ENEE4304 Instrumentation and Measurements

L7-2020

Measurement Noise and Signal Conditioning

Outline

- Measurement Noise and Signal Conditioning ,
- Sources of Measurement Noise and techniques for its reduction;
- Analog Signal Filtering and processing operations.
- Grounding Techniques.

Noise and Interference

- Noise, by definition, is the presence of an unwanted electrical signal in a circuit.
- Interference is the undesirable effect of noise.
- Where a noise voltage causes improper operation of a circuit, or its relative magnitude is of the same order as the desired electrical signal, then it is interference.
- Noise itself cannot be totally eliminated but only reduced in magnitude until it no longer causes interference.
- This is especially true in data acquisition systems where the analog signal levels from transducers measuring a physical quantity can be very small.
- Compounding this in many instances is the physical cable distance over which these signals must be transmitted and the effect that noise may have on this extended circuitry.

Noise and Interference

- Noise itself cannot be totally eliminated but only reduced in magnitude until it no longer causes interference.
- Noise is reduced by proper design practices and whatever noise that remain is treated by signal conditioning circuits
- Types of Noise :
  - Serial (differential type) which acts in series with useful output voltages of sensors and transducers and can cause significant errors
  - Common Mode noise which affects both lines in the same manner, it is dangerous since it can become differential in certain circumstances

Classification of Noise Sources

- External Sources such as motors, fluorescent lamps, monitors, mains cables, RF and audio-frequency circuits
- Internal Sources such as thermoelectric noise, shot noise and electrochemical action

Components of Noise induced Problem

Noise Source → Coupling Channel → Receiver Circuit

Noise Source	Coupling Channel	Receiver Circuit
➢ AC power Cables ➢ High voltage or high current circuits (motors etc) ➢ Switching power supplies ➢ Computer monitors ➢ Fluorescent lamps	➢ Common Impedance (Conductive coupling) ➢ Electric Field (Capacitive coupling) ➢ Magnetic Field (Inductive coupling)	➢ Transducer ➢ Transducer to signal conditioning cable ➢ Signal conditioning ➢ Signal conditioning to measurement system cable

Coupling Mechanisms

- The mechanisms for coupling noise most common to data acquisition and control applications are as follows:
  - Conductive coupling
  - Capacitive coupling
  - Inductive coupling
  - Other Coupling Mechanisms

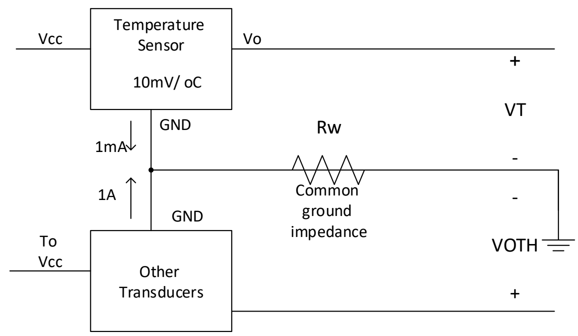
Conductive Coupling

- Conductive coupling occurs where two or more circuits share a common signal return.
- In such cases, return current from one circuit, flowing through the finite impedance of the common signal return, results in variations in the ground potential seen by the other circuits.

Conductive Coupling

- A series ground connection scheme resulting in conductive coupling is shown in Figure below.
- If the resistance of the common return lead is 0.1 Ω and the return current from all other circuits is 1 A, then the voltage measured from the temperature sensor, (V_T), would vary by 0.1 Ω × I A = 100 mV, corresponding to 10 degrees error in the temperature measured.

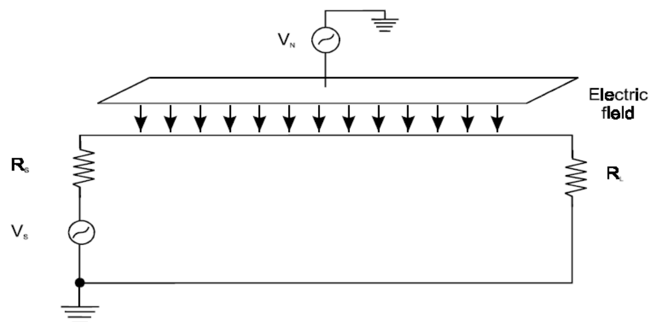
Series ground connections resulting in conductive coupling



Capacitive Coupling

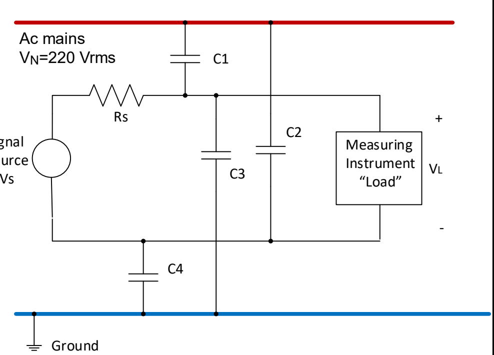
- Due to electric fields
- Electric fields occur in the vicinity of voltage-varying sources.
- Capacitive coupling ***is the transmission of external noise through mutual and stray capacitances between a noise source and receiving circuit.***
- This is sometimes referred to as electrostatic coupling, although this is a misnomer, since the electrical fields are not static.
- Since cables tend to be the longest circuit elements, capacitive coupling is best demonstrated by considering a signal circuit connecting a signal source to a measurement system by a pair of long signal-carrying conductors.

- The physical representation of electric field coupling between a noise source and such a signal circuit is shown in Figure below.



Physical representation of an electrical field coupling into a signal circuit

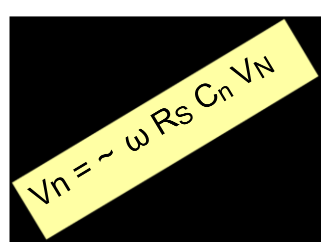
- Noise voltage $V_n = \omega R_S C_n V_N$
- where Cn is any of the parasitic/ stray capacitors C1,C2, C3, C4
- If we make C1=C2 and C3=C4 then the noise voltage caused by these caps will be equal and opposite in sign and cancel each other
- Any unbalance in capacitor values will result in net noise voltage added or subtracted from V_L
- Also Rs must be smaller than load resistance and stray capacitance impedance



$$V_n =\sim \omega R_S C_{12} V_N$$

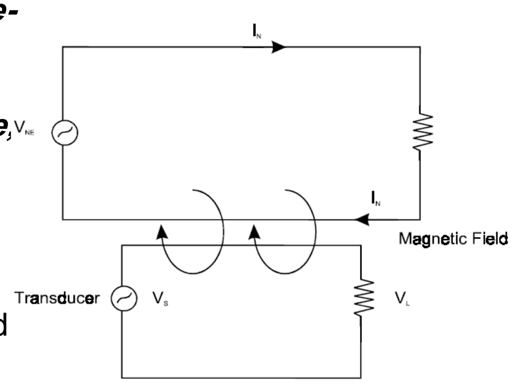
- ω , V_N - frequency and amplitude of the external noise source,
- Rs- the resistance to ground of the signal circuit
- C_{12}- and the mutual capacitance between them.
- If $R_S >> 1/ j\omega [C_{12} + C_{2G}]$), then it can be shown that the **capacitively-coupled noise voltage, is independent of the frequency of the noise source, and is much greater than in the case where the same resistance is relatively small**.

- The amplitude and the frequency of the noise source cannot be altered, the only means for reducing capacitive coupling into the signal circuit is to reduce the equivalent **signal circuit resistance to ground or reduce the mutual stray capacita**nce.



Magnetic field coupling “Inductive Coupling”

- Magnetic field coupling or inductive coupling ***is the mechanism by which time-varying magnetic fields produced by changing currents in a noise source, link with current loops of receiving circuits.***
- The physical representation of magnetic field coupling between a noise source and a signal circuit is shown in Figure



- Lenz’s law states that the voltage, Vn induced into a closed loop signal circuit of area A is proportional to the rate of change of the magnetic field coupling the circuit loop, the flux density (B) of the magnetic field and the area of the loop.
- This is represented by the formula:

$$V_n = 2 \pi f BA \cos\varphi \, (10^{-4})$$

$$Vn = 2 \pi f BA \cos\varphi (10^{-4})$$

- where
  - f = the frequency of the sinusoidal varying flux density
  - B = the rms value of the flux density (gauss)
  - A = the area of the signal circuit loop (m^2)
  - φ = the angle between the flux density (B) and the area (A).
- This equation indicates that the noise voltage can be reduced by reducing *B*, *A*, or c*os*φ.
- The flux density (*B*) can be reduced by increasing the distance from the source of the field or if the field is caused by currents flowing through nearby pairs of wires, twisting those wires to reduce the net magnetic field effect to zero and or by alternating its direction.

Inductive Coupling

- The signal circuit loop area (*A*) can be reduced by placing the signal wires of the receiving circuit current loop closer together.
- For example, consider a signal circuit whose current carrying wires are 1 meter long and 1 centimeter apart, lying within a 10 gauss 60 Hz magnetic field, typical of fans, power wiring and transformers.
- The maximum voltage induced in the wires occurs for φ = 0º.
- Vn = (2π x 60)(10)(1 × 10^–2)(10^–4) = 3.7 mV.
- If the distance between the wires is reduced to 1 mm the noise voltage is reduced ten fold to 0.37 mV.

Inductive Coupling

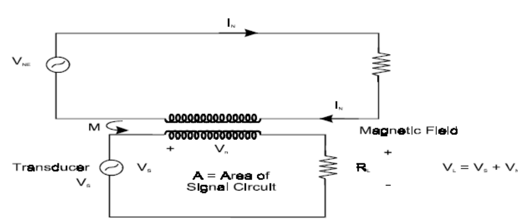
- The cosφ, term can be reduced by correctly orienting the wires of the signal circuit in the magnetic field.
- For example, if the signal wires were perpendicular to the magnetic field (φ = 90°) the induced voltage could be reduced to zero, although practically this would not be possible.
- Running the signal wires together in the same cable as the wires carrying the noise current source would maximize the induced noise voltage

---

- The equivalent circuit model of magnetic coupling between a noise source and a signal circuit is shown in Figure below.
- In terms of the mutual inductance (M), Vn is given by:

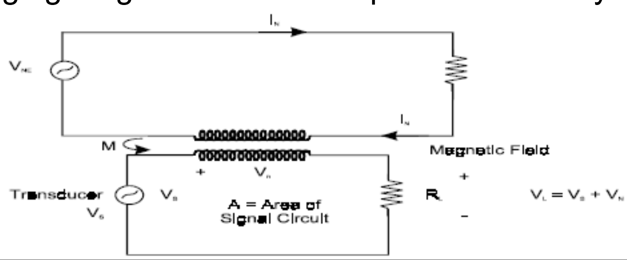
$$V_n = 2\pi f M I_N$$

- I_N - is the rms value of the sinusoidal current in the noise circuit and f is its frequency.
- The mutual inductance (M) is directly proportional to the area (A) of the signal circuit current loop and the flux density, (B).



Inductive Coupling

- The physical geometry of the current loop of the receiving signal circuit, specifically its area, is the key to why it is susceptible to magnetic fields and how to minimize the effect.
- Cables provide the longest and largest current loop.
- The effect of magnetic coupling is best demonstrated by considering the circuit of Figure below, in which the signal cable current loop is coupled by a sinusoidal changing magnetic field with a peak flux density of $B\varphi$.



- Ideally, the only voltage appearing across the load should be V_S – the source signal voltage.
- However, the magnetic flux induces a voltage in the loop that appears in series with the receiver signal circuit.
- The voltage appearing across the load is the sum of the source voltage and the unwanted magnetic field induced voltage (V_N).

Techniques for Reduction of Noise

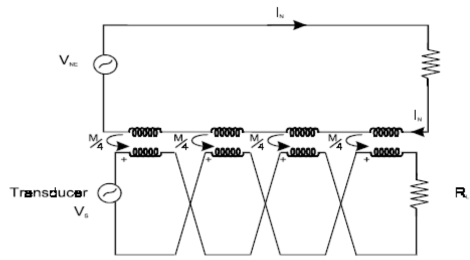
- 1. Location of signal wires:
  - Both mutual inductance and capacitance between signal wires are inversely proportional to the distance between them
  - It is recommended to place signal wires as far as possible from noise sources (at least 0.3 m)
- 2. Design of wires: Use twisted wires
- 3. Proper grounding
- 4.Shielding

Location of signal wires:

- Both mutual inductance and capacitance between signal wires are inversely proportional to the distance between them
- It is recommended to place signal wires as far as possible from noise sources (at least 0.3 m)
- The mutual stray capacitance can be reduced by
  - **increasing the relative distance of the signal wires from the noise source,**
  - **correct orientation of the conductors,**

Design of Wires: Use twisted wires

- Twisting the insulated conductors together, can greatly reduce the amount of magnetic coupling into the signal lines.



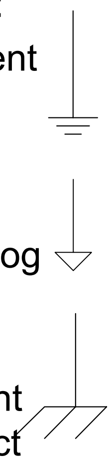
- The voltage induced in each section of the loop now alternates phases; its magnitude reduced by the reduction in area of each twisted loop (i.e. 1/4).
- Provided there is an even number of twists in the signal conductors, the voltages due to the magnetic field cancel out and only the desired signal voltage appears across the load.

Grounding Techniques

- The word ground has a historical origin that is , perhaps, the cause of the different meanings in use today.
- Originally , it referred to a point that was actually connected to earth in order to obtain zero potential
- In electronic systems, the ground point is the reference potential
- The confusion between earth and ground can be avoided if we consider that the electrical system on aircraft has aground point for voltage reference , a point that is not connected to earth
- We will use earth for connection to the earth and ground as a central reference connection

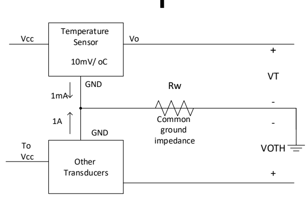
Grounding Techniques

- Usually we use the following Ground symbols:
- Power Ground PGND : provides path for current faults
- Digital Ground- DGND: common for all digital and logic circuits
- Analog Ground –AGND : common for all analog signals
- Safety Ground = Chassis Ground (Earth) : connected to all metallic parts of the equipment to protect people if power lines come in contact with metal enclosure



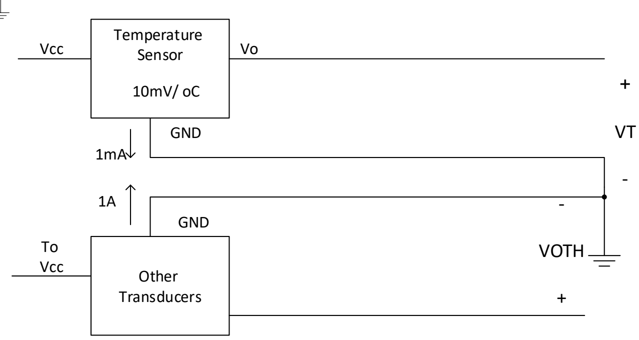
Solution to previous problem

- Don't allow the 1A current affect the output temperature sensor V_T



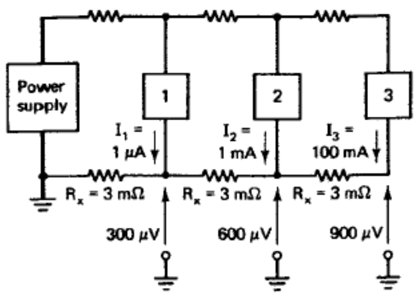
Wrong

Correct



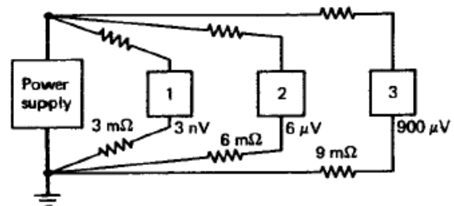
Ground Star Connection

Parallel distribution of power



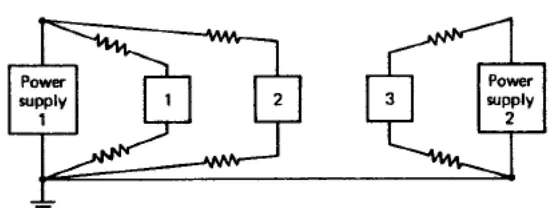
3 mohm is the resistance of 15 cm of AWG#18 wire

Radial (Star) distribution of power



32

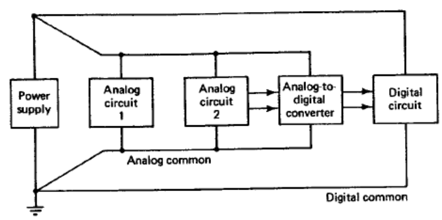
Two Power supplies can be used such that long wires are avoided for circuit 3



- **If the voltage drop on power supply path does not affect the operation of the circuits, a combination of parallel and radial distribution could be used.**
- **The star connection will then be used for ground wire**

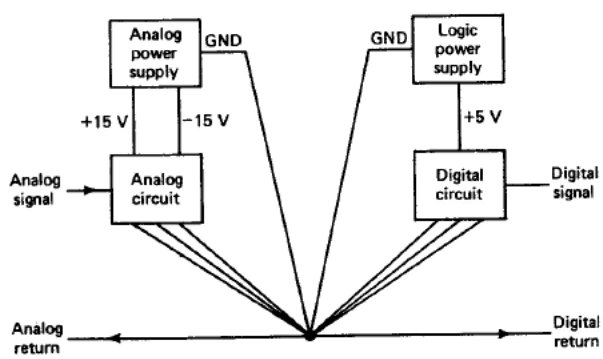
Grounding of Analog-Digital Circuits

- Consider the case when analog and digital circuits are used together
- Digital signals, create large current spikes in the ground paths due to switching
- These currents can cause much interference in analog circuits
- Even if they both share the same power supply, their ground wires must be different with only one common point as shown in the figure , this minimizes common impedances between digital and analog circuit



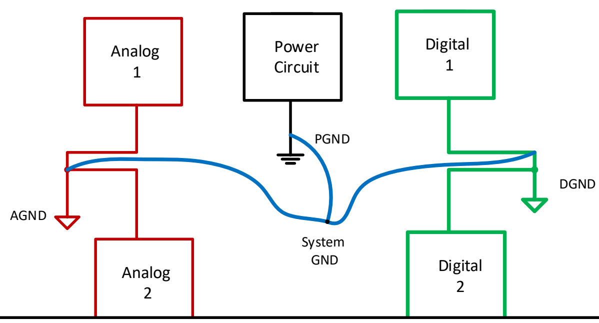
Analog-Digital Circuits

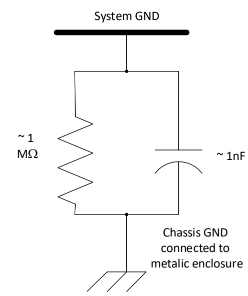
- When digital and analog circuits are powered by separate sources, each circuit must be connected to ground of its power supply.
- Then both grounds are connected to a single point ground that is called "star GND" or "system GND"



In schematic Capture (Board physical layout)

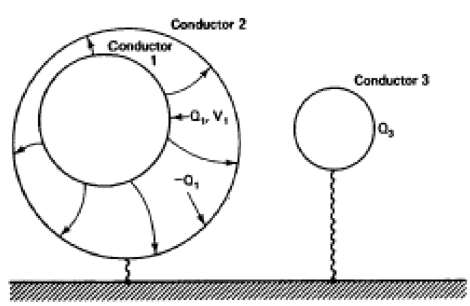
- Use Multiple grounds (with different symbols) according to type of circuit and then connect to the star point or system ground
- Connection of system ground to chassis (safety) ground can be done through a filter since system ground can act as a huge antenna that picks extra noise if connected to chassis directly, so a filter might be used





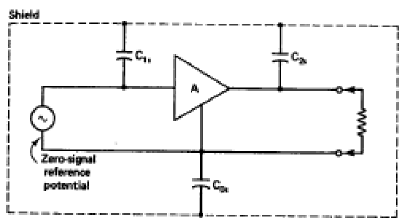
Electrical Shielding

- ***Shielding: adding a complete metallic enclosure or screen to remove capacitive coupling.***
- Conductor 2 is the shield to prevent mutual capacitance between 1 and 3

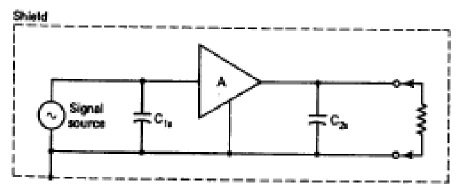


Rules of Shielding

- **Rule 1:** Connect the shield to low potential such as ground or earth , if not stray caps can appear as feedback caps affecting operation



Shield Floating
C1s and C2s appear as Feedback Caps that can affect frequency response of amplifier



Shield Grounded
C1s and C2s appear as input and output caps

- **Rule 2:** Shield conductor should be connected to zero-signal reference potential at the point the signal is earthed (point 1).
- Fig a shows an earthed signal source and **an incorrect connection between the shield and the signal reference lead at some other point (3)**. There is no connection between the shield and zero reference at point 1
- Ground 1 and 2 usually at different potentials, or in other words, there is a ground voltage V12

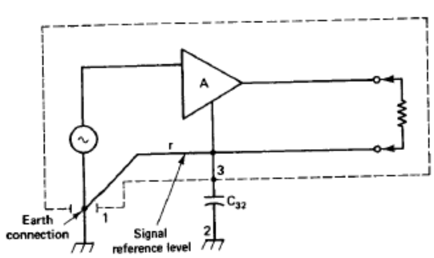


Fig a

- Fig b shows equivalent circuit of this case,
- The current due to V12 circulates through r, the reference signal wire, creating interference
- However if 1 and 3 , besides being at the same potential, coincide in a single physical point, no current passes through r, and interference is eliminated (Fig c)

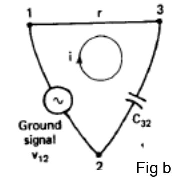


Fig b

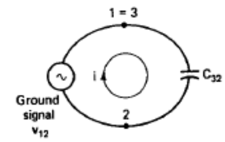


Fig c

Note that from electrical point of view Fig b and Fig c represent exactly the same circuit. However physical paths are different for the current

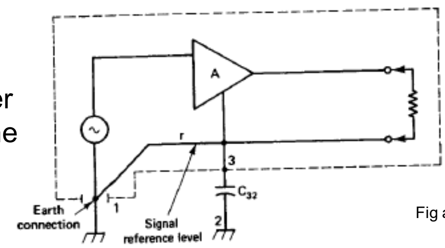
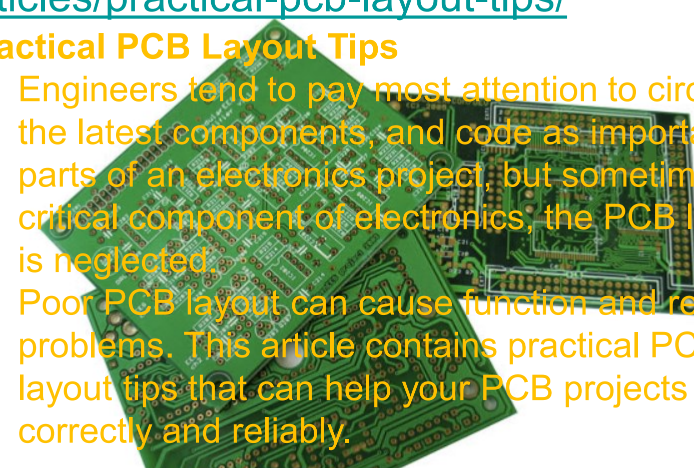


Fig a

Practical PCB Layout Tips Every Designer Needs to Know

https://www.allaboutcircuits.com/technical-articles/practical-pcb-layout-tips/



Practical PCB Layout Tips

- Engineers tend to pay most attention to circuits, the latest components, and code as important parts of an electronics project, but sometimes a critical component of electronics, the PCB layout, is neglected.
- Poor PCB layout can cause function and reliability problems. This article contains practical PCB layout tips that can help your PCB projects work correctly and reliably.

- **Sizing Traces**
- Real-world copper traces have resistance. This means that a trace has a voltage drop, power dissipation, and a temperature rise when current flows through it. Resistance is defined by this formula: $R = \frac{(resistivity * length)}{(thickness * width)}$
- PCB designers can't really change the physical properties of copper, so focus on the trace size, which you can control.
- PCB trace thickness is measured in ounces of copper. One ounce of copper is the thickness we would measure if we evenly spread 1 oz of copper over a 1 square foot area. This thickness is 1.4 thousandths of an inch. Many PCB designers use 1 oz or 2 oz copper, but many PCB manufacturers can provide 6 oz thickness. Note that fine features like pins that are close together are hard to make in thick copper. Consult your PCB manufacturer about what their capabilities are.

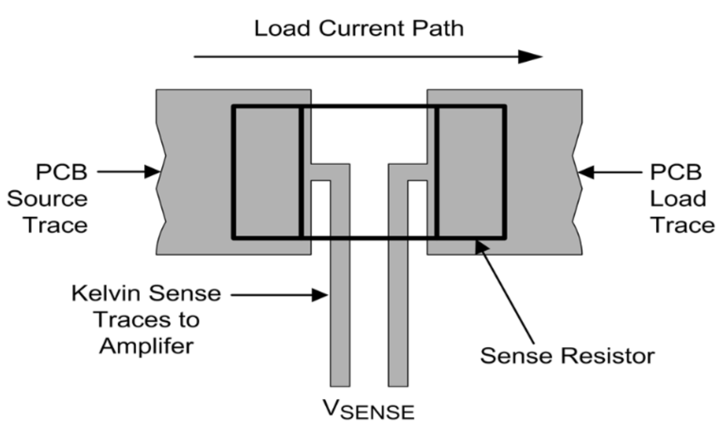
- Use a PCB trace width calculator to determine how thick and wide your traces should be for your application. Aim for a 5°C temperature rise. If you have extra space on the board, use bigger traces, as they don’t cost anything.
- When doing a multi-layer board, remember that traces on external layers have better cooling than traces on internal layers because the heat from inner layers has to travel through layers of copper and PCB material before being conducted, radiated, or connected away

Make Loops Small

- Loops, especially high frequency loops, should be made as small as possible. Small loops have lower inductance and resistance. Placing loops over a ground plane further reduces inductance. Having small loops reduces high frequency voltage spikes caused by : $V = L\frac{di}{dt}$.
- Small loops also help reduce the amount of signals that are inductively coupled into the node from external sources, or are broadcast from the node. This is what you want, unless you're designing an antenna. Also keep loops small for op-amp circuits to prevent noise from being coupled into the circuit.

Kelvin Connections

Kelvin connections are useful for measurements. Kelvin connections are made at the exact points to reduce stray resistance and inductance. For example, Kelvin connections for a current sense resistor are placed exactly at the resistor pads, not at some arbitrary place on the traces. Although on the schematic, placing the connections at the resistor pads or at some arbitrary point may look the same, real traces have inductance and resistance that could throw your measurements off if you don't use Kelvin connections.

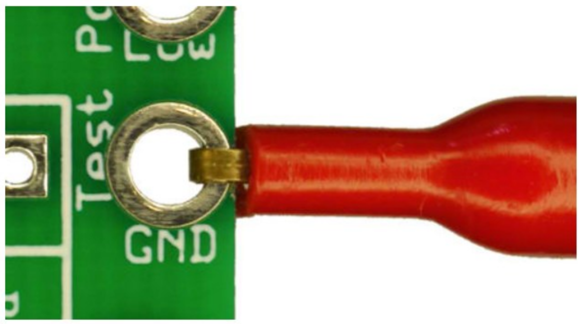


Keep Digital and Noisy Traces Away from Analog Traces

Parallel traces or conductors form a capacitor. Placing traces close together capacitively couples the signals on the traces, especially if the signals are high frequency. Keep high frequency and noisy traces away from traces that you don't want noise on.

Ground is Not Ground

Ground is not an ideal conductor. Take care to route noisy grounds away from signals that need to be quiet. Make ground traces large enough to carry the currents that will flow. Placing a ground plane directly under signal traces lowers the impedance of the traces, which is ideal.

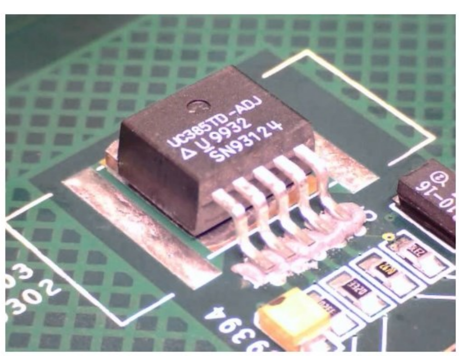


Via Size and Number

Vias have inductance and resistance. If you're routing a trace from one side of the PCB to the other and need low inductance or resistance, use multiple vias. Large vias have lower resistance. This is especially useful in grounding filter capacitors and high current nodes. Use a via size calculator like this one.

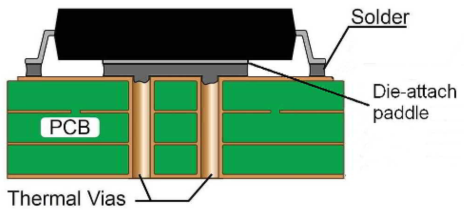
Using PCB as Heatsink

Place extra copper around surface mount component to provide extra surface area to dissipate heat more efficiently. Some component datasheets(especially power diodes and power MOSFETs or voltage regulators) have guidelines for using PCB surface area as heatsinks.



Thermal Vias

Vias can be used to move heat from one side of a PCB to the other. This is especially useful when a PCB is mounted on a heatsink on a chassis that can further dissipate heat. Large vias transfer heat more efficiently than small vias. Many vias transfer heat more efficiently than one via, and lower the operating temperature of components. Lower operating temperatures contribute to higher reliability.



Thermal Relief

Thermal relief is making connections between a trace or a fill and a component pin small to make soldering easier. This small connection is short to reduce the impact on electrical resistance. If thermal relief on component pins is not used, then the component may be a little cooler because there's a better thermal connection to traces or fills that can dissipate heat, but it'll be harder to solder and desolder.

Distance between Traces and Mounting Holes

Leave room between copper traces or fills and mounting holes; this helps prevent shock hazards. Solder mask isn't considered a reliable insulator, so take care that there is distance between copper and any mounting hardware.

Heat Sensitive Components

Keep components that are sensitive to heat away from other components that generate heat. Examples of components that are sensitive to heat include thermocouples and electrolytic capacitors. Placing thermocouples close to heat sources may throw off temperature measurements. Placing electrolytic capacitors close to heat generating components will reduce their operating life. Components that generate heat may include bridge rectifiers, diodes, MOSFETs, inductors, and resistors. The heat depends on current flowing through the components.

Conclusion

This article has covered some basic practical PCB layout tips that can positively impact the functionality and reliability of your design. Have more tips and tricks? Leave them in the comments!