

EC 601 – Project 4
Seminar Overview

After graduation, I was working as a System Engineer in a startup where I was of taking a laser diode driver PCB all the way from design to production to further optimize the laser diode intensity of one of the products. This included schematic design, PCB layout, creation of Gerber files, sending the PCB for production and assessing the first shipment. However, while evaluating the first iteration of board I realized the board was not working at its optimum capacity and there was quite a bit of room for me to develop my layout skills.

Hence, for this project I decided to watch guest speaker seminar and other related papers to understand the techniques for EMC compliance.

Main Seminar: Expert Live Training (US) [LIVE] – Rick Hartley

Topic: How to achieve a Proper Grounding

Fundamentals of PCB Design

1. It is important to avoid creating vias – connect traces from one layer to another. I realized that in my board I had multiple vias places to connect my signal place to the ground plane.
2. Gerber files tell the board manufacturer how the copper should be laid out and how the board should be drilled. Drill files – which specify where to drill holes in the x-y place and how big those holes need to be – are a type of Gerber files.
3. Width of the traces should depend on the current flowing though them. The laser diode driver board I designed was providing current to the laser diode module. Hence there were multiple current values flowing though the board. I had not taken this point in consideration when creating the PCB layout.
4. The PCB's current carrying capacity is determined by cross-sectional area of trace and temperature rise. Furthermore, cross-sectional area of trace is directly proportional to the trace width and copper thickness.
5. On a PCB that supports several different inductors in different circuits, the inductors must be aligned at right angles to avoid coupling.

Common PCB Layout Problems

1. Different return currents interfering with each other if not properly routed.
2. Ground plane that is not ideal – need to consider parasitic inductance and resistance.
3. If ground plane is near something or has sharp bends in it, then need to consider parasitic capacitance.
4. As the area of the supply path and return path got bigger, the inductance goes up and the ability to do high frequency signals goes down.

EC 601 – Project 4
Seminar Overview

Power Planes

For sensitive measurement circuits, it is important that digital and analog circuits have separate power supplies. The digital and the analog power planes should be isolated. Otherwise, noise can be modulated backwards on the power supply from the digital signal. Not doing so can share the noise with the analog signal and disrupt it.

Whenever working with standalone ADC, Microcontrollers or RF/microwave receivers it is crucial to read the datasheet for each component to reduce interference.

Ground Planes

A PCB ground is where return current flows to complete a circuit. It is a reference point for zero volts. It is a flat metal layer on a PCB. It has two major uses. Firstly, it is needed to keep the parasitic inductance and resistance in check and reduce the overall impedance. Secondly, it acts as a shield to the EMI from the outside circuit.

Designs with high accuracy measurements, high-speed ADC measurements, analog audio signals, low level signals, RF / microwave receiver, high speed digital signals (example USB) and switch-based power supplies or DC-DC converters are more susceptible to the problems in the ground plane or interference caused by currents in the ground plane.

When working with high frequency digital signals, ground currents try to flow directly under the signal trace (lowest Z path). While for DC signals (that does not change state at all), current will try to take the shortest path of lowest impedance. The path of least impedance is the path of lowest inductance and the path of highest capacitance.

Types of Ground Plane

1. Single Ground Plane

Just use a single ground plane on its own layer in multilayer PCB layout. Layout the design so that the digital signal is in one section of the ground plane and analog signals in another section. Just because an IC has an AGND and DGND does not mean they have to be on physically separated ground planes.

2. Split Ground Plane

Has analog and digital planes separated with one or more small connection points. This method adds design complexity without much benefit over the Single Ground Plane method.

EC 601 – Project 4
Seminar Overview

3. Isolated Ground Plane

Ground planes are not electrically connected (isolated) and no traces cross between them. If signals or data need to be passed between the isolated circuits, optoisolators, optocouplers, transformers or mini fibre optic cables should be used.

This approach is typically used for measurement equipment like digital multimeters and oscilloscopes.

How to achieve proper grounding

The energy in a circuit is entirely in the EM fields. These fields are located in between the spaces between the traces and the planes – in the dielectric. The energy thus travels in the plastic and fibreglass material on the PCB.

These are the steps that can be taken to achieve proper grounding in a PCB:

1. No need to power planes in analog circuits
2. Have the forward and the return trace directly over one another in the board stack and keep the dielectric space small – make sure small field results in small inertia and thus a small inductance.
3. Position components by function/ family
 - a. Analog in one area and digital in another.
 - b. Devices operating at different voltages in their own area. Otherwise, it can lead to crosstalk problems.
 - c. Devices operating at different frequencies in their own area.
 - d. All ICs routing to connectors must be placed near their respective connector. This helps reduce common mode currents in circuits.
 - e. Keeping all routes confined to the stage or section to which they are assigned. This can be done by keeping all the digital traces, low level analog and RF/Microwave each in its own section.
 - f. If traces are isolated in their own sections, the need to split analog and digital ground is only necessary where the analog section is operating under 20KHz.
4. It is not advisable to route signals on a ground plane layer.
5. A zero-ohm resistor can be used to cross slower signals over other signal lines at lower frequencies.

This seminar made me understand important concepts regarding PCB routing and layouts. I am interested in working in this line of engineering in the future and would use these concepts as required in my future designs.

EC 601 – Project 4
Seminar Overview

References

1. Altium. (2019, November 11). *[LIVE] How to Achieve Proper Grounding - Rick Hartley - Expert Live Training (US)* [Video]. YouTube.
<https://www.youtube.com/watch?v=ySuUZEjARPY>
2. *Everything You Need to Know about Designing a PCB Layout*. (2021, November 16). [Video]. PCB Assembly, PCB Manufacturing, PCB Design - OURPCB.
<https://www.ourpcb.com/pcb-layout-the-ultimate-guide-best-for-beginners.html>
3. ForceTronics. (2018a, April 4). *Ground Considerations for PCB Layout of Mixed Signal Designs Part 1* [Video]. YouTube. <https://www.youtube.com/watch?v=t-reltrutMI>
4. ForceTronics. (2018b, April 9). *Ground Considerations for PCB Layout of Mixed Signal Designs Part 2* [Video]. YouTube. https://www.youtube.com/watch?v=Lag09ml_0zk
5. HACKADAY. (2018, September 28). *Inductance in PCB Layout: The Good, the Bad, and the Fugly* [Video]. YouTube. https://www.youtube.com/watch?v=OQm0aBw_ep8
6. Montrose, M., I. (2000). *Printed Circuit Board Design Techniques for EMC Compliance: A Handbook for Designers* (2nd ed.). Wiley-IEEE Press.
7. *StackPath*. (n.d.). Retrieved October 25, 2022, from
<https://www.electronicdesign.com/power-management/whitepaper/21248133/analog-devices-the-golden-rule-of-board-layout-for-smps>