TK18
PCB Design Guide

This Document is for TK18 Series

Revision 1.0
Oct. 2014

Headquarters

4F-1, No.9, Prosperity Rd.,

Science-based Industrial Park,

Hsinchu City, Taiwan, R.O.C

TEL: 886-3-6662888

FAX: 886-3-6662999

Revision History

Revision	Description	Date
1.0	First release.	2014/10

Contents

1. Operation Mode

ENE TK18 MCU series is embedded with flash memory and TK (Capacitive Touch Key) sensors. It works with Host and Non-Host operation modes.

In Host operation mode, there is a system MCU in the designed system and TK18 behaves as a slave controller. TK18 communicates with host MCU through I2C/UART interface or programmed GPIO. Please refer to datasheet or programming guide for detail settings.

In Non-Host operation mode, TK18 works as the system MCU or an independent MCU working together with others.

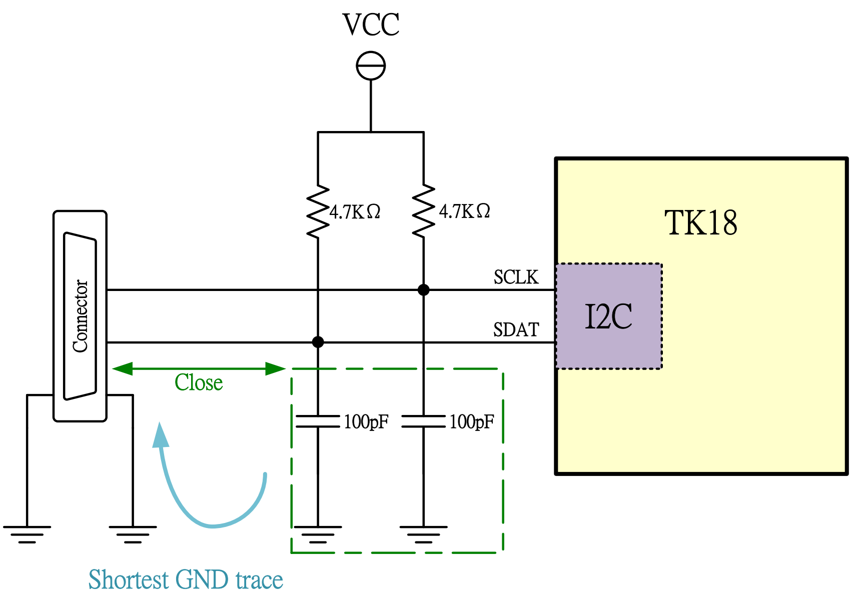
P/N	Host Interface	Non-Host Mode
TK18	I2C/UART/GPIO	Supported

1.1 The host interface for Host operation mode

I2C Interface:

When using I2C interface, the 4.7KΩ pull-up resistors are required for SCLK and SDAT signals. The pull-up resistors' values may vary depends on system design. TK18 I/O embeds internal pull-up resistor with 4.7KΩ or 40KΩ, users may enable either of them to fit system designs.

The decoupling capacitor (<=100 pF) is recommended to reduce power and signal noise, Decoupling capacitors should be placed close to the connector that interfaces with host MCU, ground loops should be short, direct back to the connector ground (GND).



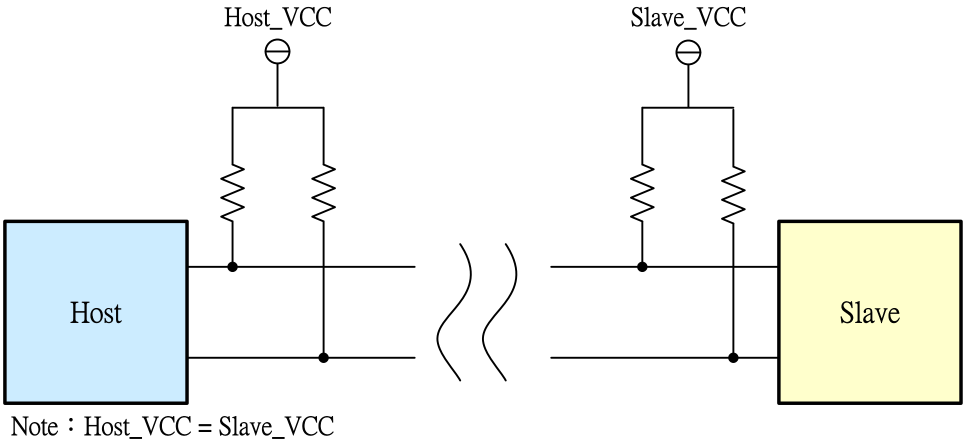
TK18 support I2C bus speed up to 200 KHz.

Note: The pull-up resistors on both system MCU and TK18 sides should refer to the same power level (VCC).

UART Interface:

User may define any two of the GPIOs to be UART TX and RX signals through F/W programming. System required baud rate can also be defined through F/W.

Note: The pull-up resistors on both system MCU and TK18 sides should refer to the same power level, Host_VCC = Slave_VCC.

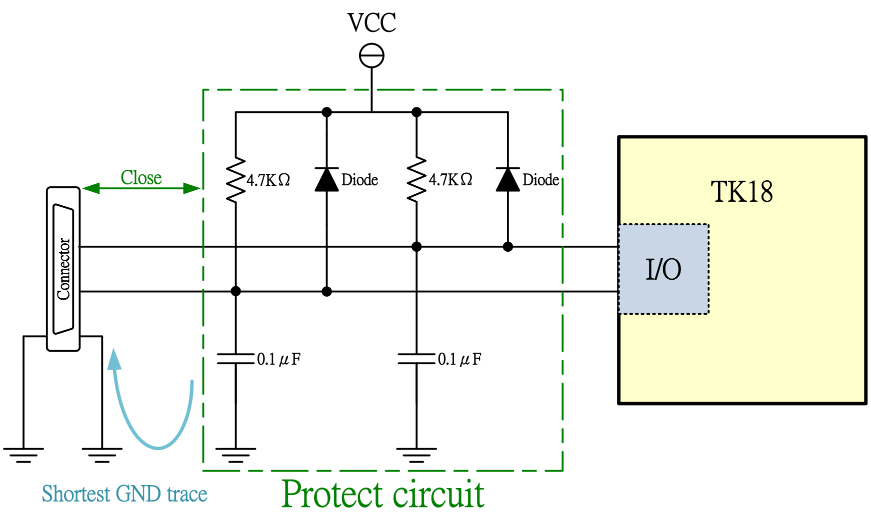


GPIO:

Using TK18 GPIO to communicate with Host MCU or system power board, the protection circuit is required. The protection circuit should be placed near to the connector to avoid unexpected noise, such as EFT noise.

To pass EFT 4KV test, there needs a path to ground to vent the abrupt noise pulses. If the designed PCB does not provide enough ground plane, it's recommended to have two I/O pins to connect to host MCU or power board with the protection circuit to protect TK18 and components. I2C or UART may behave as these two I/O pins once they are used to communicated with host MCU. The protection circuit is composed of one 100Ω series resistor, 4.7KΩ pull-up resistor and 0.1uF capacitor. The protection circuit is depicted below, where diode is optionally used to enhance the protection, please note the 0.1uF is required.

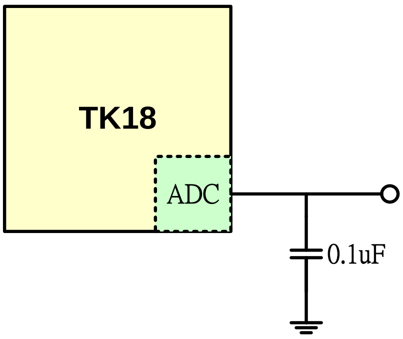
Note: The ground path between capacitor and connector should be as short as possible.



ADC:

There are 10-bit ADC embedded inside TK18 that may be used to detect environmental power and temperature status, and feedback information to host MCU through I2C and UART interfaces.

Note: 0.1uF capacitor is required at ADC input to filter noise. If abnormal system noise exists, a low pass filter (LPF) at ADC input should be considered.



1.2 Non-Host operation mode

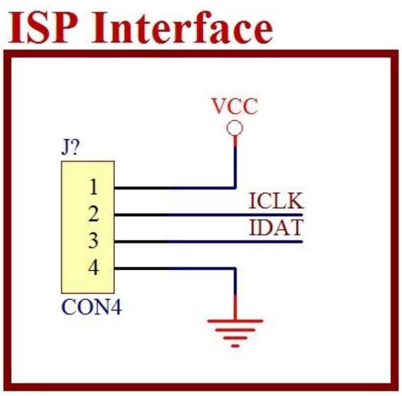
TK18 non-host operation mode is mainly used for:

1. Save Host MCU cost.
2. Reduce PCB layout complexities and area.
3. Reduce efforts and cost for FW programming.

1.3 Embedded Flash Programming Interface

TK18 embeds MCU and flash memory, the flash programming interface is ISP (ICLK and IDAT). There are flash programming tools available, please consult with ENE technical support.

If user may in-system program flash code, or want to in-system monitor TK performance, the ISP interface/connector should be reserved on PCB.



2. Power Supply

The quality of system power is a key factor to TK sensing technology. A good power system design improves TK performance dramatically. In some power noise tests, TK18 provides HW and FW solutions to compensate system power variations during tests.

2.1 TK18 Internal Power (Internal LDO)

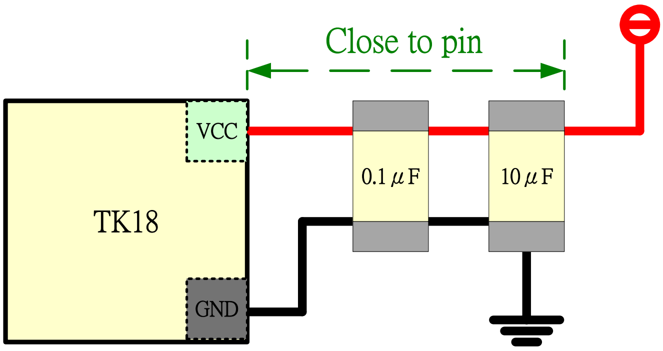
LDO_1.8V

There is an internal LDO to provide TK18 stable 1.8V for circuit operation. This power is not used for power source of other components. One 4.7uF capacitor is required for internal 1.8V LDO, it's recommended to place capacitor close to CLDO1.8 input pin.

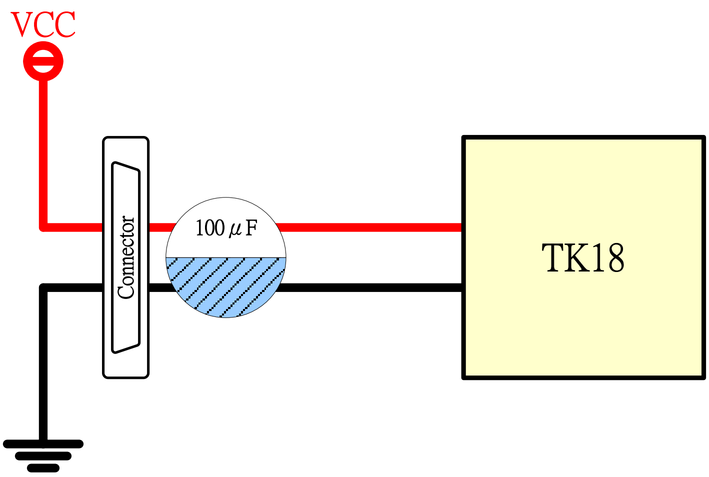
2.2 System Power – VCC

VCC

A pair of low ESR and ESI X7R ceramic surface-mount appliance (SMD) 0.1uF capacitors in parallel an X7R or X5R ceramic surface-mount appliance (SMD or electrolytic capacitor) 10uF capacitor should be placed as close to the VCC and GND pins of TK18.



Note: For EFT +-4KV test, adding a 100uF bypass capacitor to power input is recommended.



2.3 VCC Power Range and Requirements

The VCC power range for TK18 is 3.0V~5.5V. That means TK18 works normally with VCC power varies between 3.0V~3.6V or 4.5V~5.5V, instead of supplying VCC varies between 3.0V ~ 5.5V.

Although TK18 is able to maintain instantaneous voltage variation and power supply noise at different frequencies, please note the voltage noise level must be less than ±100mV. Impact of any instability or impulse noise applied to the VCC will seriously affect TK behavior.

If possible, please have a good PSRR LDO for VCC, and position it as close as possible to TK18.

3. PCB Design

In typical TK18 applications, the capacitive sensor may be constituted by the PCB routing traces and different shapes of sensor pads. The following sections depict PCB design guidelines for using TK18.

Please note that any noisy components, such as switches, transformers, oscillator, etc., must be placed as far away as possible from TK18, sensor pad and sensor routing traces. Other metal base or digital signals may cause capacitive crosstalk to affect TK18 performance.

3.1 PCB Layout

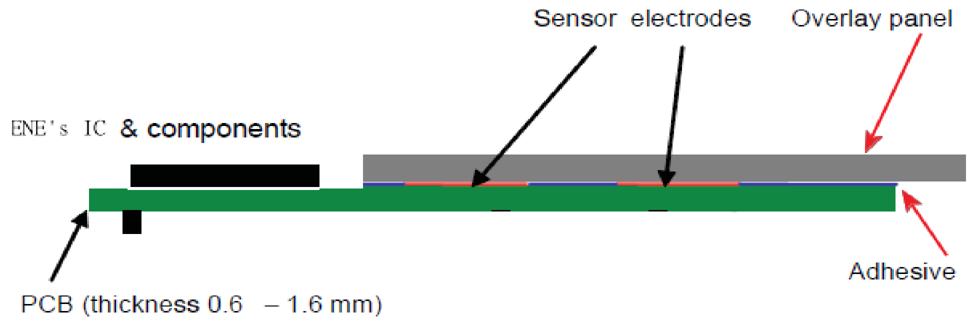
Components:

Place the sensor pads on top or bottom layer of PCB, regardless how other components are placed.

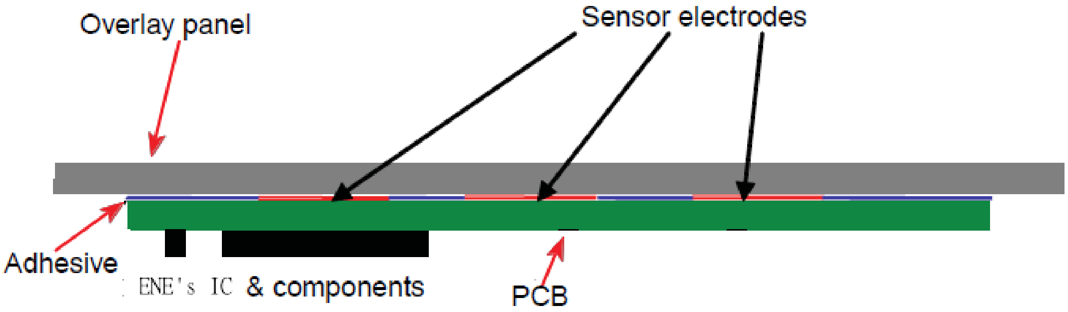
Sensor Pads:

The sensor pads are recommended to placed on the PCB top layer to optimize the sensitivity.

Case I: ENE TK MCU is placed on the same PCB layer of sensor pads:



Case II: ENE TK MCU is placed on the different PCB layer of sensor pads:



3.2 Multi-Layer PCB Stack-ups

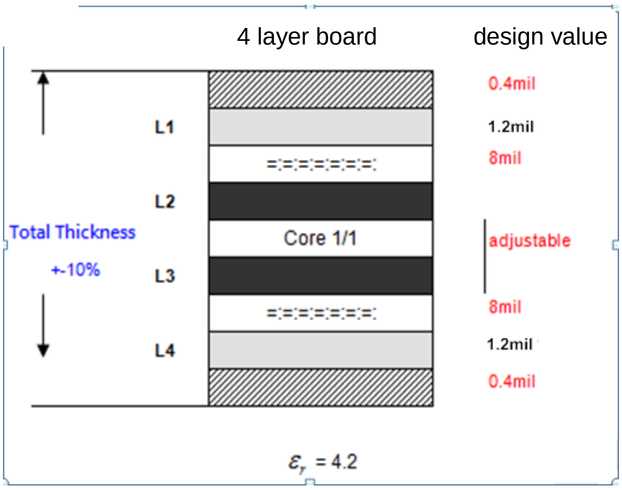
The standard FR4 PCB thickness is 1.0mm. TK18 is capable to work from 0.6mm to 1.6mm.

2-Layer PCB

Sensor traces maybe routed on top and bottom layers, and the mesh ground is recommended. If the top-layer is used as shielding layer, the sensor traces and mesh ground should be routed on bottom layer.

4-Layer PCB

Taking FR4 PCB as an example of multi-layer PCB, the stack-up information is below:



- Top layer (L1): There is only sensor pads, copper mesh and shield layer.
- The 2nd layer (L2): There is only sensor traces.
- The 3rd layer (L3): The logic control lines, LED lines and power/ground lines.
- Bottom layer (L4): The power/ground lines, copper mesh and components.

3.3 Power and Ground Design

The quality of power and ground design is fundamental and key to capacitive touch key performance, please note and follow the design rules below:

Mesh Ground:

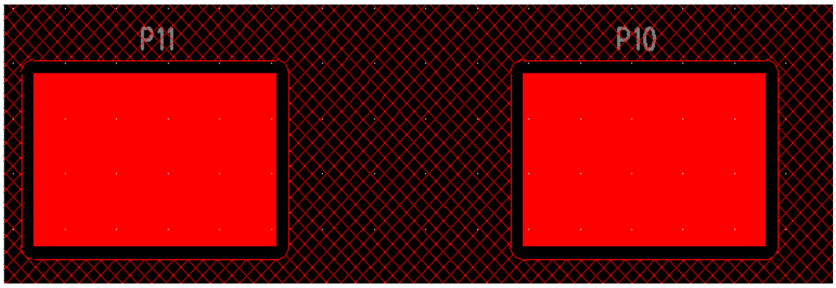
If shielding layer is not used, it's recommended to have mesh ground on top and bottom layers. The mesh ground improves not only TK sensitivity but also noise immunity.

To obtain good SNR and RF noise immunity, it's recommended to route the ground trace with 5mil trace width. The spacing between traces is 20mil, the hatch grid is 20mil, and the spacing between sensor pad and mesh ground is 20~40mil.

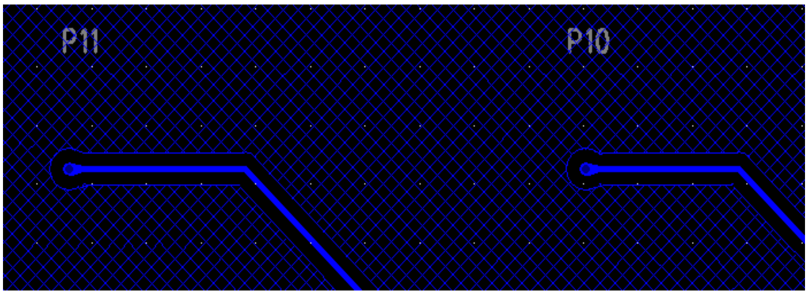
However, some of PCB manufacturing houses require the routing trace width to be more than 10mil to achieve better production yield, in this case, it's recommended to have ground trace width is 10 mil, and the hatch grid is 40 mil accordingly.

The following pictures illustrate the sensor pads on top layer and the sensor traces on bottom layer.

The sensor pad and mesh ground on top layer:



The sensor traces and mesh ground on bottom layer:



3.4 PCB Layout

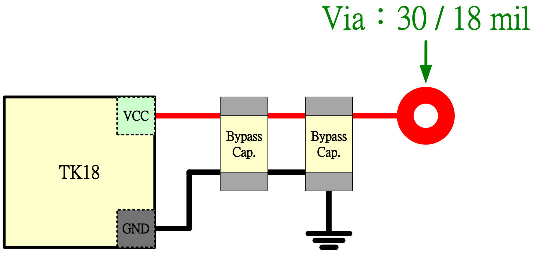
Please follow general PCB layout rules, for instance, to route the traces with 45-degree instead of right angle to avoid EMI issues.

VCC/GND traces and through holes for power source:

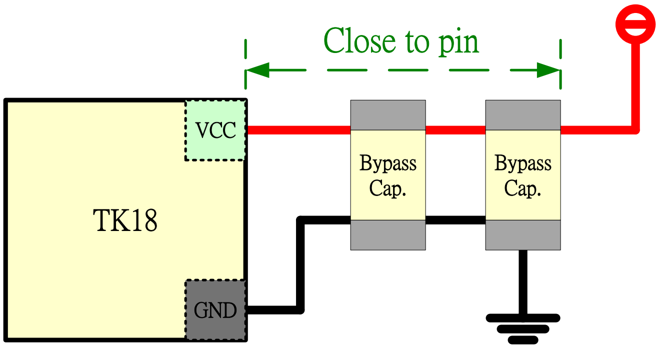
VCC/GND traces are recommended to be **25~30mil** width to reduce equivalent resistance and inductance.

The decoupling capacitors are recommended to add to through holes of power source as illustrated below, the noise introduced by power source will be decoupled by the capacitors. The through holes of VCC/GND are recommended to be **30/18mil** to reduce equivalent inductance.

TK18 VCC trace and through hole:



Note: The VCC traces between VCC and TK18 should go through the decoupling capacitors instead of routing branches to capacitors. The decoupling capacitors should be places as close to the TK18 VCC pin.

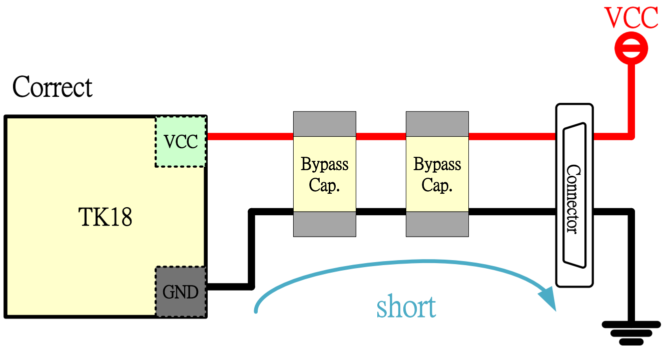


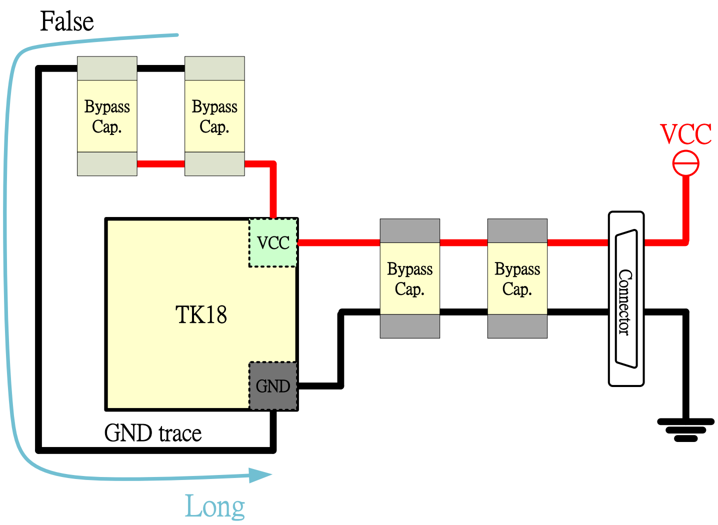
The trace for internal LDO bypass capacitor:

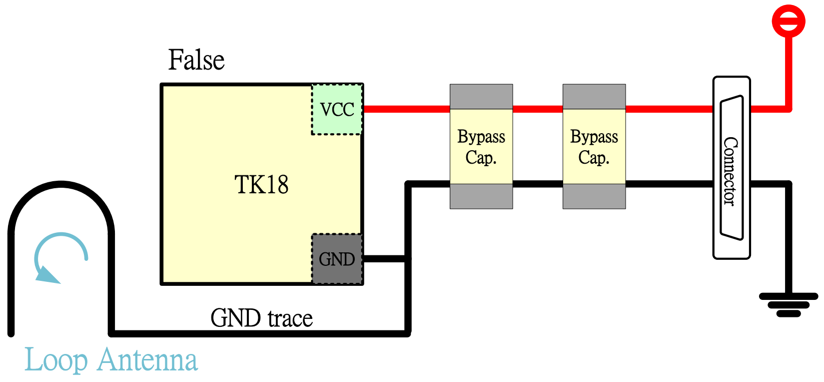
It's recommended to have **10~20mil** trace width for internal LDO bypass capacitor routing to provide TK18 a stable power source.

The ground return path of IC:

The ground return path of TK18 should be connected to the ground of VCC decoupling capacitors, and route the ground trace as short as possible to the connector. Avoid redundant routing traces as well as becoming "Loop Antenna".







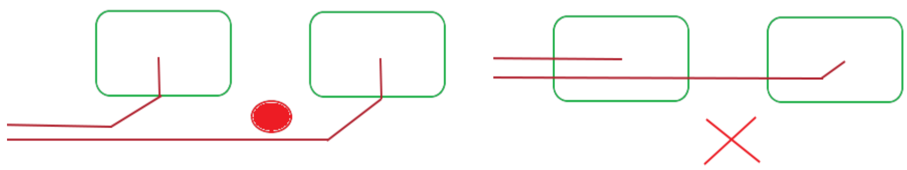
Sensor Pad/Touch Key

For 2-layer PCB case, it's recommended to have **5~10mil** trace width to connect sensor pad/touch key with TK18. The distance between every two sensor traces is recommended to be twice of the sensor trace width (**10~20mil**) to avoid interference.

The longer sensor trace length introduces larger parasitic capacitor, so the sensor traces should be as short as possible, it's recommended to be within **150mm**.

Note: Do not route any signal trace to overlay sensor pad, that will cause severe crosstalk to fail TK function.

Preferred vs. inappropriate sensor trace routing examples:

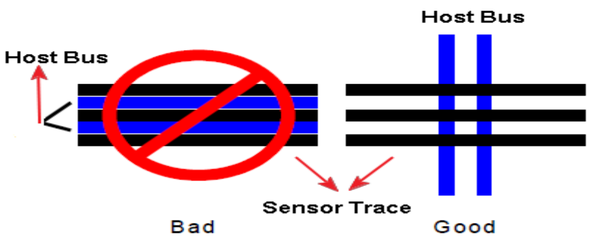


The digital signal traces between host MCU and TK18:

The digital signals communicate between host MCU and TK18, the digital signals switching frequency is higher than TK signals, it's recommended to have **5~10mil** trace width. The trace spacing between clock signals and other signals, such as power, data, control and TK sensor, is recommended to be at least twice of the trace width (**10~20mil**).

The digital signals are prohibited to be routed in parallel with the sensor traces, the parallel structure will cause crosstalk. If any of the digital signals cannot be avoided to be routed close to sensor traces, it's recommended to route the cross section with right angle to minimize crosstalk effect.

Inappropriate vs. preferred digital signals and sensor traces routing example:

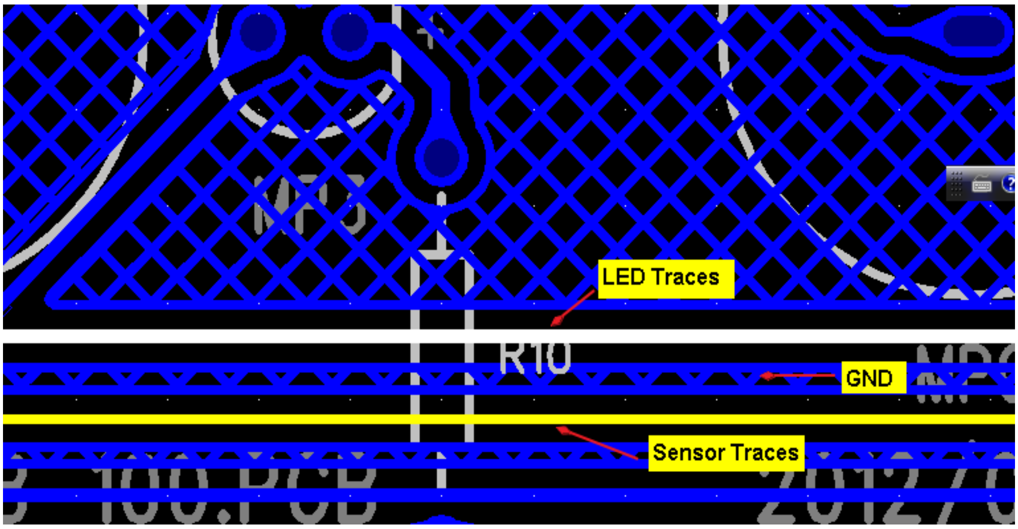


LED signals and sensor traces

The LED control signals are prohibited to be routed in parallel with the sensor traces, please insert ground shielding lines between LED control signals and sensor traces, otherwise, the traces routing spacing should be at least 50mil.

Note: The inserted shielding lines should be connected to ground at both sides. Otherwise, it will form a uni-polar antenna that suffers EFT immunity.

The ground shielding lines between LED control signals and sensor traces:



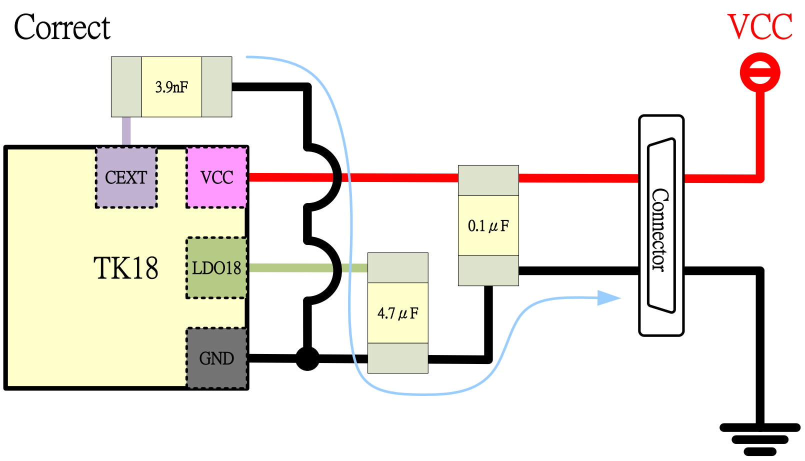
Note: If LED control signals crossover sensor traces, it is recommended to use another plane jumper to cross lines instead of using SMT components. By doing this, the vertical distance between LED control signals and sensor traces will maintain in PCB thickness (1.6mm) to guarantee EFT immunity.

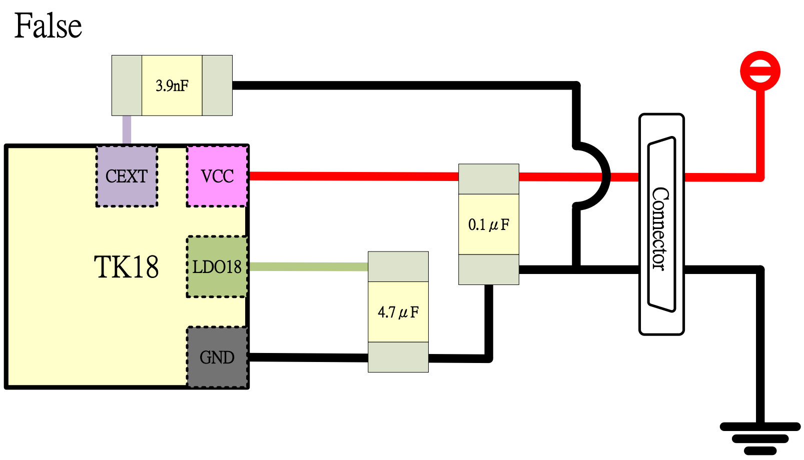
3.5 Cext

Cext is the external capacitor used for TK charge transfer operation, this is one of key factors to capacitive sensing technology, it's recommended to place this component close to TK18 and ground, the trace width connected to ground is at least **10mil.**

The recommended Cext for TK18 is 3.9nF (ceramic capacitor).

Note: The ground of Cext needs the shortest path back to the ground of VCC decoupling capacitor (0.1uF) and LDO18 capacitor (4.7uF), then connect to the Power/GND connector. The routing example is illustrated below.





3.6 Sensor pad and through hole for TK

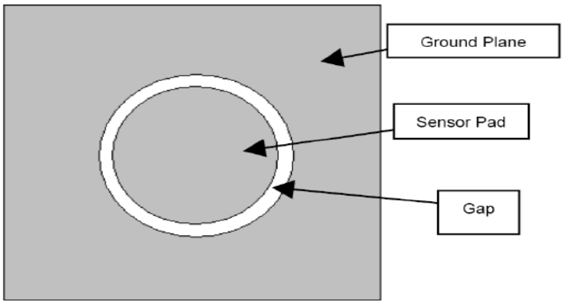
The sensor pad design is fundamental to TK performance.The following chapter depicts the design guideline for sensor pad shape, size, placement and trace routing.

The size and shape of sensor pads (TK)

The proposed shape of sensor pad is round or rectangular, please refer to the diagrams below. The recommended size is **10mmx10mm (square), 8mmx12mm (rectangular), or 11mm diameter (round).**

Preferred vs. undesirable shapes of sensor pad

The sensor pad design example for round shape



A typical round shape sensor pad is illustrated above. There is a round gap that isolates the sensor pad and the ground plane.

In the case of sensor pad with 15mm diameter or larger, the expansion of sensor pad size is ineffective to increase sensitivity. In contrast, larger sensor pad size will decrease the spacing between each other that may cause false trigger due to crosstalk.

In the case of sensor pad with 5mm diameter or less, the sensing capacitor will be too small to achieve adequate sensitivity.

The spacing between sensor pads

When the round shape sensor pad is used, it is recommended to reserve at least radius spacing between sensor pads. When the rectangular shape sensor pad is used, the recommended spacing between sensor pads is **2.5mm**.

The spacing between sensor pad and ground plane

It's recommended to reserve **20mil (0.5mm)** spacing between sensor pad and ground plane.

Through hole

It's recommended to route the shortest trace length between TK18 and sensor pad without through hole, if the through hole is inevitable, the through hole number should be less than 2.

The through hole can be placed anywhere of sensor pad, please refer to the following diagrams. The through hole at the center of sensor pad has symmetry physical characteristics. However, the through hole at the edge of sensor pad has shorter routing trace that depends on system design flexibility.

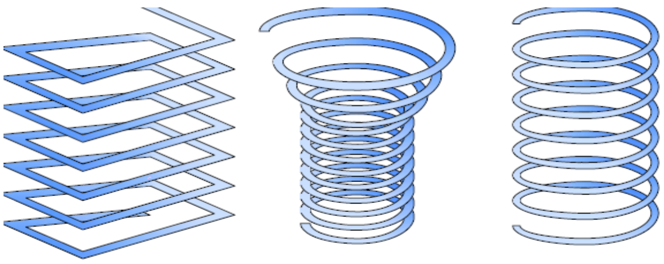
Through hole of sensor pad

Special structure sensor pad (non-PCB sensor pad)

For some of the home appliance applications, there needs a longer distance (>**5mm**) between touch cover and sensor pad due to system designs, in these cases, the spring will be used to compensate the distance.

It is recommended to use spring with diameter>**10mm** and length>**5mm** to maintain better SNR, the proposed shapes of spring are illustrated below:

Shape of spring



3.7 LED Applications

For touch applications, LED light is usually used as a touch indication.

For the case to place LED light inside sensor pad to respond corresponding TK event, it's recommended to place LED light at the center of sensor pad, and have the PCB opening size about 1.2 ~ 1.3 times of the size of LED light, that maintains better LED lighting performance.

3.8 CS Test

To pass CS 10V class-A criteria, ENE provides X-LIB (F/W library) to work with adequate serial resistor (Rs) on each sensor trace.

The serial resistor (Rs) on sensor trace

If the spring is used for sensor pad, 10KΩ to 20KΩ Rs is recommended. If PCB sensor pad is used, 5.1KΩ to 10KΩ Rs is recommended. The Rs provides good noise immunity to RF and CS interference.

Signal strength and Threshold value

Adjust TK ON Threshold value to be 40% ~ 50% of signal strength (F_Signal) is a shortcut to pass CS test. Please refer to X-LIB for detail descriptions.

Contact Details

- Headquarters
  4F-1, NO.9, Prosperity Rd I, Hsinchu Science Park, Taiwan
  TEL: +886-3-6662888 FAX: +886-3-6662999

- Taipei Sales Office
  4F, No. 88, Bauchiau Rd, Shindian City, Taipei, Taiwan
  TEL: +886-2-89111525 FAX: +886-2-89111523

- Kunshan Sales Office
  Suite 1620, Building A, Modern Plaza, No.18 Weiye Road, Kunshan, JiangSu
  TEL: +86-512-50315768 FAX: +86-512-50315798

- Shenzen Sales Office
  Suite 625, East Block, Tian An Technology & Innovation Plaza Phase II, Tian An Cyberpark, Futian, Shenzhen, Zip code 518040
  P.R.C.
  TEL: +86-755-82507658 FAX: +86-755-82507532

