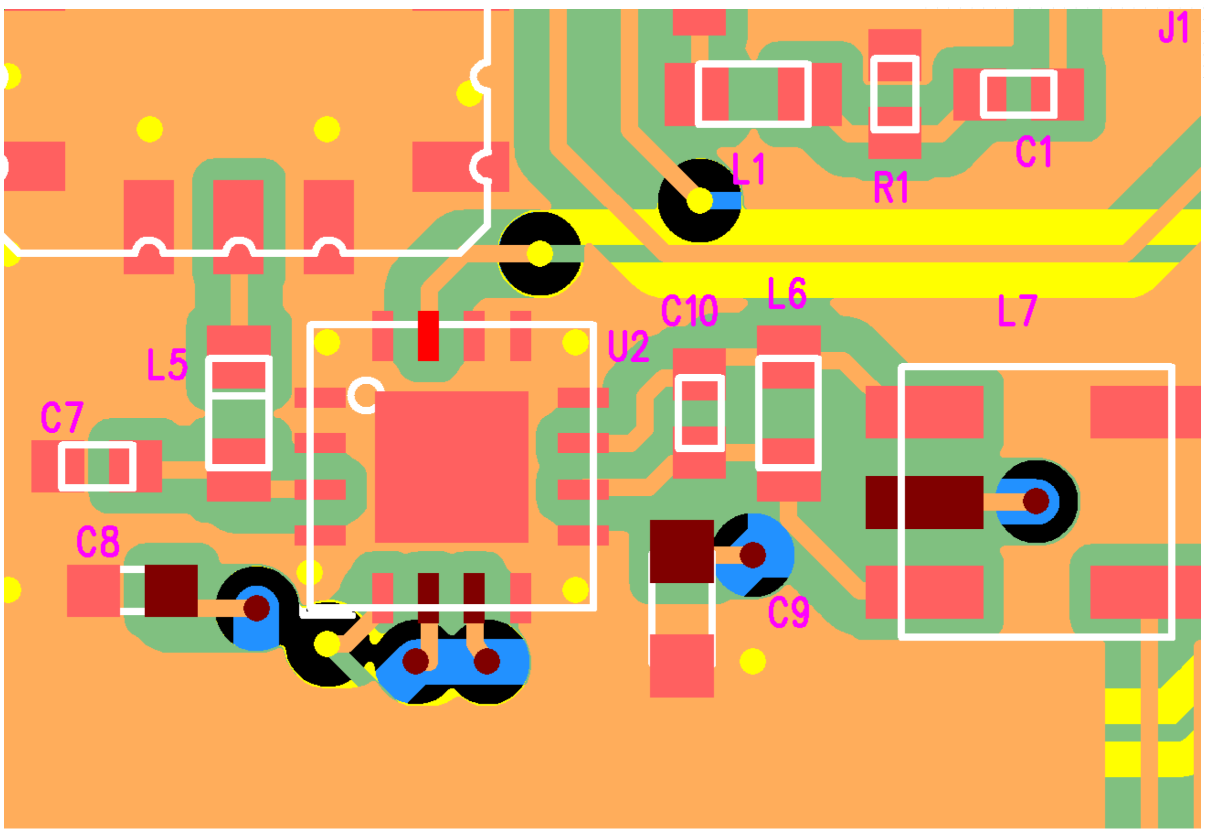


Absence of local GND vias in the mixer ground slug causes excessive return current loop area.

Mixer LO input



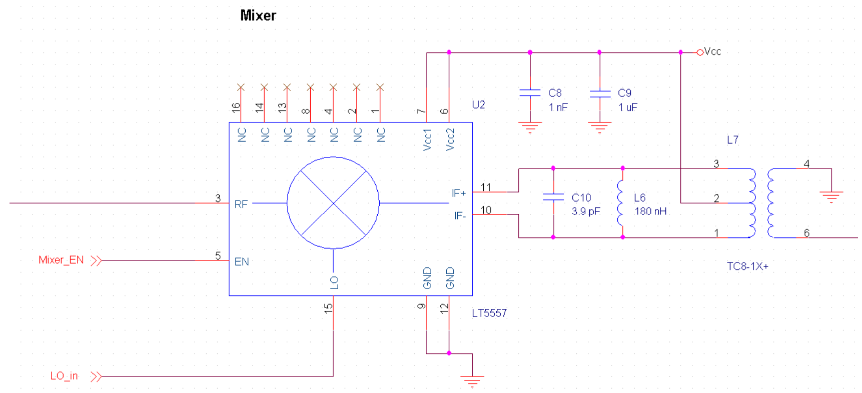
Mixer LO_IN pin is highlighted



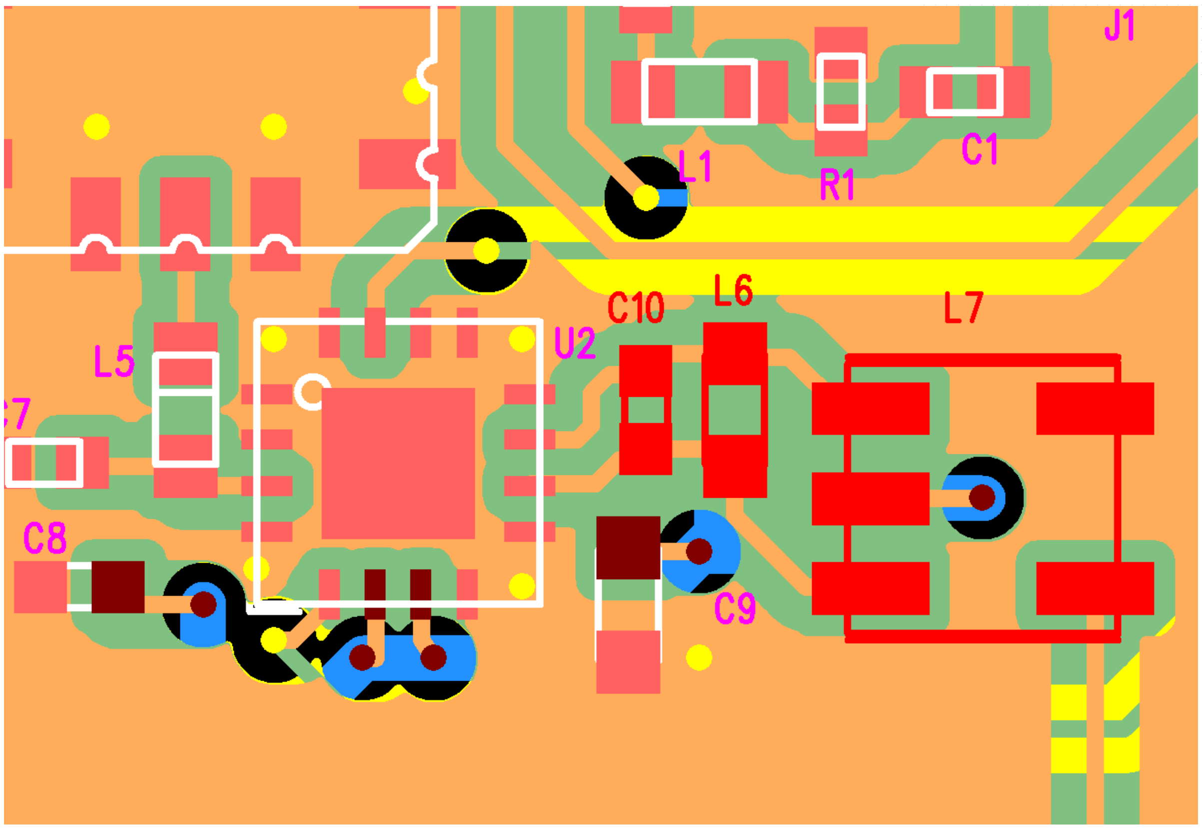
This is a sensitive RF line, routed directly under an adjacent control line from the LNA.

Its via also lacks a local GND via to form a return path.

Mixer differential output

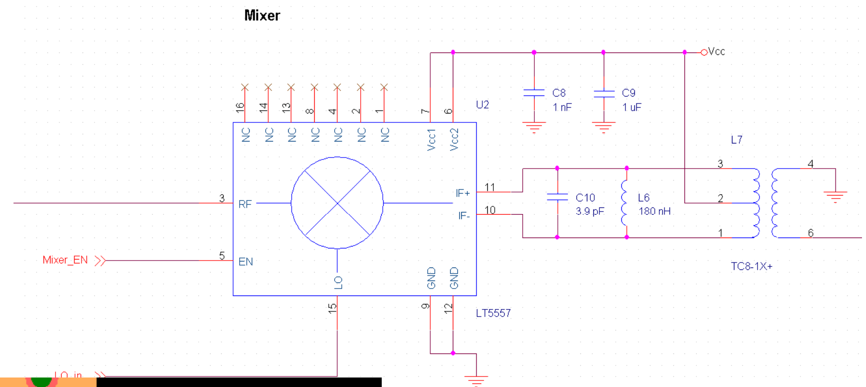


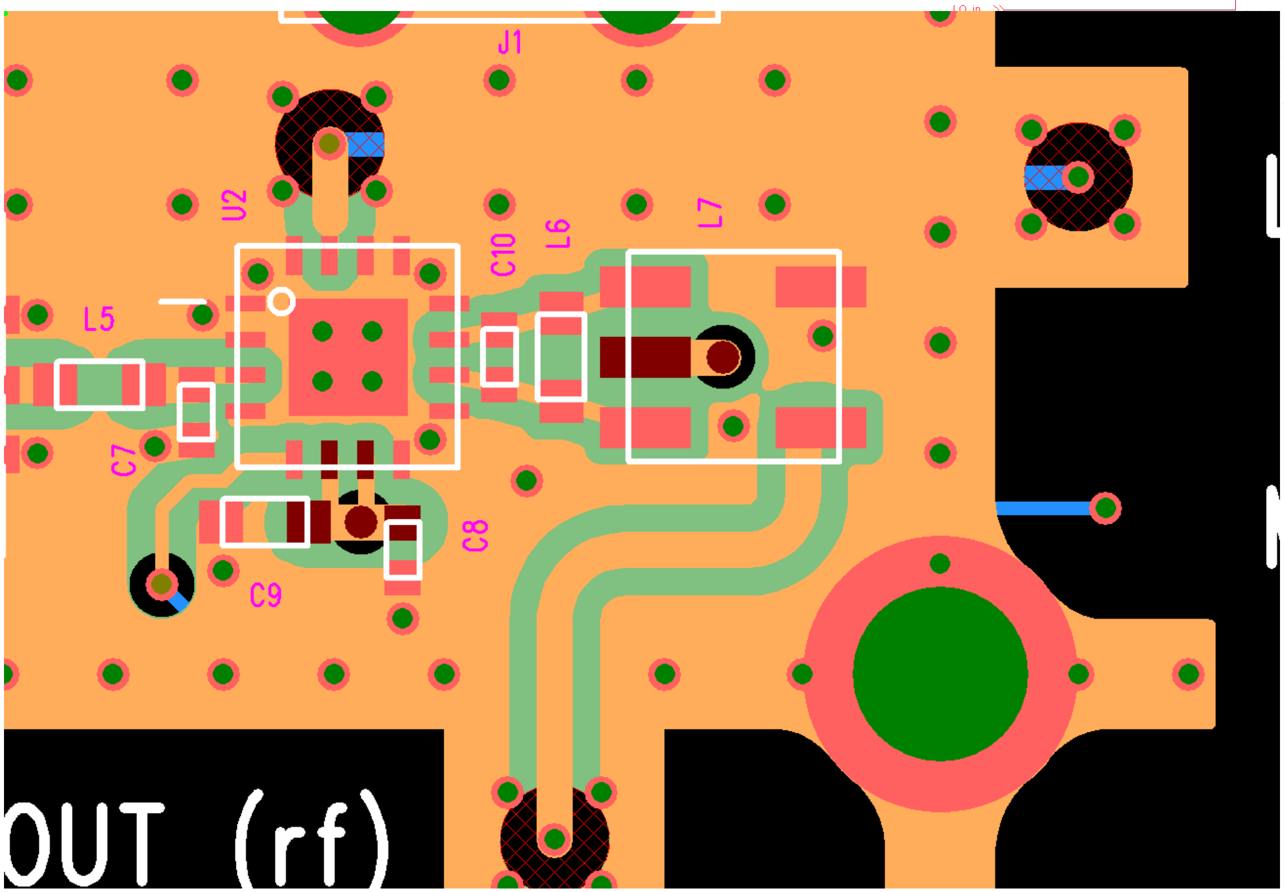
Mixer output parts are highlighted



It's actually not that bad. A little more symmetry would be nice.

Mixer PCB layout, corrected





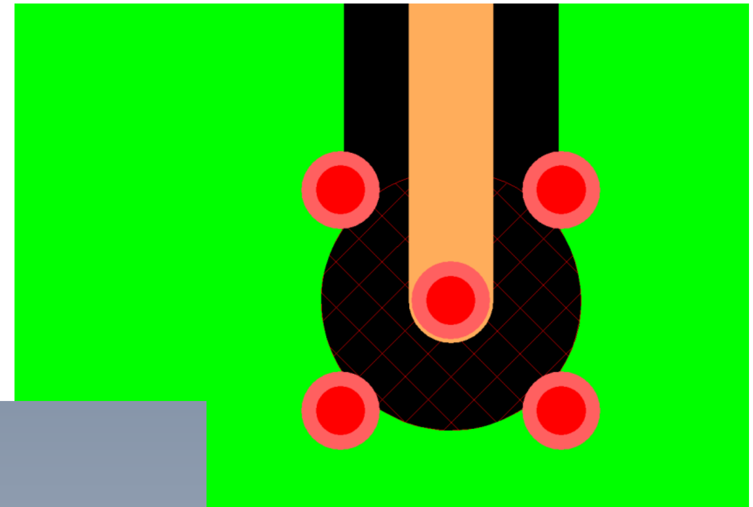
- The IC is correctly bypassed
- GND vias are present
- LO line has an RF via
- Output is nicely symmetrical.

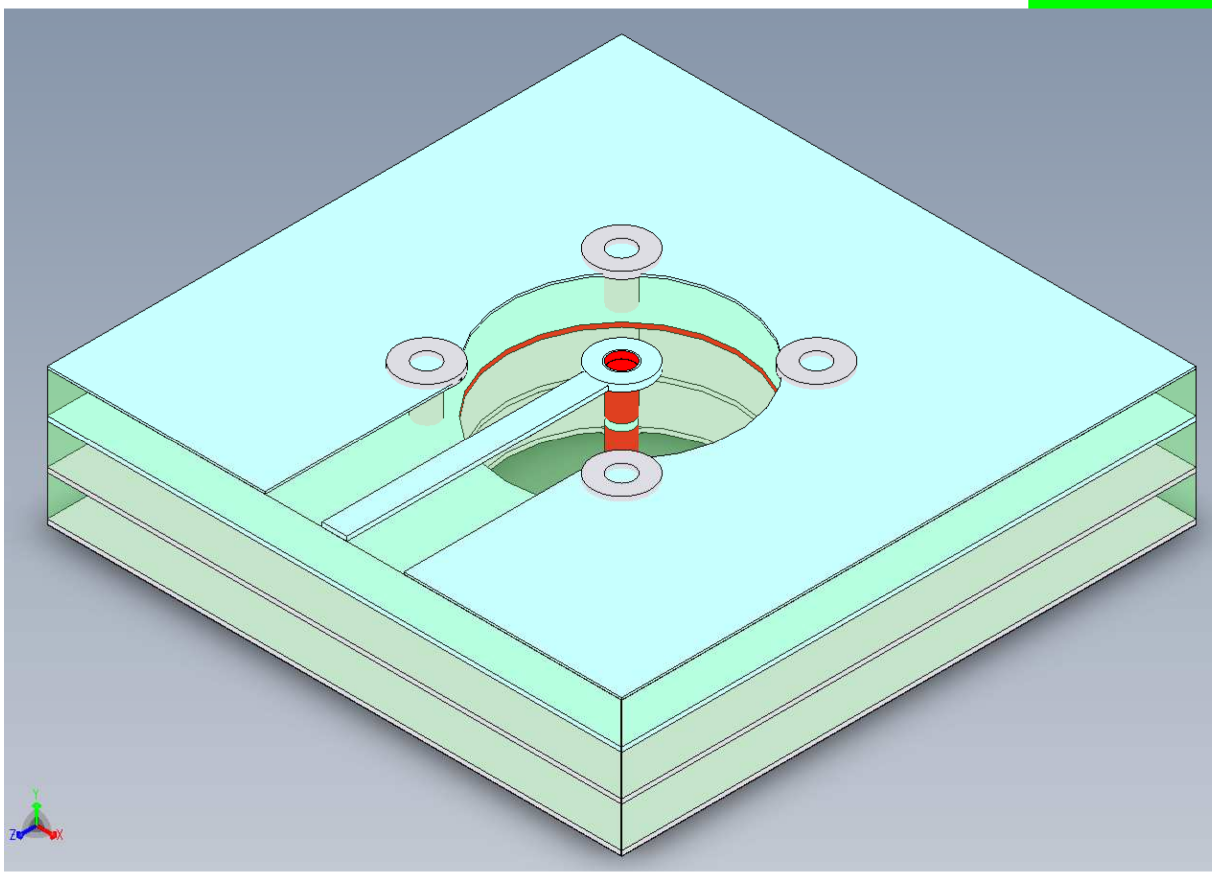
TIPS AND TRICKS

Simple ways to expedite the layout while catching sneaky RF layout problems.

Impedance controlled RF via

RF via with embedded local return path

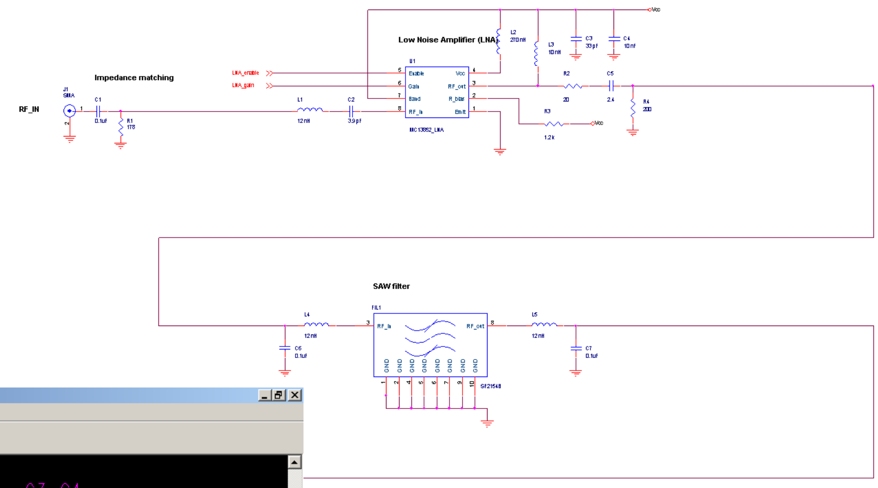


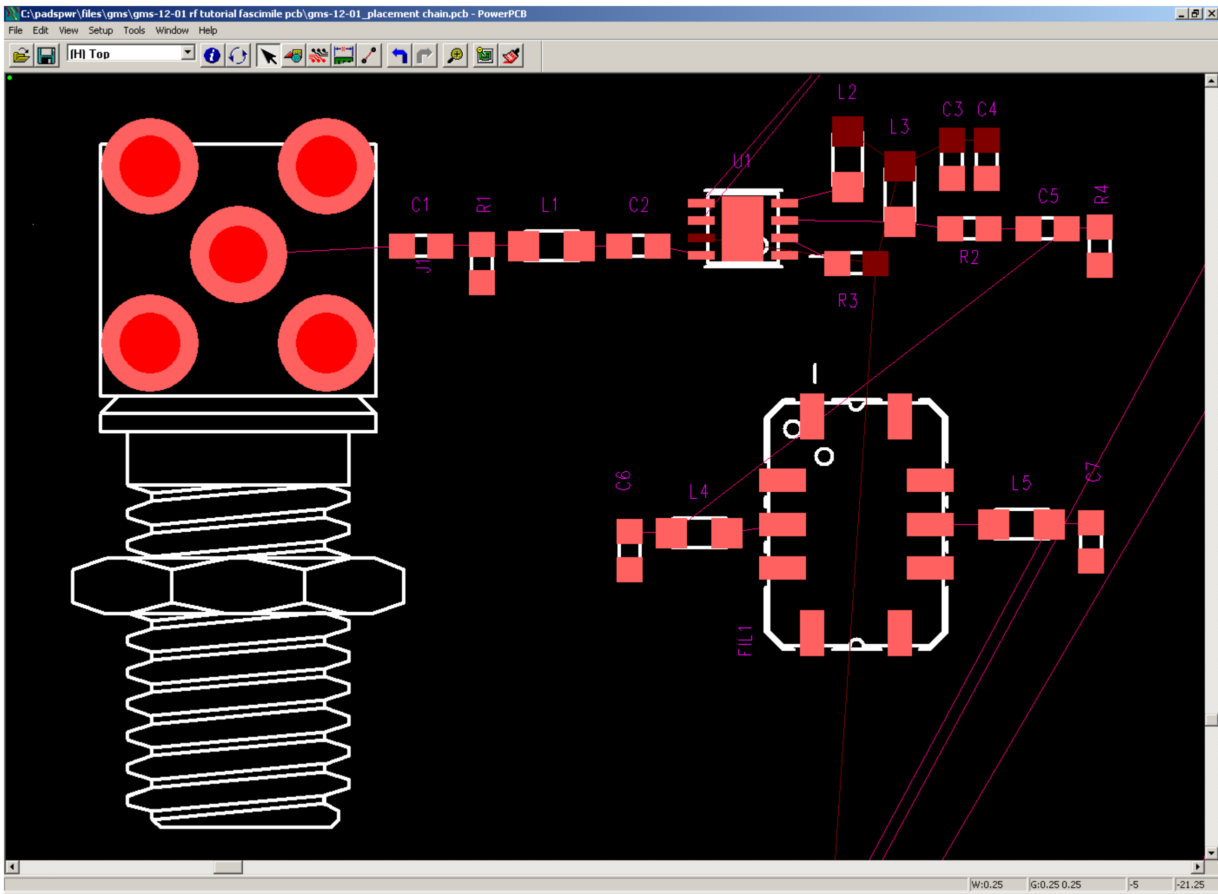


A web search of "impedance controlled via" will reference an excellent article on the topic.

Mimicking schematic placement on PCB

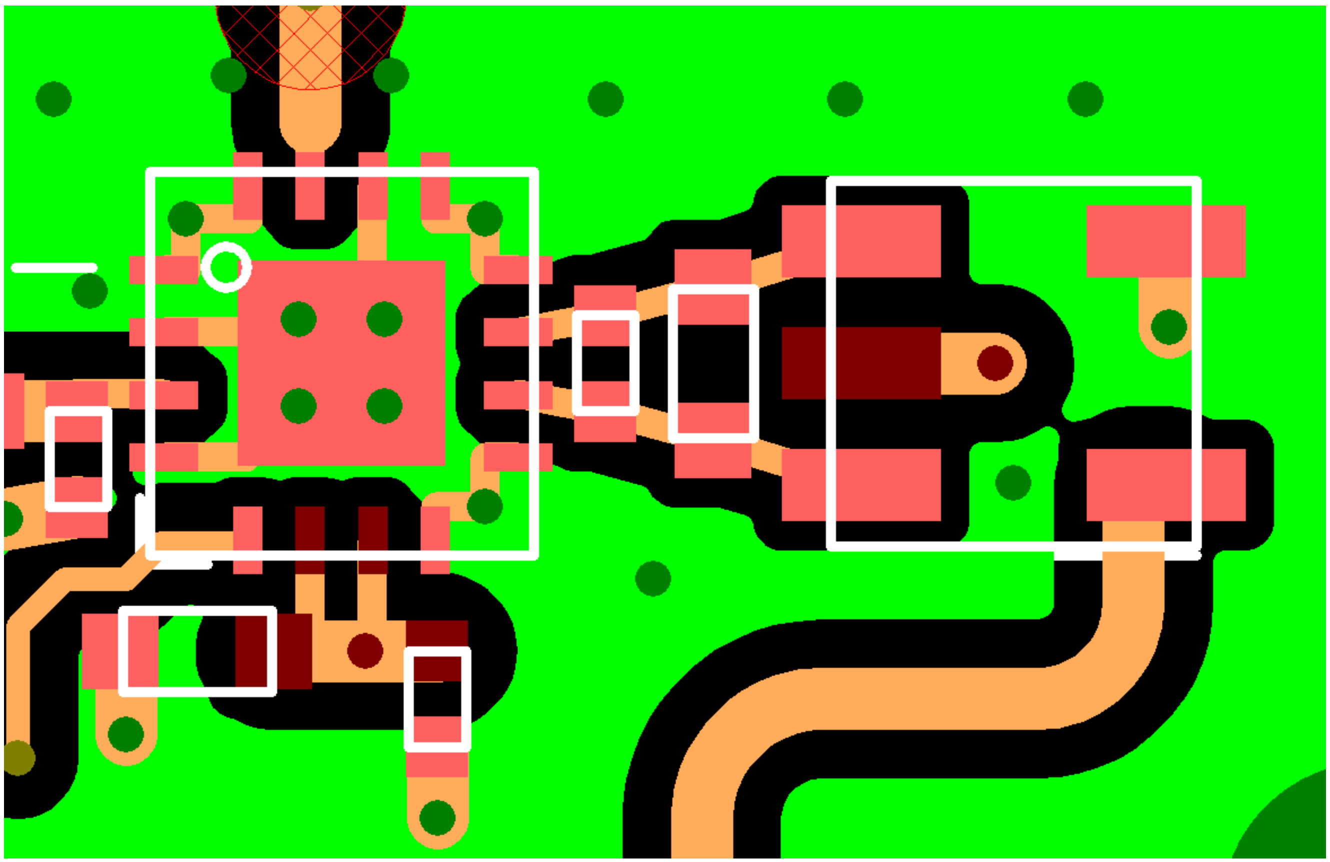
First organization of parts in the layout database





Cloning the relative locations of the schematic symbols in the schematic into the layout database is a good starting point, and expedites parts placement.

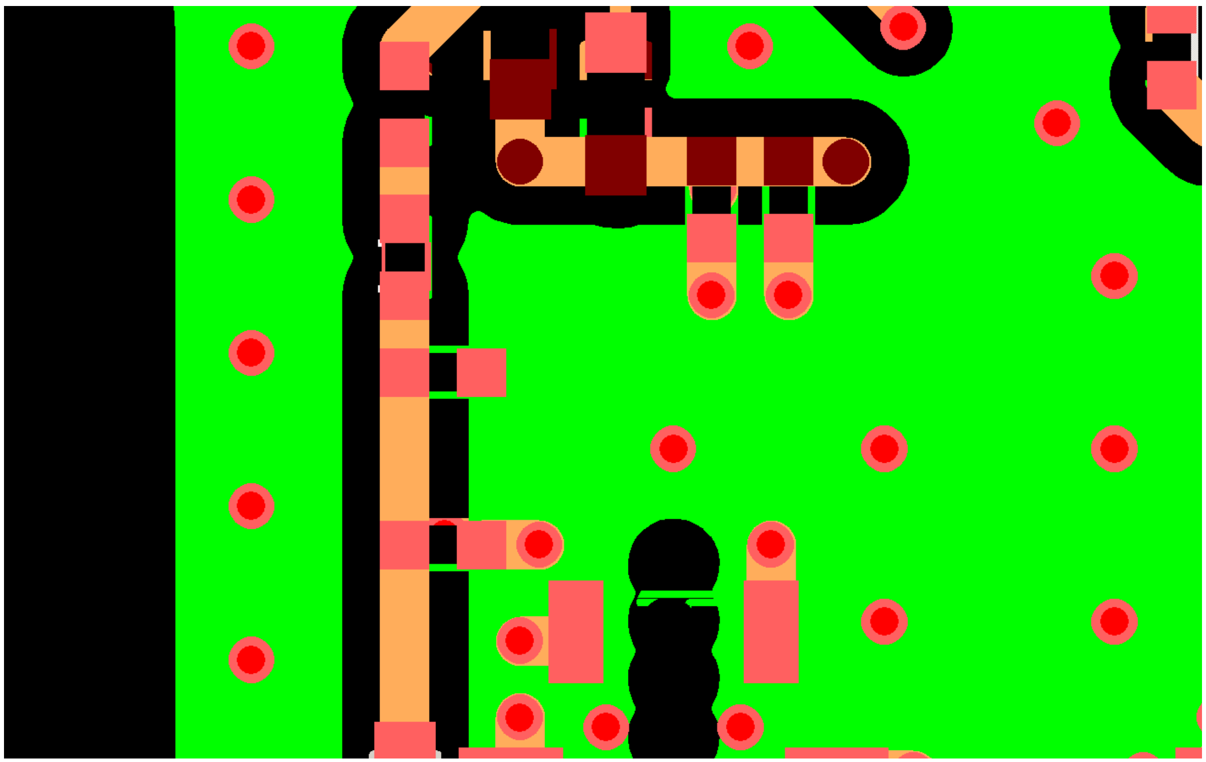
Cloning Evaluation boards



Cloning the general placement and layout of the evaluation board is a good rough draft for the PCB layout itself.

Finding GND pour stubs

Shunt components in Layer 1 GND plane, “bad pass”

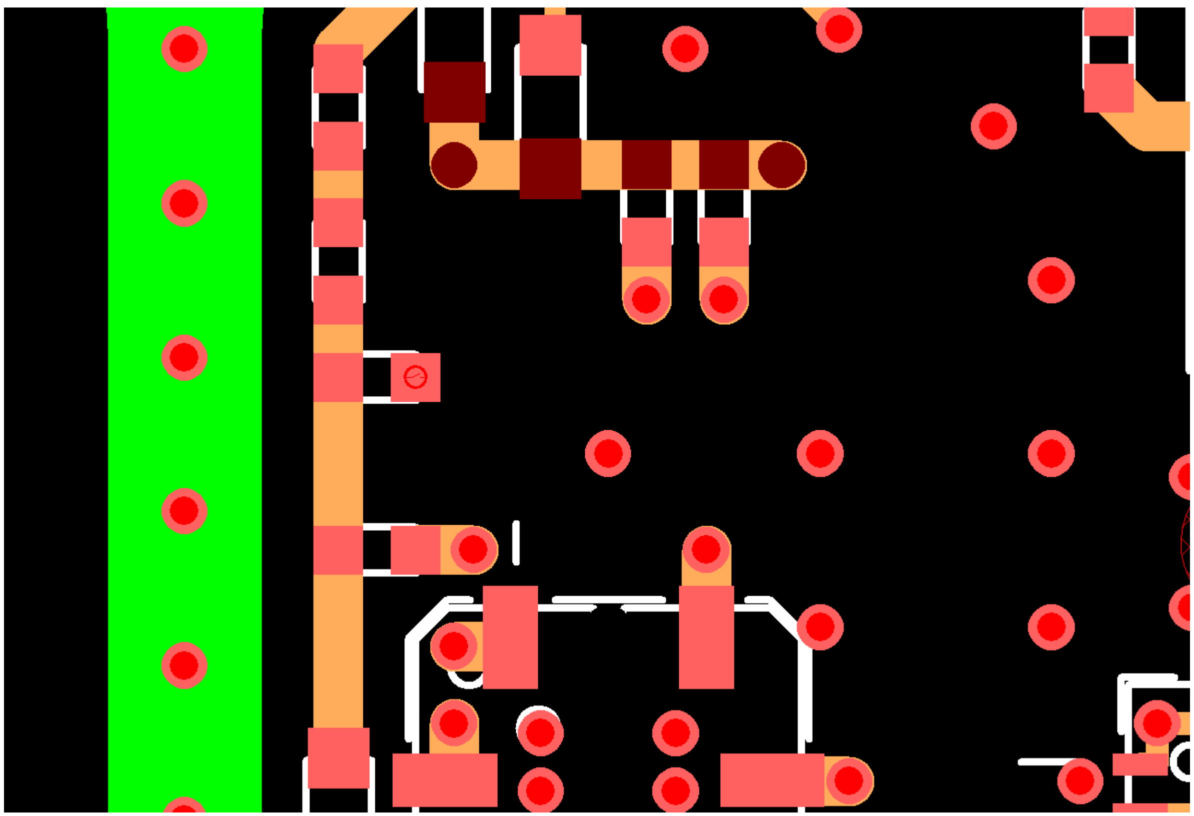


This connectivity check was performed with GND pour “on”

As a result, a relatively long GND stub was given a pass.

GND pour stubs, cont.

Shunt components in Layer 1 GND plane, “good flunk”



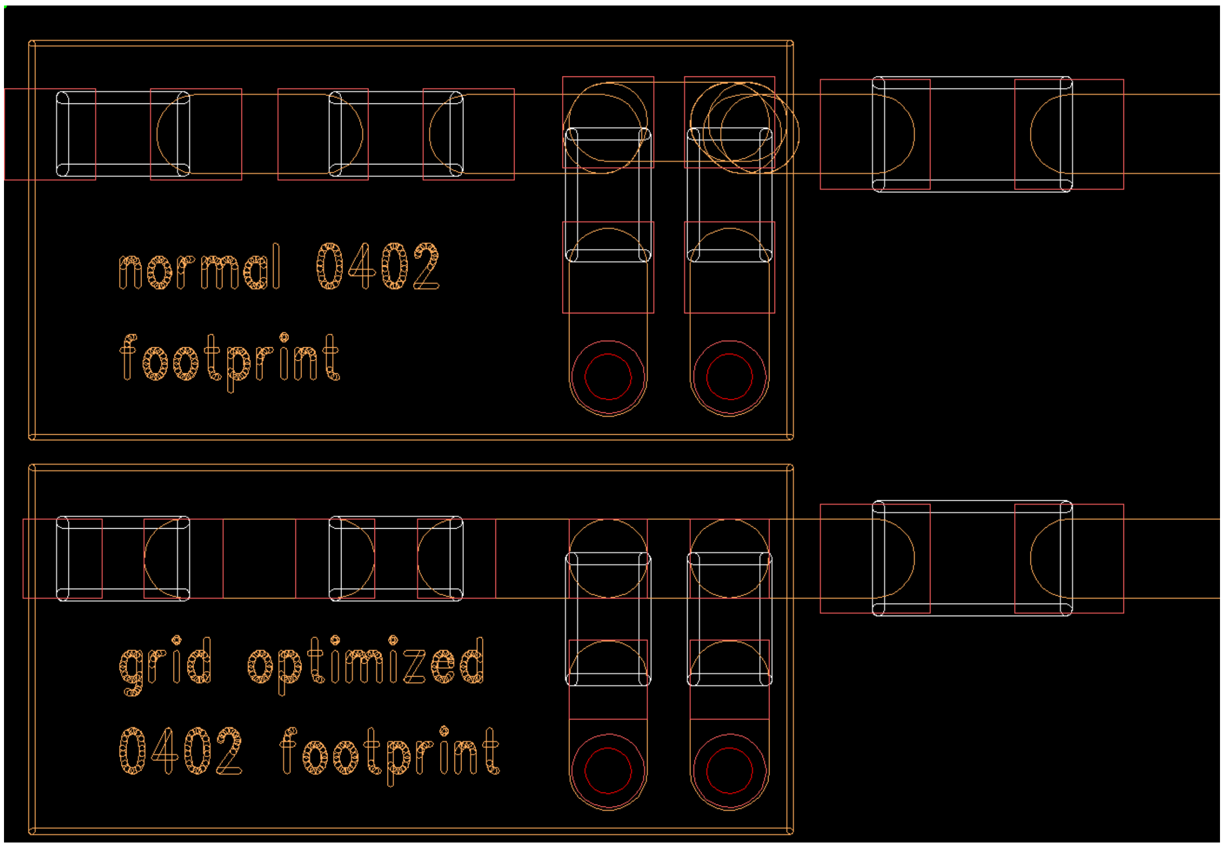
This connectivity check was performed with surface pours off, instead of on.

GND connections with missing local vias are flagged as errors. Long stubs become more visible.

This trick works well to detect missing GND slug vias.

RF optimized footprints

RF optimized 0402 vs. manufacturing optimized 0402



Reduced pad size improves parasitic capacitance

Pad grid is explicitly defined "on grid" to avoid off grid routing and internal angles.

0402 and 0603 pads are relatively easily optimized to match trace widths.

Organization of RF and control nets

Assigning RF and control nets to different classes makes it easier to mentally organize the nets.

Facilitates preferential routing and checking of the more sensitive signals.

Thank you very much!

Rick Gendreau

Gendreau Microsystems, Inc.

PO box 1234, Rustburg VA

rhg@gendreaumicrosystems.com