Self-Capacitance Touch PCB Design Guide

1. Applied for

The MCU with self-capacitance touch function.

2. Design instructions

The PCB Layout of capacitive touch products has a great influence on the touch sensitivity, so some suggestions on the PCB Layout of touch MCU can be referred to.

3. Basic principles of PCB design for touch panel

3.1. Follow the basic principles of analog - digital hybrid circuit design

The analog circuit of precision capacitance measurement is integrated in the capacitive touch sensing chip, so it should be treated as an independent analog circuit in PCB design. Follow the basic principles of analog - digital hybrid circuit design.

3.2. Adopt star connection to ground

The ground wire of the touch IC shall not be shared with other circuits, and shall be separately connected to the grounding point of the power input of the board. It is recommended to use “star connection” to ground. Refer to Figure 1 for the connection mode.

3.3. Influence of noise generated on power supply on touch MCU

The same procedure should be followed for the power circuit. The touch IC is best to use a separate cable from the board power supply point, do not share the power loop with other circuits. Refer to Figure 1 for connection mode.

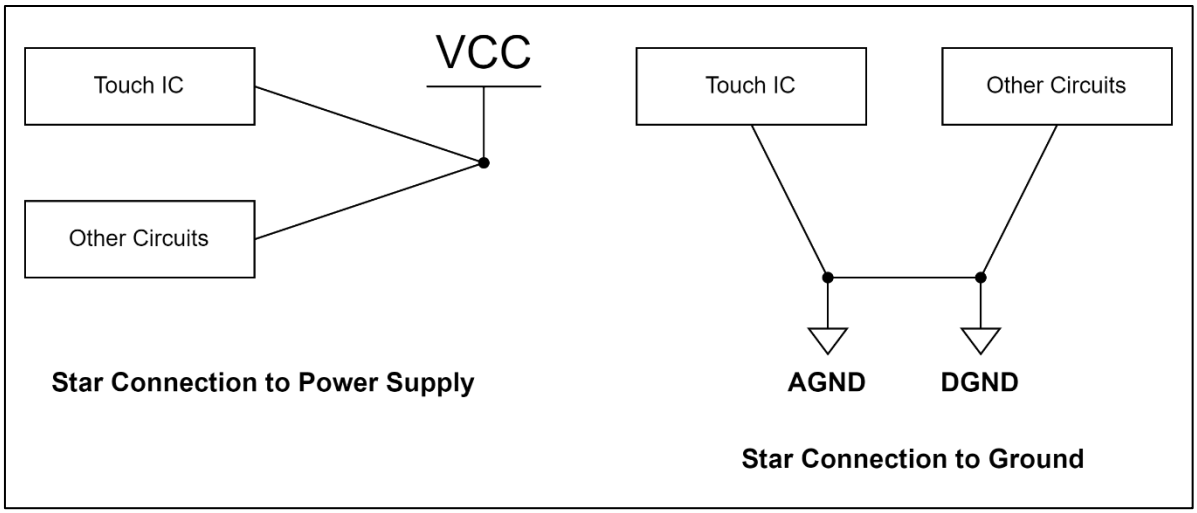

Fig. 1: Schematic diagram of connection method of VCC and GND

4. Touch Pad Design

4.1. Touch pad material

The touch pad includes PCB copper foil, metal sheet, flat-topped cylindrical springs, conductive cotton, conductive ink, conductive rubber, ITO layer of conductive glass, etc. No matter what material is used, the key sensing pad should be as close as possible to the panel, and there should be no air gap in the middle. When flat-top cylindrical springs are used, the diameter of the hollow floor at the connection between the touch line and the spring should be slightly larger than the diameter of the spring column to ensure that the spring will not touch the floor even if compressed onto the PCB board.

4.2. Touch pad shape

In principle, it can be made into any shape, and holes or hollows can be left in the middle. The general application of round and square is more common.

4.3. Area size of touch pad

The actual area size depends on the requirements of the sensitivity, and is proportional to the sensitivity. In general, the diameter of the button sensor pad should be greater than 3 times the thickness of the panel. The shape and area of each sense element should be the same to ensure the same sensitivity. In general, 12mmX12mm is typical for most applications.

4.4. Distance between touch pads

The distance between each touch pad should be as large as possible. The suggested minimal spacing between electrodes or ground pad should be larger than 0.5 time of the panel thickness. Appropriately widening the distance between touch pads is helpful to improve touch sensitivity.

4.5. Connection mode between touch pad and various touch structures

a. When using copper foil of PCB for touch pad make sure touch pad is snug to PCB.
b. Use a spring-attached patch touch pad and place the touch pad on top of the panel.
c. Use conductive rubber or cotton. The bottom of conductive cotton or rubber is glued to the copper foil of PCB, and the top is glued to the panel as an induction.
d. A flexible PCB made of conductive ink or ITO is inserted into interface of the touch port.

5. Design recommendations for passing EMC tests

Please refer to the following suggestions for passing rigorous FCC tests.

5.1. Use decoupling capacitor

Please add a decoupling capacitor for the power supply of the touch MCU. Generally, a 0.1uF/1uF MLCC capacitor and a 10uF/100uF MLCC capacitor are connected in parallel at the VCC and GND terminals of the chip, which can play the role of decoupling and bypassing and also can provide less ESR than ceramic capacitor. Decoupling capacitors should be placed as closer to the chip as possible.

5.2. Use lower LDO voltage

To have the sufficient tolerance for the signal. A selection of lower LDO output voltage is preferred.

5.3. Reasonable copper wires layout

Whether using a single-sided PCB or a double-sided PCB, the blank area of the PCB should be covered with a 1/4 fill hatched ground copper. And the wiring between the key sensing pad and the input pins of the IC should be wrapped with the copper. This can absorb electromagnetic radiation and improve the EMC index. It is recommended to use double layer PCB.

6. Component layout

6.1. The location of the chip

There is a function of line length correction in touch sensing IC, so the difference of the line length between each key sensing pad and IC pin will not lead to significant difference in key sensitivity. However, if the PCB board space allows, the touch chip should be placed in the middle of the touch pad as far as possible, so that the distance difference between the pins of each induction channel of IC and the sense element is minimal.

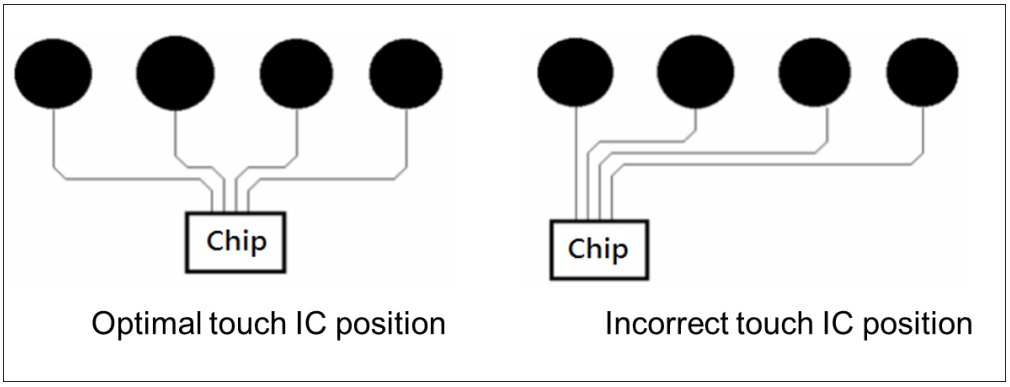

Fig. 2: Touch the lead wire

6.2. Clearance of the touch trace line

The space between the trace line of sensing pad and the ground is the good method to reduce the parasitic of the self-capacitance. However, the larger separation makes the touch line more susceptible to radiation noise. The common clearance is about 12% of panel thickness.

6.3. Touch channel series resistance

6.3.1 Position of touch channel series resistance

Touch channel series resistance (the series resistance between the touch button and the touch chip), it is better to place the key pin close to the touch chip.

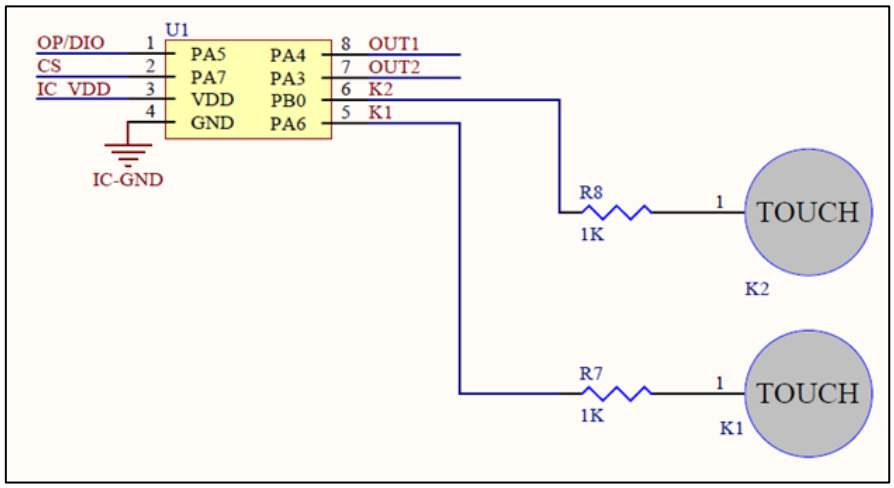

Fig. 3: Schematic diagram of touch channel series resistance

6.3.2 Selection of touch channel series resistance

The series resistors work for the system required Electrostatic Discharge (ESD) protection. They also help to lower the system emissions and improve RF noise immunity.

To evaluate the suitable input channel resistance, please check the highest operating frequency and the maximal supported self-capacitance range first for the touch MCU. Take PMS160B for example, if the channel needs to support the maximal self-capacitance value, that is 243pF, and operate in 100KHz. The recommended channel series resistance should be between 500Ω and 4KΩ.

6.4. Configure the appropriate sensitivity

a. Sensitivity is proportional to the power value of the touch library. When the value increases, the sensitivity increases; when the value decreases, the sensitivity decreases.
b. Sensitivity is inversely proportional to the thickness of the panel.
c. Sensitivity is proportional to the size of the touch pad. For the susceptible radiation noise, it is vice versa.

7. PCB Board Wiring

7.1. Double layer PCB board wiring

If the copper foil on the PCB board is directly used as the touch sensing pad, it is recommended to use a double layer PCB board, and the touch chip and the connection from the sensing disk to the chip pin should be placed on the bottom. Induction panel should be placed on the top layer, installed close to the touch panel.

7.2. Single layer PCB board wiring

If a single-sided PCB board is used and a spring or other conductive object is used as the sense element, it is recommended to avoid or reduce the tripping line between the induction board and the IC pin.

7.3. Line idth

If the PCB process permit, the connection between the sense element and the chip should be as thin as possible. Short and thin sense line is recommended.

7.4. Wire routing rules for touch trace line

To avoid the interference from signal cables or LED PWM signals, the touch trace line must keep at least 4mm away from the parallel digital signals or can cross them vertically. (in double layer PCB board, it is not recommended that the two layers of the panel be closely parallel). Please check the Figure 4 for the better way of connection and wire routing between IC and the sense element.

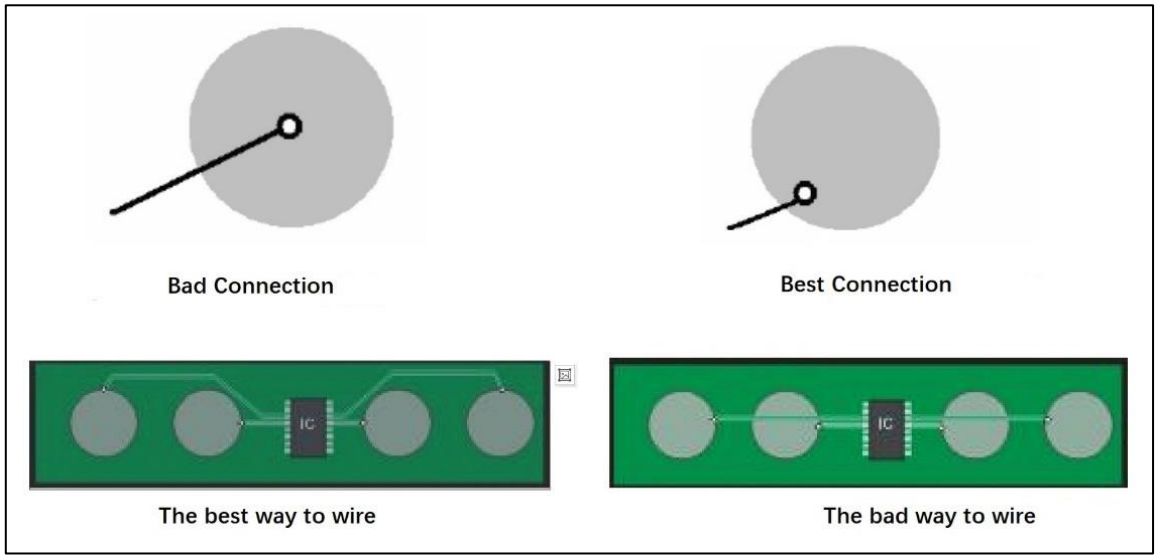

Fig. 4: Good and Bad Wiring

8. Place copper pour for the ground plane

The grounded copper pour can effectively improve the anti-interference ability of the product. It is recommended that the touch pad be a 0.5 time of panel thickness away from the grounded copper pour. In this distance, the interference immunity and touch sensitivity of the system can be balanced effectively. If the distance is reduced, the immunity of the system can be improved, but the touch sensitivity will be reduced. On the contrary, when the distance increases, the sensitivity of touch can be improved, but the immunity is slightly decreased, so users can adjust it appropriately according to each own need. Having a 1/4 grounded hatched plane can effectively reduce the parasitic capacitance for SNR improving. The designer can change the percentage for the balance of sensitivity and noise immunity.

8.1. Double Layer PCB Board

8.1.1. The Top Layer

The sensors can be placed here with the grounded copper pour in the form of a grid. The grid line width and grid size can take 8mil and 64mil, respectively, for a typical 1/4 hatching requirement. The hatching percentage should not exceed 40%.

Fig. 5: Laying grid copper outside the top key sense element

8.1.2. The Bottom Layer

All of the components including MCU can place on the bottom layer, but it needs to avoid directly placed under the sensor by instead of the 1/4 hatched ground pour.