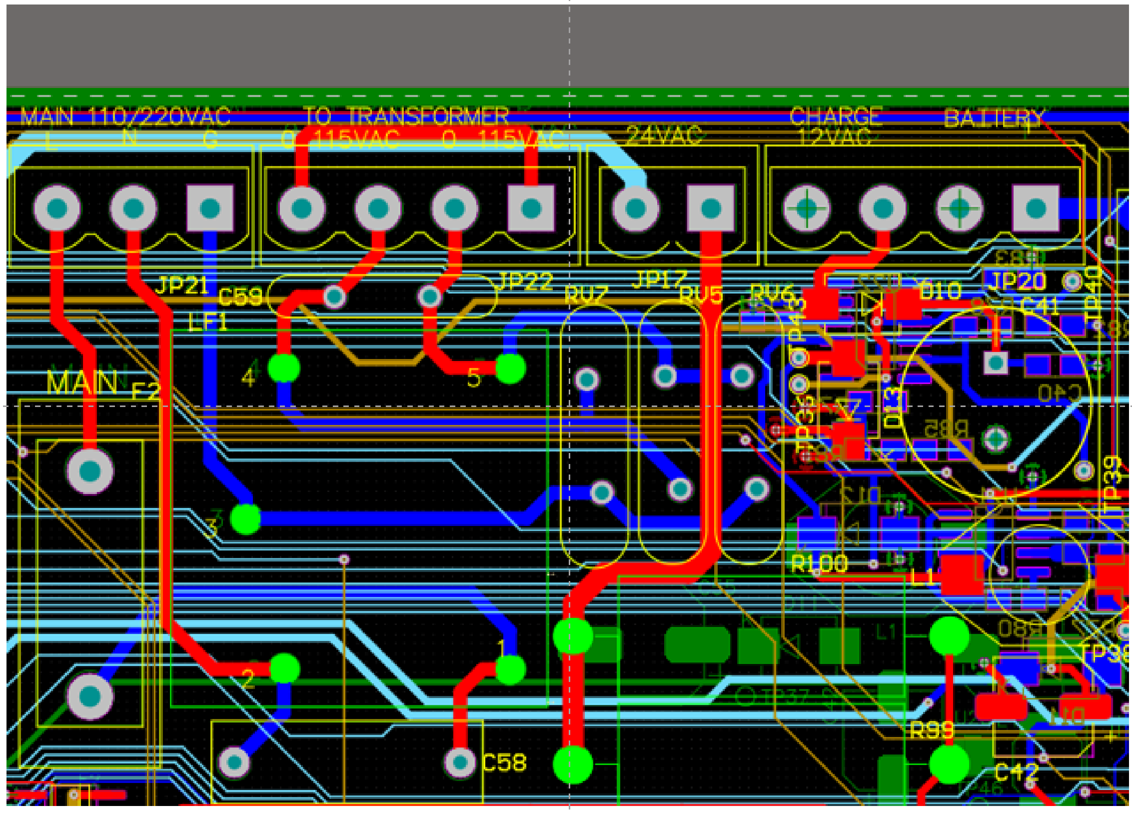
Review schematic and layout.

Issue: burning out Wi-Fi module and MCU of Device, Wi-Fi sitting on VCC2 and MCU on VCC power lines.

PCBA issues;

1) AC Input/output lines and AC filter should be assembled as independent part of device with transformer and connected to main PCB with short wires. Or should be placed on PCBA as a different part without impact on DC lines and DC power planes.

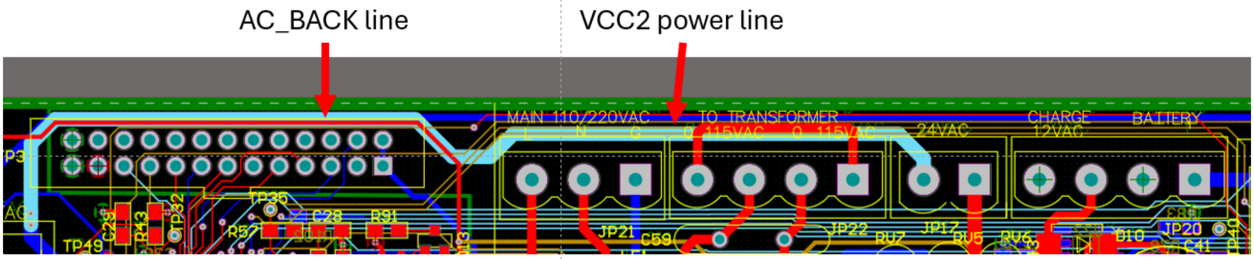
VCC & GND Power planes under AC filter has potential risk of EMI



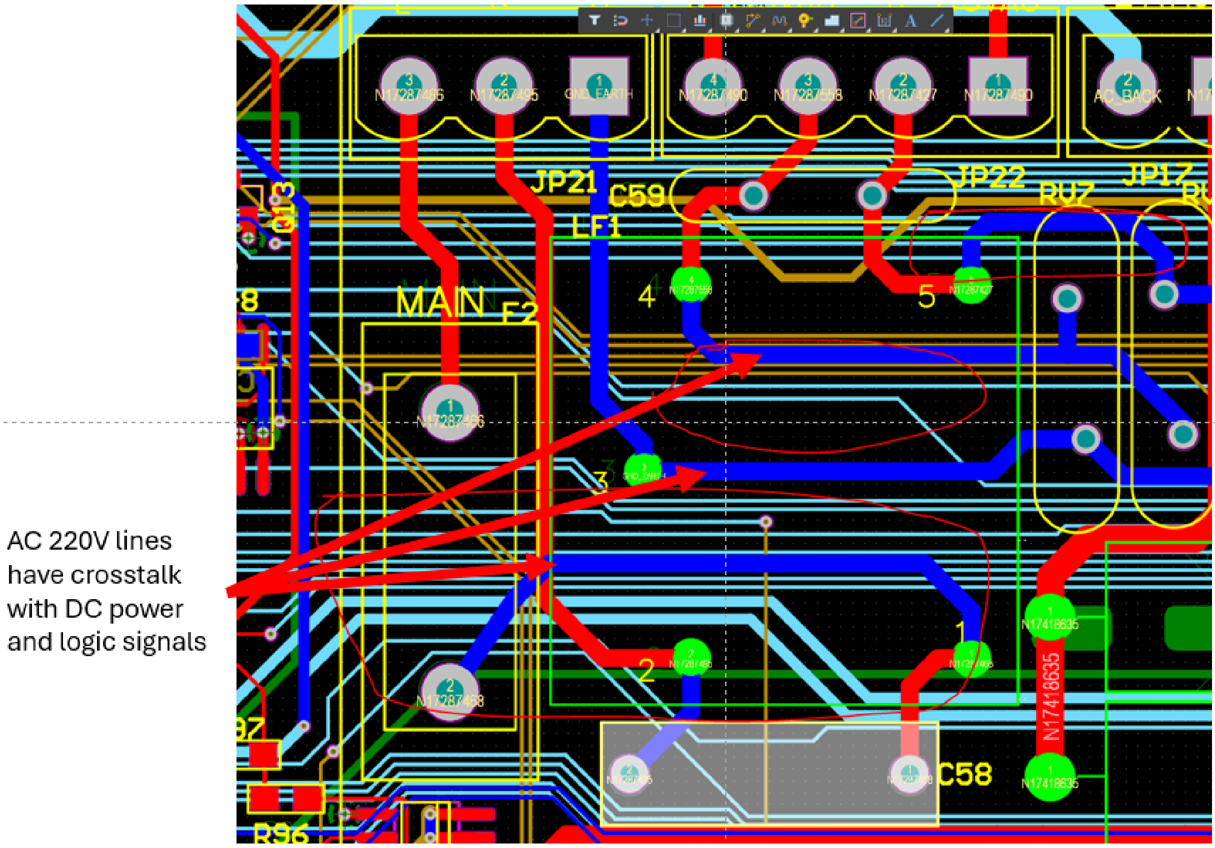
2) If “a” cannot be done, power planes under AC filter should be removed at all, as well it should be done in all places where there are AC lines on the PCB and all other DC signals should be layout not close to AC lines (2-5mm).

2024-10-25

3) There is long crosstalk on the PCB between VCC2 power line and AC_BACK line, which has very potential risk that ESD event near the solenoid cable will reflects to VCC2 line.

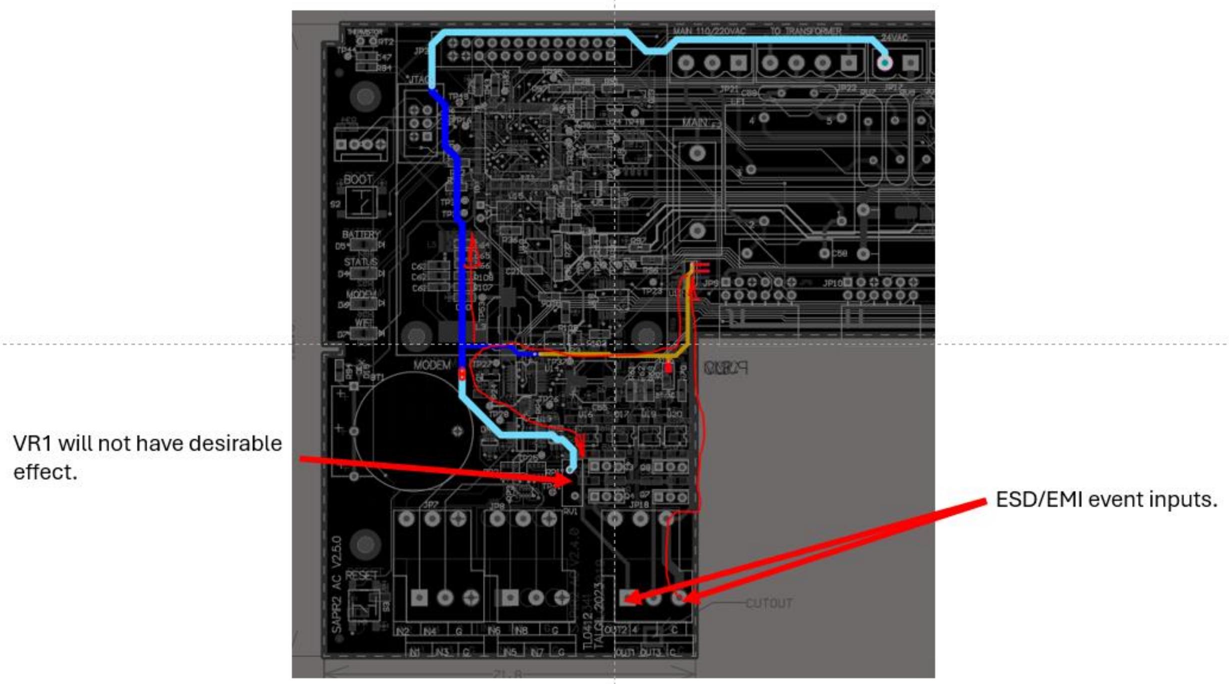


As well there are other crosstalk issues between AC 220V and DC lines VBAT and VBUS and other DC signals.

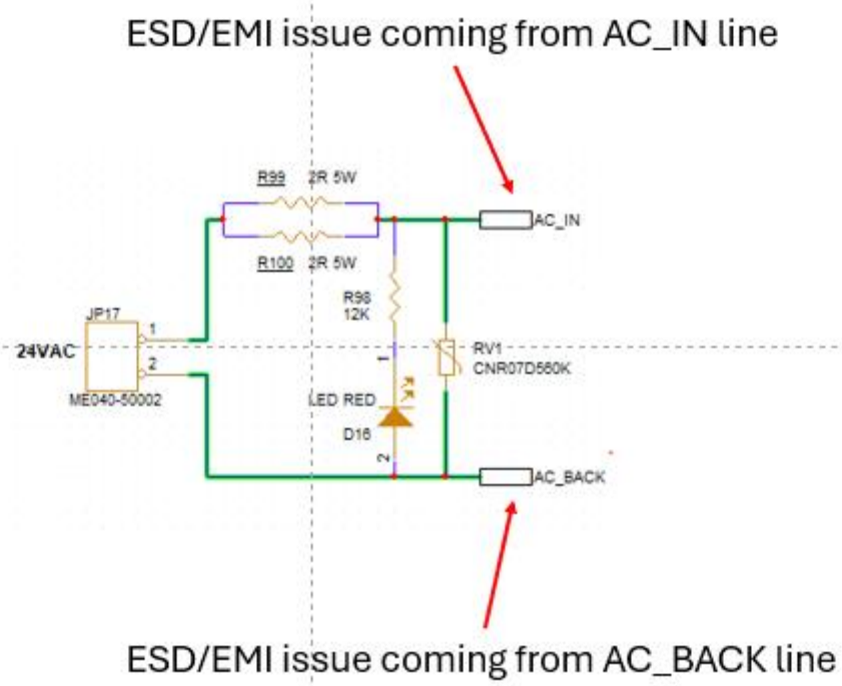


2024-10-25

4) ESD/EMI Event propagation on PCBA via 24VAC line and impact of RV1 use case.



Varistor use case;

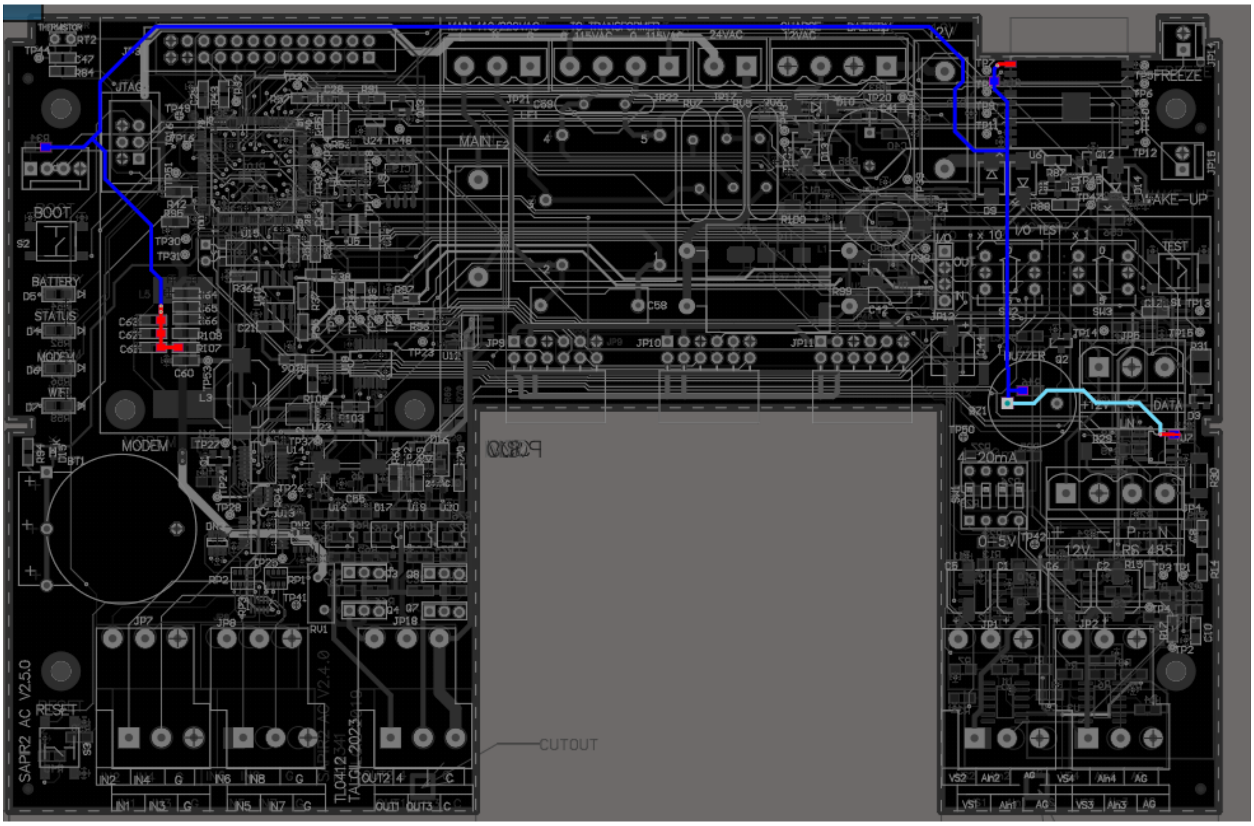


Action when both wires have the same voltage level: If both wires (AC_IN and AC_BACK) have the same overvoltage level at the same time, the varistor may not operate because the potential difference between the two wires will not be enough to activate it.

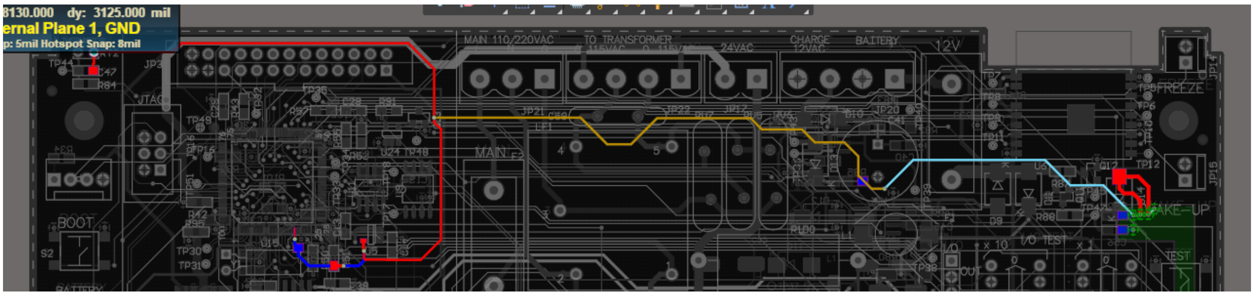
To avoid such a situation, we need to use varistors or TVSs from both lines to power GND.

2024-10-25

5) VCC2 should be polygon and not strip line, has potential dropping voltage on such line.

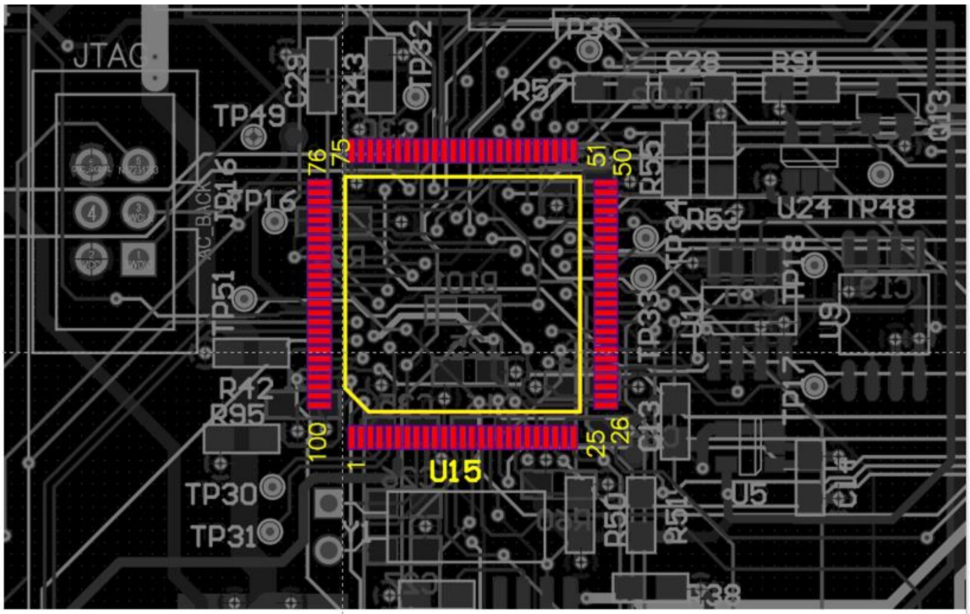


6)GND Field Signal should be polygon and not strip.

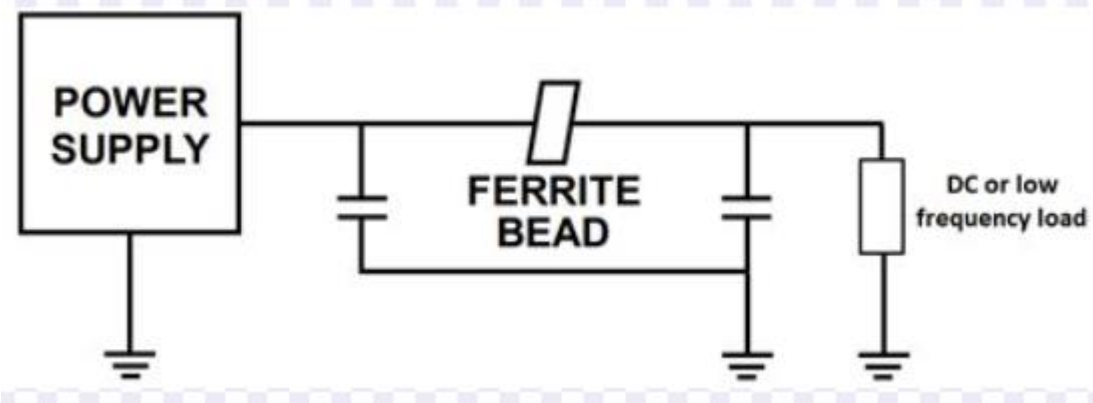


7) MCU should be surrounded with GND polygon.

Need solid GND polygon surround MCU.

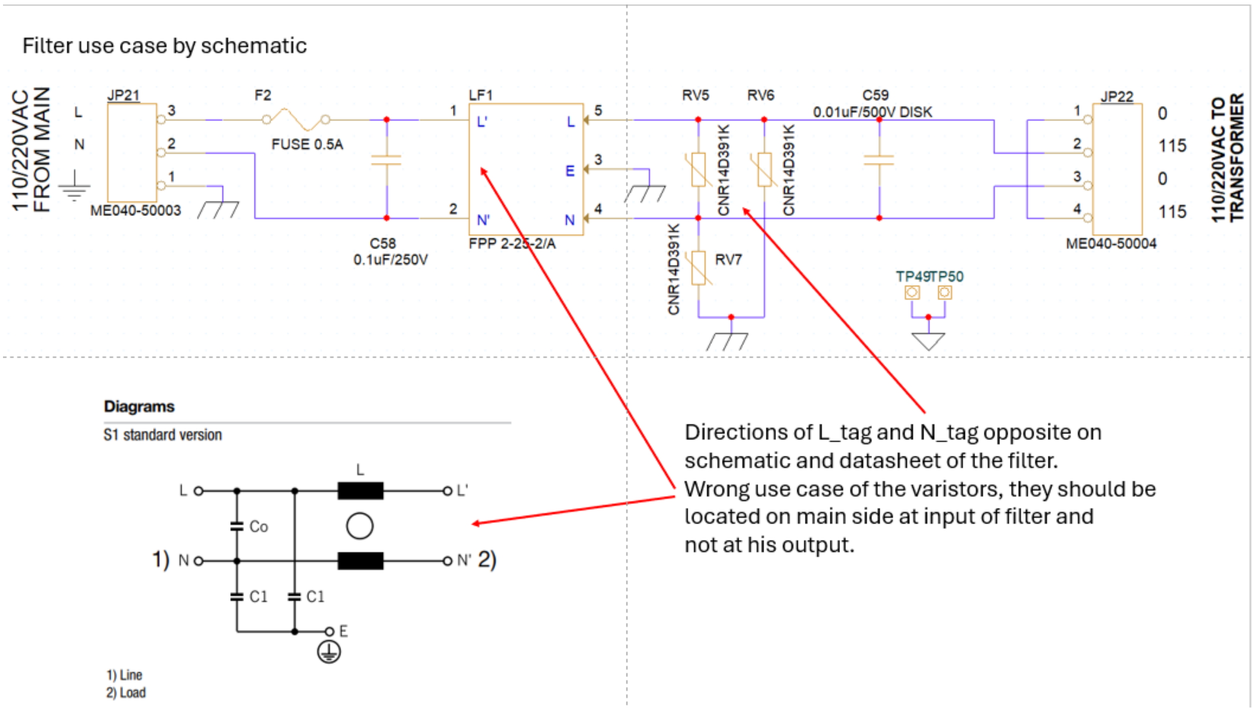


8) MCU, Wi-Fi and other logic circuits power inputs should be filtered with ferrite beads like in the picture. It will reduce or eliminate EMI event and save the power input of components.



9) AC filter use case issue.

Theoretically, if we have voltage spike the high voltage will pass through filter will reduce but stay.



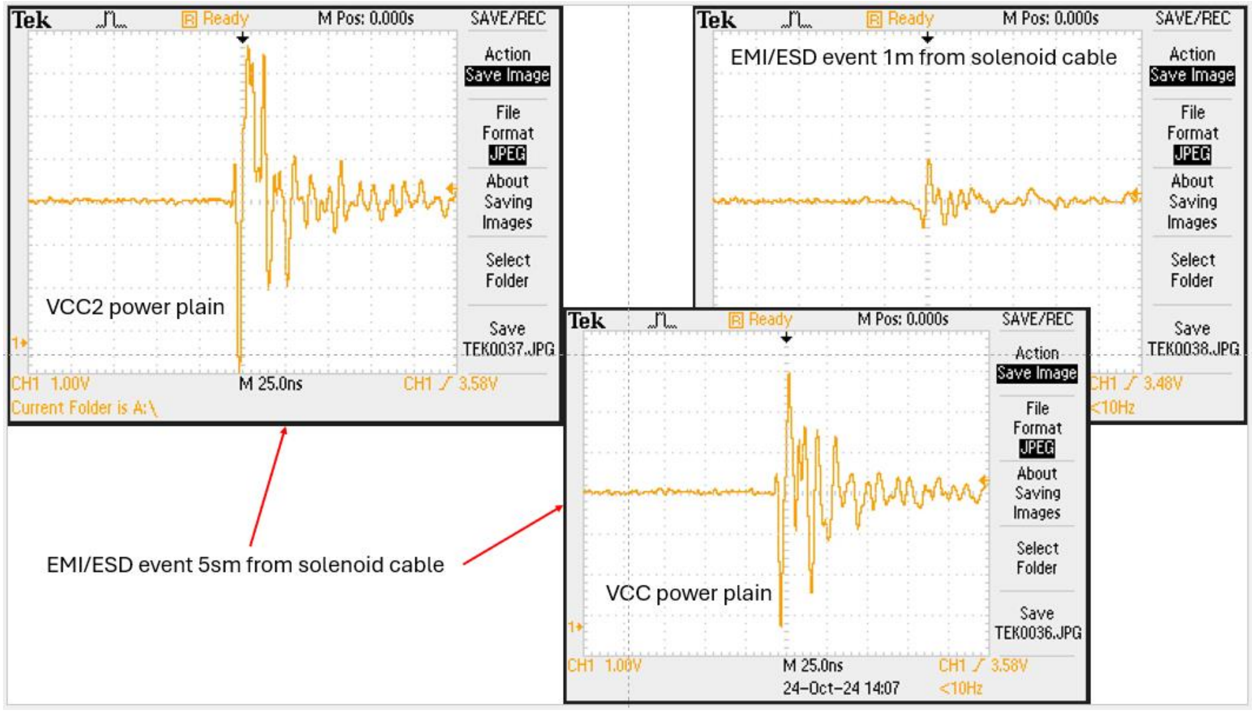
Here's what can happen in this situation:

The filter partially attenuates the pulse: The filter suppresses the high-frequency components of the pulse, but does not completely eliminate its amplitude, especially if the pulse has low-frequency components or if the pulse energy is too high for the filter to completely suppress.

Not enough voltage to trigger the varistor: After filtering, the pulse voltage may be below the varistors' trigger threshold, causing them to not turn on and not absorb the pulse energy. In this case, the circuit components after the filter are still vulnerable to damage.

Damage to components: Despite filtering, the attenuated pulse may still be sufficient to damage sensitive circuit components, especially if they are designed to operate at low voltage levels, such as 3.3V or 5V.

10) EMI/ESD event and impact on the device and power planes.
Energy of event **0.02–0.15 mJoule.**
As can be seen from the presented slides, the impact of the event is quite significant and can lead to failure of the processor or Wi-Fi module. (Scope dimensions, 1v/div 25mSec/div)



Short term solutions;
ESD suppressors on each line of solenoids wires towards to ground
ESD suppressors on VCC2 line of wi-fi module close to power pin.
ESD suppressors on VCC line of MCU close to power pins.
ESD suppressors at input AC filter.

Long term solutions;
Will include short term solutions applied on PCB and include all remarks above, related to schematic and layout of the device.

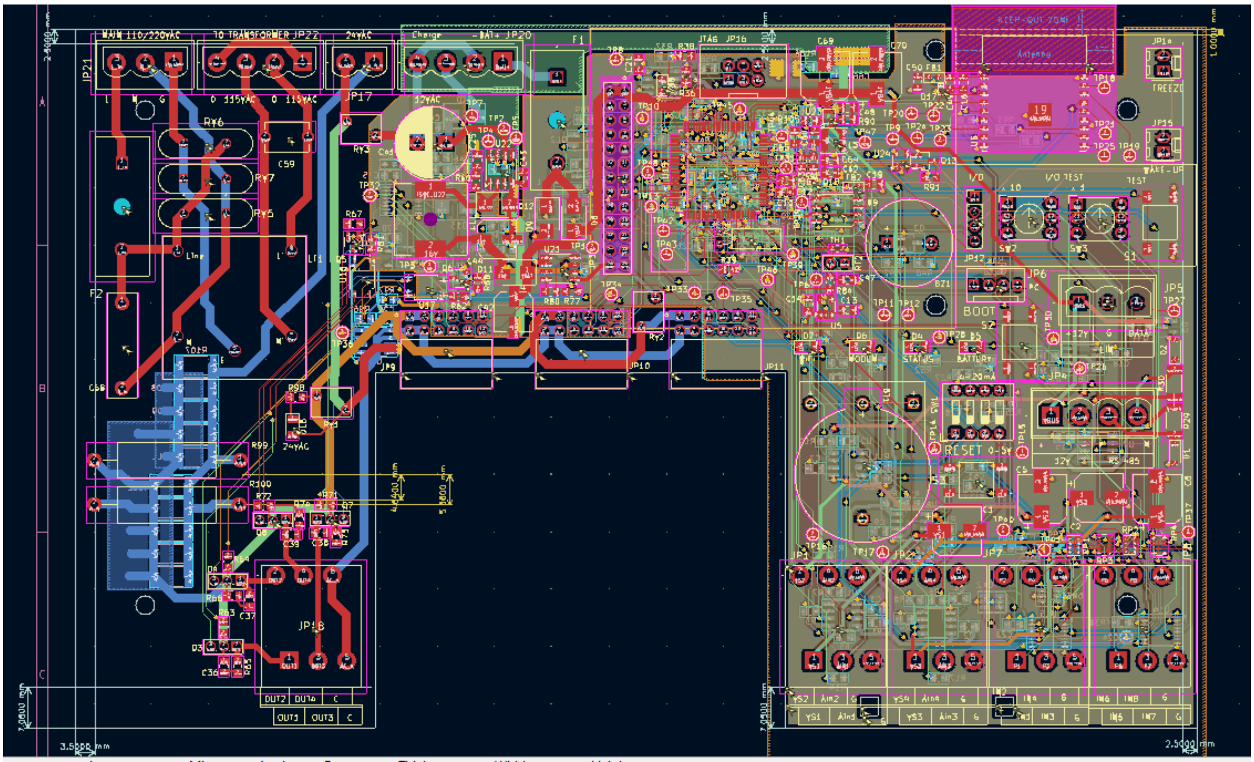
2024-10-25

Generally, we should avoid layout AC and DC together;
The routing of DC and AC circuits on the DC plane of a PCB is undesirable for several reasons, primarily related to electromagnetic interference, noise, and potential disruptions in circuit operation:

- **Inductive Interference**: Alternating currents generate changing magnetic fields, which can induce voltages in DC circuits. This can lead to noise and oscillations in the DC circuits, causing instability in the power supply and affecting the performance of sensitive components.
- **Capacitive Coupling**: AC lines can transmit high-frequency noise through capacitive coupling between conductors, especially if they are placed close to each other. This can induce interference in DC circuits, leading to voltage instability or disruptions in control circuits.
- **Electromagnetic Compatibility (EMC)**: Routing DC and AC together can degrade the electromagnetic compatibility of the system. AC lines can emit electromagnetic radiation, which can negatively affect other parts of the circuit, causing issues with meeting EMC requirements.
- **Different Routing Requirements**: DC and AC lines have different routing strategies. DC circuits typically require minimizing resistance and providing a stable path for current, while AC lines need to account for reactive components (inductance and capacitance) to minimize losses and interference.
- **Component Disruption**: High-frequency interference from AC lines can affect the operation of components, especially digital and analog circuits powered by DC. This may cause malfunctions, false triggers, or degraded system performance.

Solution

After the redesign and separation of the AC and DC lines, the addition of ESD suppressors on the MCU and Wi-Fi module power rails, and the optimization of the power and ground polygons, the problem was eliminated.



The redesign was carried out by the outstanding team of engineers at CircuitCopper.com.

2024-10-25