Capacitive Sensor Layout Recommendation Manual

Revision: A

Publication: August 22, 2023

This application note is a design and layout recommendation for button, slider, and proximity sensor applications with Lumissil CS89XX/SE51XX series MCU. Liquid-tolerance function via shield electrode is to prevent sensors mis-trigger due to touch surface with water, mist, ice, or other liquids. This application note is for the generic design and layout recommendation. Different applications might need different considerations of design and layout. For applications in which liquid-tolerance is an important requirement, copper hatch around the sensor needs to be connected to shield pin of MCU to have a good performance to prevent touch keys mis-triggered. If liquid-tolerance is not a requirement, then copper hatch around the sensor needs to be connected to ground to reduce noise and acquire a better SNR. Sensor environment will play an important role in deciding what kind of design and layout scenarios.

1. Button Sensor Design

C_P: Parasitic capacitance

This parasitic capacitance is an equivalent capacitance which includes the capacitive effects of the sensor pad, the overlay, the trace between the MCU sensor pin and the sensor pad, and the pin capacitance of the MCU. C_P is related to the electric field around the sensor pad.

C_F: When a finger touches the sensor pad through the overlay, it will form a parallel plate capacitor which we call finger capacitor C_F.

C_F is the equivalent capacitance of the finger, effect of the human body and the return path to DC circuit board ground. A large C_F is achieved by a large overlay dielectric ε_r, a large sensor area A and a thin overlay thickness D.

$$C_F = \varepsilon_0 \varepsilon_r A / D$$

Where:

ε_0 = Free space permittivity

ε_r = Dielectric constant of overlay

A = Area of the practiced parallel area formed by finger and touch pad (The area is decided by human finger contact size 8mm.)

D = Overlay thickness

The guideline for capacitive sensor layout is to have a higher noise immunity, a lower Cp, and signal-to-noise ratio (SNR > 5:1) based on PCB design and sensor board arrangements. A high Cp needs to be avoided because of the drive limits of the MCU internal current sources and a longer response time for application.

There is no general relation between Cp and PCB layout. An increase in sensor size, an increase in layout trace length and width, and a decrease in the annular gap (a copper free area between the sensor edge and the adjacent PCB) all cause an increase in C_P. One way to reduce C_P is to increase the air gap between the sensor and ground. Unfortunately, widening the gap between sensor and ground will decrease noise immunity.

Any button shape like round or rectangular with sufficient size will work, however buttons with 90 deg sharp corner should be prevented to avoid possible ESD discharging path to MCU sensor pins. Round corner is recommended.

1.1 PCB Board Layer

For two-layer PCB, sensors are suggested to be on the top layer while MCU and other components are on the bottom layer. For a complex circuit with limited PCB size, four-layer PCB is recommended.

1.2 PCB Board Size

FR4-based PCB designs perform well with board thicknesses ranging from 0.5 mm to 1.6 mm. Flexible PCB has no problem for sensor application and is specially recommended for curve surface. For Flexible PCB, we suggest having PCB thickness of at least 0.25mm.

1.3 Self-Capacitance Button Design

a) The button diameter is suggested to be in the range of 5 mm to 15 mm, with 10 mm being suitable for most applications.

b) For larger button diameters, overlay thickness should be larger.

c) The annual gap size should be equal to the overlay thickness. The annual gap can be between 0.5 mm ~ 2 mm.

d) The spacing between the two adjacent buttons should be large enough that a finger should not reach the annular gap of the adjacent button when one button is pressed. The average human finger size is around 8mm. It is recommended that the spacing between the adjacent sensors is 4 mm + touch overlay thickness. If there is a multiple of sensors within limited PCB area, tradeoff might need to be considered for the spacing between two adjacent sensors.

1.4 Sensor and Components Placements

a) 2 – layer PCB: Top layer is for sensor. Bottom layer is for components and sensor signal.

b) 4 – layer PCB:

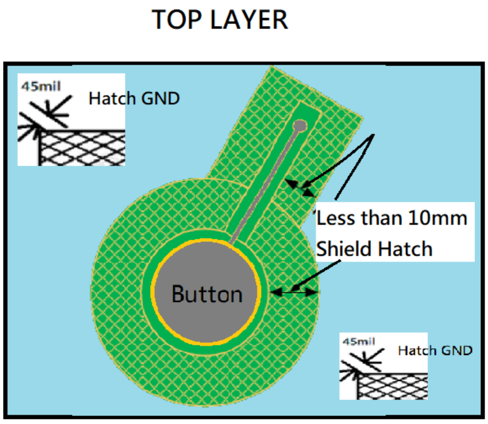
Top layer is for sensor

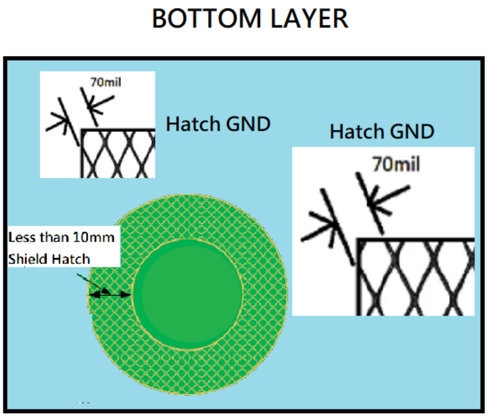
2nd layer is for sensor trace routing

3rd layer is for hatch fill of 7 mil line and 70 mil spacing and connect it to ground

Bottom layer is for components and the unused area can be filled with copper fill of 7 mil line and 70 mil spacing to Ground.

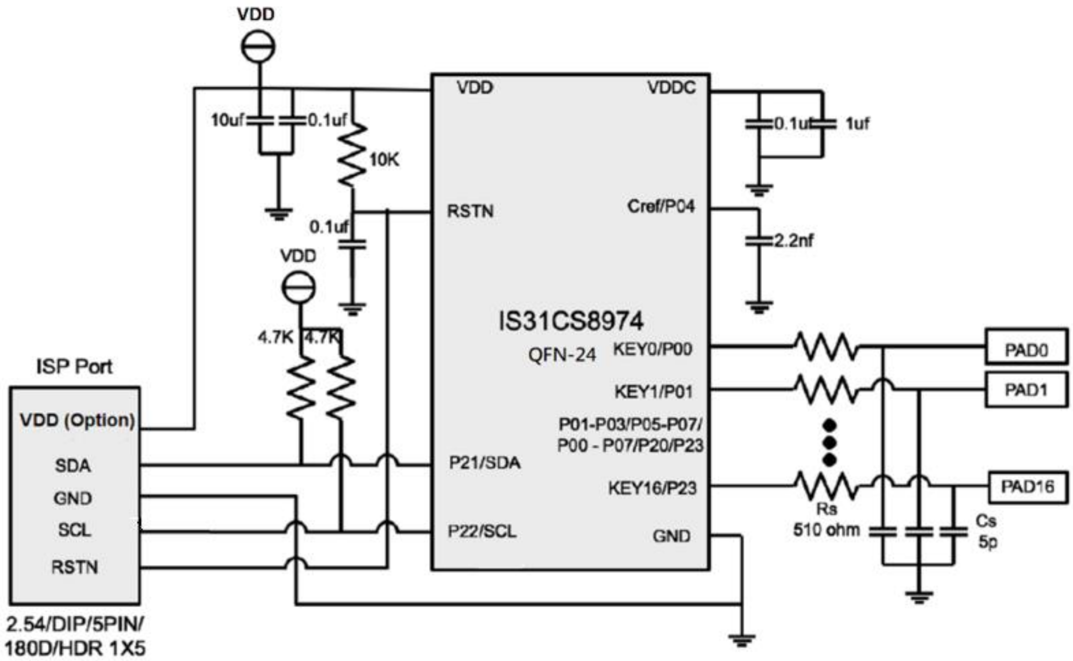
c) If liquid-resistant is necessary, please use a less than 10mm width shield electrode to cover sensor button and connect MCU shield pin with shield electrode. Suggest keeping a 3mm gap between hatch ground and shield hatch (copper hatch for shield) with two 0-ohm 1206 resistances between hatch ground and shield hatch as a reserved tuning option. Please refer to the below button sensor layout recommendation figure.





d) Minimize the trace length from MCU pins to sensor pad. Traces connected to sensor pads can be at the edge of the pad, not the center of the pad to have a shorter trace length.

e) For items e ~ i, please refer to the related chip datasheet and the figure below.



f) Suggest adding ISP and Writer mode interface for firmware and Boot code update.

ISP: Pin 21 (P21) as SDA, Pin 22 (P22) as SCL

Writer mode: Pin 19 (P17) as CEB/CS, Pin 20 (P20) as SCK, Pin 21 (P21) as SDI, Pin 22 (P22) as SDO, Pin 24 (RSTN)

g) Place all decoupling capacitors close to the VDD, VDDC.

h) Keep all sensor series resistors (Rs) and parallel capacitors (Cs) within 10 mm of the MCU pins to reduce RF interference and provide ESD protection.

i) Keep Cref capacitor 2.2nF close to MCU pin.

j) Isolate switching signals, such as PWM, communication lines, and LEDs from the sensor and the sensor traces by placing them at least 4 mm apart and fill a hatched ground between sensor traces and non-sensor traces to avoid crosstalk.

k) Avoid connectors between the sensor and the MCU pins because connectors increase C_P and reduce noise immunity.

l) Minimize the via numbers for trace between the sensor and the MCU pins.

m) Place a guard ring on the perimeter of the PCB board and connect the guard ring to chassis ground to prevent possible discharge event.

1.5 Trace Length and Width

Sensor trace capacitance is minimized when they are short and narrow.

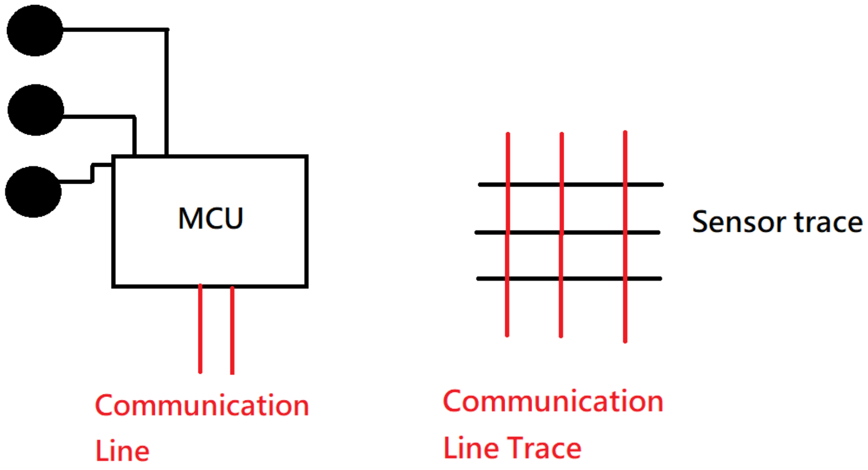
The maximum recommended **trace length** is 12 inches for a standard PCB and 2 inches for flexible PCB.

Trace width should not be greater than 7 mil.

Sensor traces should be surrounded by hatched ground with trace-to-ground air gap of 10 ~ 20 mil.

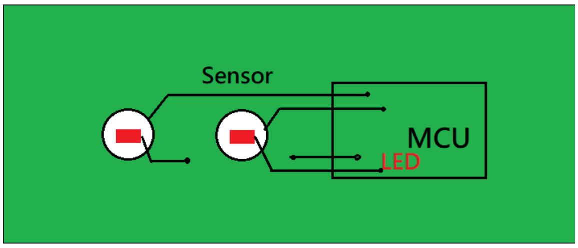
1.6 Sensor Trace Routing

Keep capacitive sensing traces away from communication lines, such as I2C or SPI masters. If it is not avoidable to cross communication lines with sensor pins traces, make sure the intersection is at right angles as below figure.

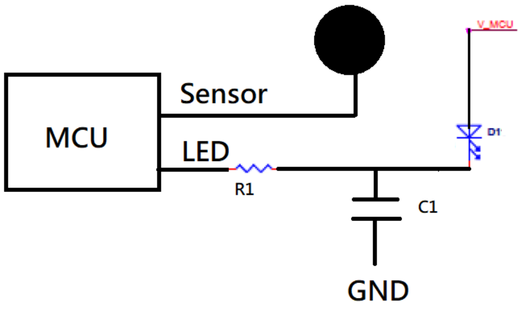


1.7 LED Crosstalk Solution

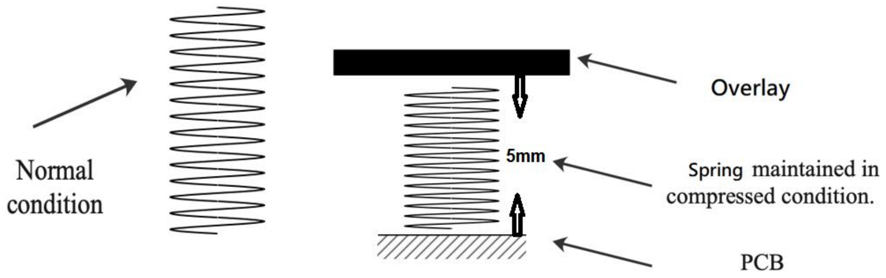
For the button sensor with LED application, LED is put under the sensor pad, and it shines through a hole in the middle of the sensor. When the LED is switched on or off, the voltage transitions on the LED driving trace can couple into the capacitive sensor input and generate noise. Keep a minimum separation of 4 mm between traces of sensor signal and LED signal. A hatched ground plane can also be added between both traces for isolation. LED drive traces and sensor traces should not be routed together.



Another way to reduce the crosstalk problem is to add one parallel 0.1uF capacitor to slow the rising and failing edge of LED driving voltage of the LED as the figure below.



1.8 Spring Button



The introduced capacitor C_F from the finger will increase with the increase of spring height, spring diameter and spring wire thickness but will decrease with the increase of overlay thickness.

a) Springs have higher side sensitivity and the adjacent spring sensors must be placed as far as possible to prevent false detections.

b) The distance between the PCB and the overlay must be 5 mm or more.

c) With thick overlay, the spring diameter must be larger than the overlay thickness by at least 2 or 3 times.

d) Provide the ground hatch or shield hatch near the footprint of the spring with an air gap of 1 mm between the sensor ring and the ground/shield hatch.

e) Minimize the PCB contact area of the spring pad to reduce the parasitic capacitance as below figure 1

f) Maximum the contact area of the spring and overlay as possible as figure 2 to increase the SNR (Signal-to-Noise ratio). Small surface area might result in low sensitivity and low SNR.

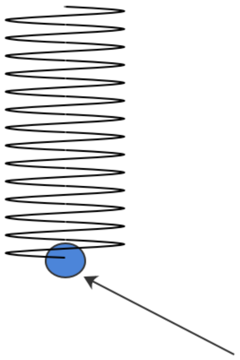


Fig. 1

Minimize the PCB contact area of the spring pad

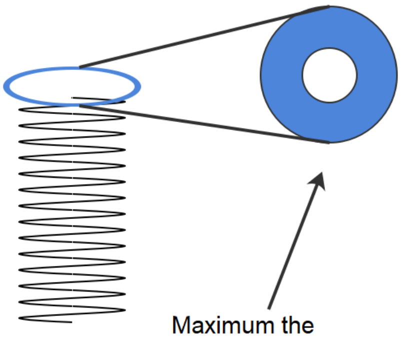


Fig. 2

Maximum the contact area of the spring and overlay

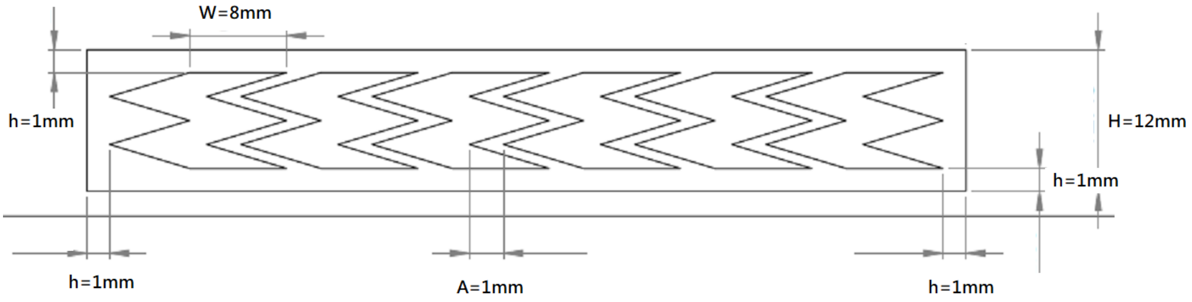
2. Slider Sensor Design

Slider sensor design uses similar philosophy as Section 1. Button sensor design. Users can refer to Section1 for more design and layout recommendations. Below are main design and layout recommendations for slider application.

1. Slider mechanical design recommendation

Why ZigZag pattern (double chevron) design is suggested? When the finger approaching a certain slider segment, the adjacent slider segment will also be detected. Software will calculate the geometric center of the finger touch (centroid position) and there will be more centroid positions than the number of slider segments. So, less GPIO pins should be possible to meet the product requirements. If the slider mechanical design recommendations as below slider mechanical design figure and parameters table are followed, there will be more dynamic finger positions calculated when your finger is cross the whole slider.

Suggest two (if possible) shield pins from MCU for most left and most right slider segments.



Parameter	Minimum	Maximum	Recommendation
Width of Segment (W)	7mm	15mm	8mm
Height of segment (H)	8mm	15mm	12mm
Gap between Segments (A)	0.5mm	2mm	0.5mm
Gap between Hatch and slider (h)	0.5mm	2mm	Equal to overlay thickness

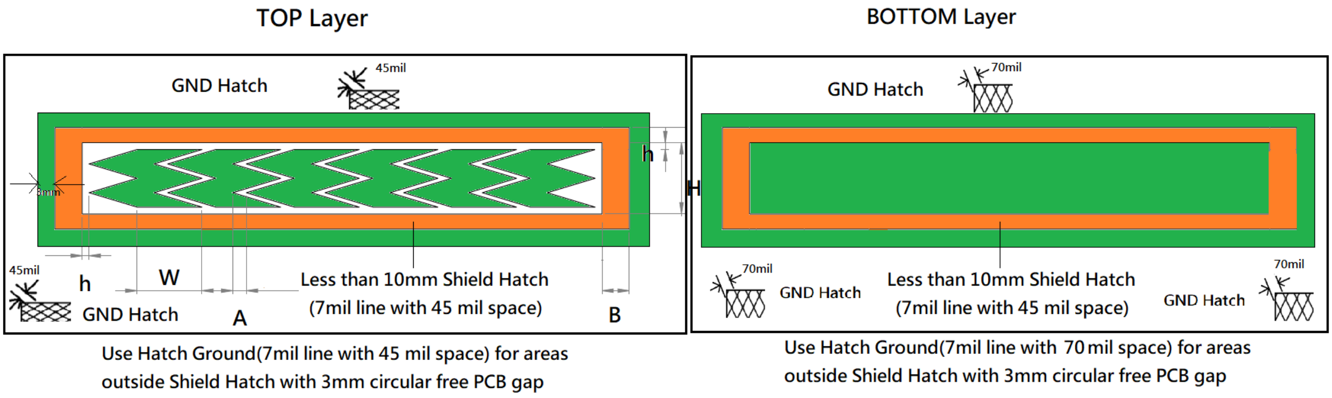
2. Slider board layout recommendation. Please refer to the below Slider board layout recommendation figure.

a) Suggested hatch ground: 25 percent on the top (sensor) layer (7 mil line, 45 mil spacing) and 17 percent on the bottom (MCU) layer (7 mil line, 70 mil spacing).

b) For top and bottom layers, suggest using copper hatch (7 mil line, 45 mil spacing) as shield hatch around slider with h distance gap (equal to overlay thickness) to slider. The width B of the shield hatch as below figure should be less than 10mm and we recommend using 8mm width.

c) For top and bottom layers, suggest build hatch ground outside shield hatch (in orange color) with a 3mm gap between hatch ground and shield hatch. Using two 0-ohm 1206 resistances (left and right sides of slider) to connect hatch ground and shield hatch as a reserved tuning option.

d) Leave the back side of slider placement without trace, power, or ground.

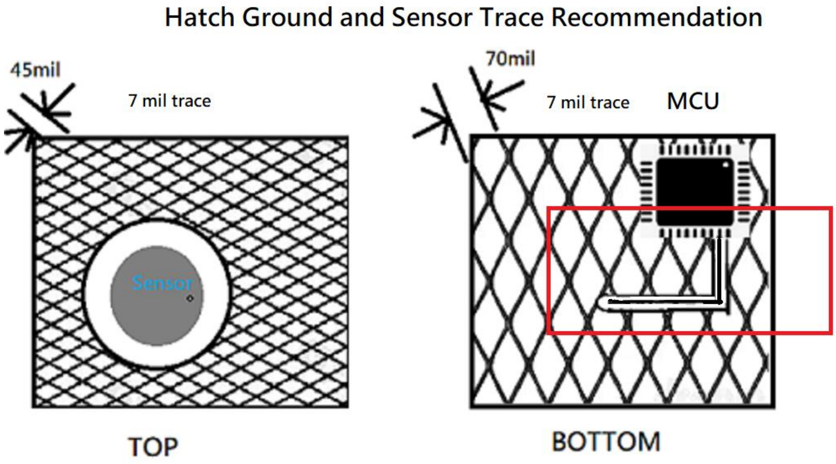


Slider board layout recommendation

3. Shield signal trace from MCU needs to be covered by shield hatch.

4. Traces of sensor to MCU pins are protected with hatch ground or hatch shield. Please refer to the below figure (use button sensor as an example).

Hatch Ground and Sensor Trace Recommendation



5. Suggest adding ISP and (Fast) Writer mode interface for firmware and Boot code update.

6. Place all decoupling capacitors close to the VDD, VDDC.

7. Keep all sensor series resistors (Rs) and parallel capacitors (Cs) within 10 mm of the MCU pins to reduce RF interference and provide ESD protection.

8. Keep Cref capacitor 2.2nF close to MCU pin

9. Isolate switching signals, such as PWM, communication lines, and LEDs, from the sensor and the sensor traces by placing them at least 4 mm apart and fill a hatched ground between sensor traces and non-sensor traces to avoid crosstalk.

10. Avoid connectors between the sensor and the MCU pins because connectors increase C_P and reduce noise immunity.

11. Minimize the via number for trace between the sensor and the MCU pins.

12. Place a guard ring on the perimeter of the PCB board and connect the guard ring to chassis ground to prevent possible discharge event.

3. Proximity Sensing

Proximity sensing is the process of detecting an approaching object without any physical contact. With proximity, a device can be woken up when the expected object approaching the sensor and accelerate its response time from low power mode and possibly light up the LED for users to press keys easily in a dark environment. Proximity sensor uses an electromagnetic field, beam of electromagnetic radiation, or changes in ambient conditions to detect the proximity of a nearby object. Capacitive proximity sensing has gained a strong popularity due to its advantages of low cost, high reliability, low power, sleek aesthetics, and seamless integration with existing user interfaces.

Below are four possible implementations for proximity function. Users can choose the suitable implementation.

Sensor: limited proximity distance based on general button diameter 5mm~15mm

Sensor ganging: Connect multiple sensors together to MCU Sense circuitry and MCU scans them as a single proximity sensor. Prevent the total parasitic capacitance Cp exceeding 45pF which is the drive limit of MCU's internal current sources. Sensor ganging can generate a longer proximity distance compared with one button sensor.

PCB trace: Use a straight-line PCB trace or a rectangle loop PCB trace. Using PCB trace as a sensor has a less parasitic capacitance Cp and yields out a longer proximity distance compared with other sensor types. PCB trace solution is easy for mass production. Suggest using PCB trace sensor with 1.5mm width.

Wire: A single wire can generate a long proximity distance, but it has a higher cost and a more complex mass production process compared with the PCB trace sensor.

3.1 Factors of proximity sensing distance

3.1.1 Hardware parameters affecting proximity sensing distance

Sensor type, size of sensor, parasitic capacitance Cp of the sensor, overlay material and thickness, and nearby floating or grounded conductive objects.

Sensor type: Proximity distance decides what kind of sensor type needed to be adopted. PCB trace can meet a long proximity distance inquiry and is easy for mass production.

Size of sensor: There is not a simple relationship between the sensor size and proximity sensing distance. Proximity sensing distance is highly related to the application environment. Users can use copper tape to form a rectangle loop or a circular loop as the sensor. The diagonal of a rectangle or the diameter of a circular loop is based on the required proximity sensing distance. If you can't achieve the required proximity sensing distance, you can increase the sensor loop diagonal or diameter until the required proximity sensing distance is met.

Parasitic capacitance Cp of the sensor: The proximity sensing distance depends on the ratio of the finger capacitance C_F to the C_P. The proximity sensing distance increases with an increase in the C_F/C_P ratio. For a given sensor size, the value of C_F depends on the distance between the sensor and the target object. To maximize this ratio, you need to increase C_F and decrease C_P. The C_P of the sensor can be minimized by selecting an optimum sensor area, reducing the sensor trace length, and minimizing the coupling of sensor electric field lines to the ground.

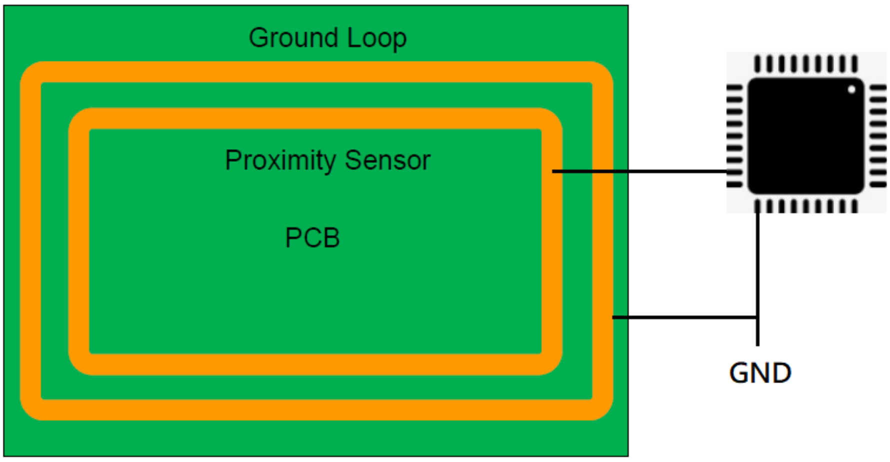
Driving the hatch fill in the top and bottom layer of the PCB with the MCU shield signal can reduce coupling of sensor electric field lines to the ground. The shield signal is a replica of the sensor signal with equal amplitude, frequency, and phase.

Nearby floating or grounded conductive objects: The proximity sensing distance reduces drastically if there is any nearby floating or grounded conductive object in the path of the approaching object to the sensor. When conductive objects are placed close to the proximity sensor, parasitic capacitance C_P will increase drastically. Larger sensor C_P requires slowing the sensor switching frequency, causing the proximity sensing distance to decrease. A grounded conductive object catches some of the sensor electric field and reduces the capacitance added by the target. Users can remove the conductive object or use the shield electrode to isolate the conductive object.

However, using a shield hatch or floating conductive object under the PCB back side of sensors will be very helpful on upper side proximity distance.

Reduce noise: Noise reduces the SNR and the proximity-sensing distance. To attenuate the noise in the sensor output and improve the ESD performance, surround the proximity sensor with a ground loop, as below figure shows. However, keep in mind that a ground loop around the sensor reduces the proximity-sensing distance. The minimum recommended width for the ground loop is 1.5 mm, and the minimum recommended clearance between the proximity sensor and the ground loop is 1 mm. a larger clearance width can increase proximity distance.

Ground Loop Surrounding the Proximity Sensor



3.1.2 Software parameters affecting proximity distance

Resolution of sensor, software noise filter

Resolution of sensor: With a high-resolution value, you can detect small changes in C_F with SNR > 5:1 and get a long proximity sensing distance.

Software noise filter: Proximity sensors are more susceptible to noise because of their large sensor area and high sensitivity settings. High noise decreases SNR and hence reduces the proximity-sensing distance. Users can use IIR, median, average or ALP filters to reduce the noise. ALP filters might reduce the sensor area by half at a given proximity sensing distance.

3.2 System consideration

Battery power system, Power consumption, Response time, and EMI/EMC/ESD performance

Battery power system: Connect system ground with earth ground to acquire the better sensing sensitivity.

Power consumption and Response time: A large proximity sensing distance needs more power. Large proximity sensing distance needs scanning the sensor at a high resolution. A higher resolution spells out a longer scan time and increases the device active time, which leads to a higher power consumption and a longer response time.

EMI/EMC/ESD performance: There is a tradeoff between the proximity-sensing distance and the EMI/EMC performance. A longer proximity-sensing distance requires the sensor to be tuned for high-sensitivity and reduces the EMI/EMC performance.

4. Revisions

Revision	Detailed Information	Date
A	First Release	2023.08.22