tenx Touch Application Design Reference

For TM57PT20/TM57PT16/TM57MA21/ TM52M5258/F5278 Series

Application Note

Rev 1.0

AMENDMENT HISTORY

Version	Date	Description
V1.0	Jun, 2015	New Release.

CONTENTS

1. CLD Capacitor Selection

- CLD capacitor should select Precision polyester capacitors or NPO, X7R capacitors, for better temperature stability of the capacitor, the touch data is relatively stable.

- CLD capacitor can be used to increase or decrease the value of the touch counter value, can also use TKTMR to fine-tune.

- CLD capacitor to IC pin should serial a 100 ohm resistor, touch keys to IC pin should serial a 1K ohm resistor. Resistor and CLD capacitor should be placed as close as possible to the IC pins. Serial resistor can avoid Interference and improve the touch performance.

2. PCB Design Guide

- Place the IC in the middle of the PCB as possible if PCB space allowed, so that the IC to the Touch Pad connection could be as equal length as possible. Place the Touch Pad and IC first while planning the PCB layout design then other layout. Touch pads to IC pin connections should be short and thin, width 7-10mil (the thinner the better), line length as short as possible (no longer than 300mm). Should avoid Touch Pads to IC pins through jumpers.

- Touch Pad can use copper on PCB, also can use flexible printed circuit board (FPC), ITO or silver paint printed conductive material to complete. ITO require higher signal, pay special attention to design.

- Touch Pad area should not be too small, otherwise the changes of touch amount will be not enough, with Touch Pad sensitivity is proportional to the area. Area recommendations about a human finger contact area (10 ~ 12.5mm * 10 ~ 12.5mm or 12 ~ 20mm diameter), too small will affect the touch sensitivity.

- Sensing Pad can be any shape, but it is recommended that concentrated in a square or circular aspect ratio, the maximum bonding finger contact surface to ensure good effect, sensors designed to elongate shape should be avoided (except for proximity sensing or the induction of non-human finger contact).

- Slider, wheel Pad also apply, but the wiring requirements more stringent recommendations, please sent the former PCB to our PAE support engineers to make a preliminary assessment and guidance before PCB proofing.

- Distance between Sensing Pads better be more than 5mm (larger the space, less the interference), at least to have three times more than the width of the pitch is required (traces came out of IC pins should follow this rule).

- Try to keep the same length for each sensor Pad traces, and away from the edge of the PCB to reduce noise interference.

- Sensing Pad traces are preferably away from the high frequency signal line, never parallel to the high frequency signal line, if necessary in a vertical way across the high-speed signal lines.

- Do not circling the sensor Pad line around the sensor Pad, so as to prevent antenna effect. Sensing Pad trace can not across directly beneath the other Sensing Pad.

- The Via (through hole) number of the Sensing Pad trace should be as less as possible, if necessary, it is best to use only a hole, through hole diameter is as small as possible to reduce the effects of parasitic capacitance (suggested Via size for no more than Hole size: 8mil, Pad Diameter: 16mil).

- At double-sided PCB layout recommendations of the PCB Top Layer best placed only Touch Sensing Pad, no other parts. The IC and other passive components is placed at the Bottom Layer of PCB. Recommend PAD trace going through the Via in the Pad area and connect to IC in the same bottom layer.

- Single side PCB generally recommended to use the touch spring, a leaf spring Pad lead can be on the top or bottom, the other rules are the same.

- Figure 4, 5 are double-sided PCB design hint diagram.

- Without Ground Plane, the change rate would be much better, if the ground plane is necessary, please stay away from the Touch Pad and it's trace.

- Usually PCB rarely exposed directly to end users, but adding the covering material among the PCB, in order to avoid direct contact of the circuit board or a circuit board in direct contact with the external environment. Touch sensing applications coverings must not be a conductor. When the metal or other conductive material placed between two conductive plate, such as the fingers and the PAD, can not form the capacitor dielectric. Please refer to the plate capacitance formula.

- The thickness of the insulating dielectric will affect the sensing capability of the Touch Pads, the thickness of the surface material is inversely proportional to the sensitivity, it is recommended 2-4mm thickness with better dielectric constant material such as glass, acrylic, etc. in order to maintain good sensing capability.

- A gap between the Pad and Dielectric is not recommended (proximity sensing is an exceptions).

3. Debug Mode Requirement

- To facilitate debugging, Please reserved simulate communication I/O test points close to the IC pin.

4. PCBA Testing

- Make sure the board is clean without flux residue to relevant circuits of Touch Pads to ensure stability of the touch data.

- Ensure accurate positioning fixture thimble, recover test switch after completion.

5. End Product Assembly

- Housing or other structures fixed PCB assembly must use insulated paint, can not contain metal powder or toner. The interaction between the chassis and Sensor circuit could affect the sensitivity of the sensor. Grounded metal chassis can lower the sensitivity.

- The four most commonly housing element that affects the touch sensitivity is the metal components, communication cable, battery and plating layer of the cover of the sensor PCB.

- Whenever possible, metal components away from sensor elements and traces. If necessary to use the components recommended non-metallic components. If you must use a metal component or as a garnish placed next to the sensor, it must be grounded. The chassis can also be connected to a passive shield.

- No matter when, communication cables should stay away from sensor elements and traces.

6. Power Requirements

- TENX series voltage range can be up to 2.5V-5.5V (depending on the specifications for each model). In operating state, IC supply voltage must be ensured a relatively stable (usually less than 250mV / min). Therefore, in most cases, we recommend that the power supply using a three-terminal regulator IC (LDO), to ensure a stable IC voltage. The use of batteries, will have to consider the impact of the entire touch sensitivity by voltage fluctuation after prolonged use.

- In many applications, even the use of the three-terminal regulator IC, but because the system interference, big loading effect and other reasons, IC voltage will have some ripple interference. In order to prevent such interference, in order to get a better touch effect, it is recommended to add some filter circuit.

Fig. 1: RC filter circuit can prevent system interference.

Fig. 2: LC filter circuit can prevent a large high-frequency interference.

Fig. 3: Diode circuit can prevent the voltage fluctuations from motor starts, buzzer and infrared emission or other.
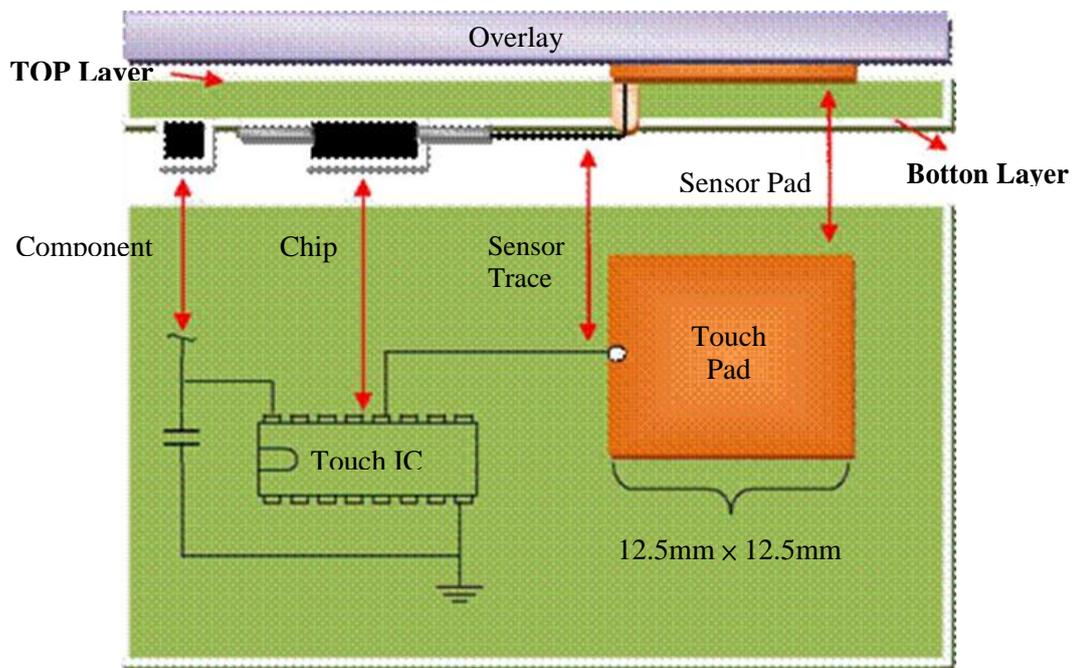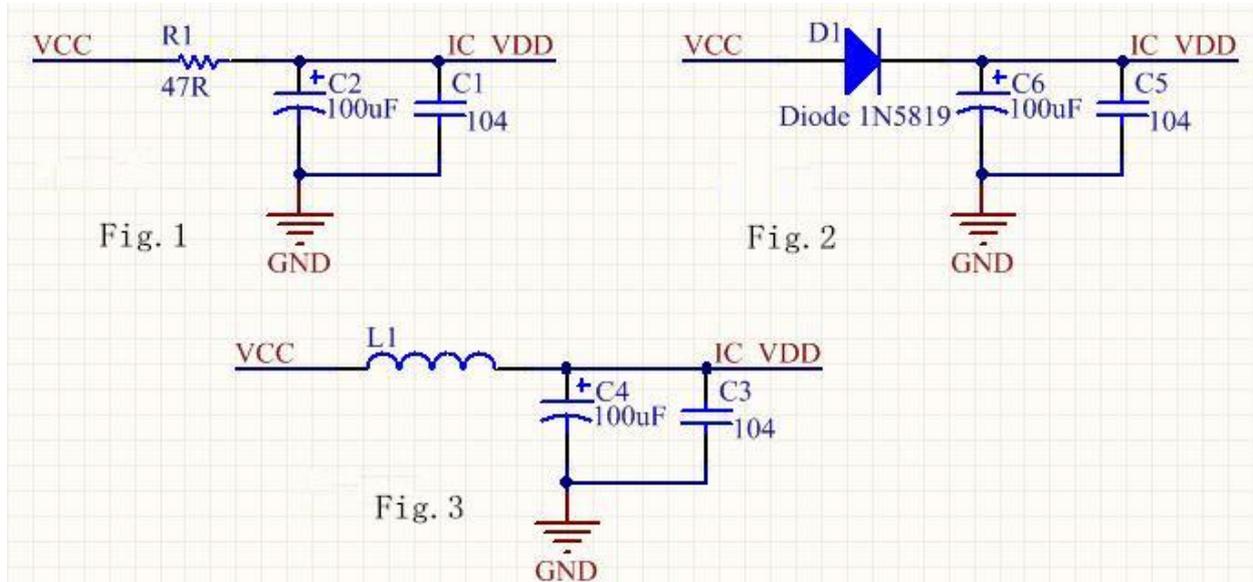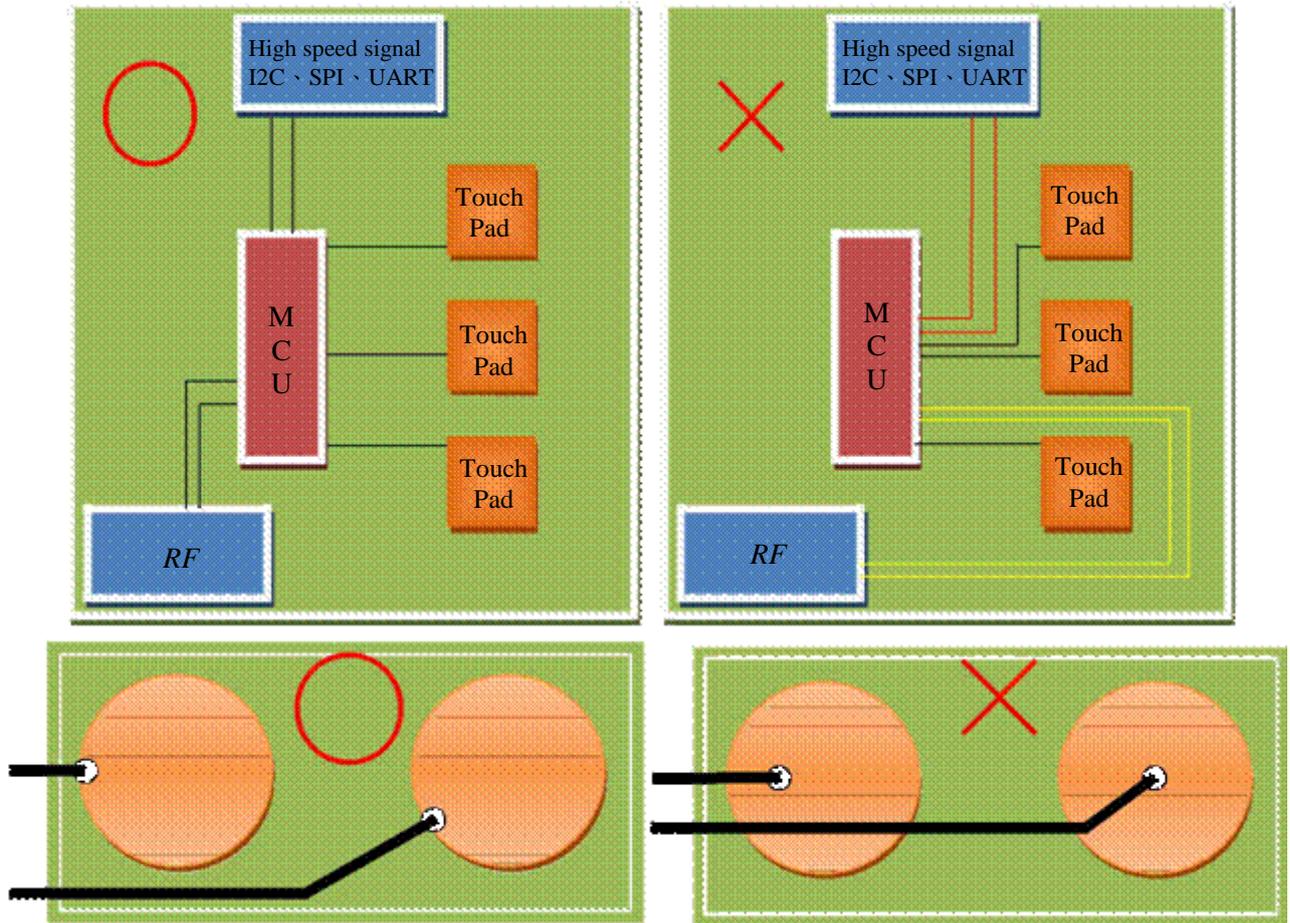

Fig 4, double-sided PCB touch keys sectional and aerial view

Fig 5, Touch Pad PCB layout considerations