Telink FR1 PCB Design Guideline

AN-22051900-E1

Ver.1.0.0

2022/05/19

Keyword

Layout, FR1, PCB

Brief

This is Telink FR1 PCB design guideline which mainly introduces considerations when designing FR1 boards.

Telink Semiconductor

Published by

Telink Semiconductor

Bldg 3, 1500 Zuchongzhi Rd,

Zhangjiang Hi-Tech Park, Shanghai, China

© Telink Semiconductor

All Rights Reserved

Legal Disclaimer

This document is provided as-is. Telink Semiconductor reserves the right to make improvements without further notice to this document or any products herein. This document may contain technical inaccuracies or typographical errors. Telink Semiconductor disclaims any and all liability for any errors, inaccuracies or incompleteness contained herein.

Copyright © 2022 Telink Semiconductor (Shanghai) Co. , Ltd.

Information

For further information on the technology, product and business term, please contact Telink Semiconductor Company (www.telink-semi.com).

For sales or technical support, please send email to the address of:

telinksales@telink-semi.com

telinksupport@telink-semi.com

Telink Semiconductor

Revision History

Version	Change Description	Date	Author
V1.0.0	Initial release.	2022/05	Junyao MAO, Weixiang WANG

Telink Semiconductor

Table of Contents

Telink Semiconductor

List of Figures

Telink Semiconductor

1. Overview

With the same PCB size and the same quantity of components, generally the fewer the number of PCB layers, the more difficult the design.

Due to cost concern, PCB designs are increasingly preferred to use FR1 boards, single-layer boards, which leads to more obvious problems in wireless communication, including power interference, RF high harmonics, and etc.

This document uses the Telink SoC chips as a basis and the remote control design as an example to illustrate how to guide the design of FR1 boards to achieve fast development and avoid multiple iterations.

Telink Semiconductor

2. Application Board Structure ID

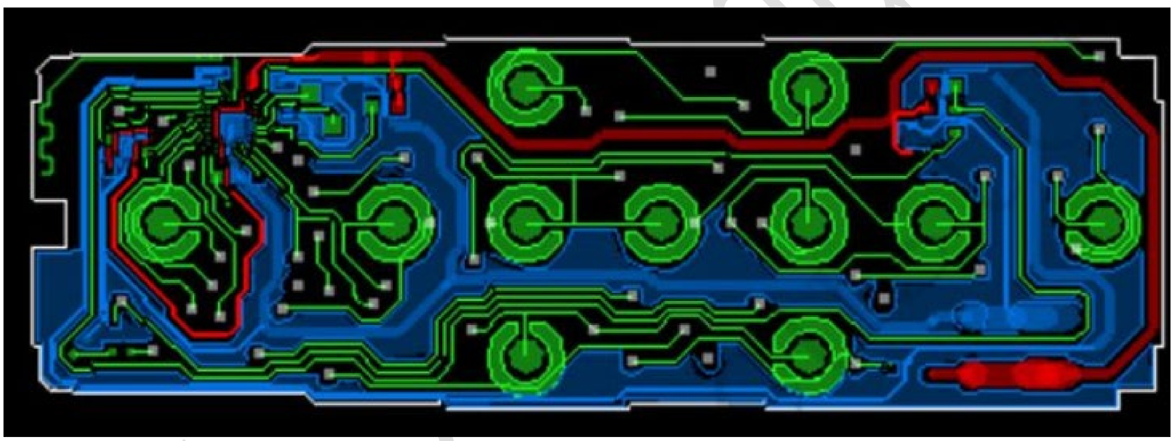
According to the complexity of the components, taking into account cost and the difficulty of routing, the the PCB design for Telink chips can be divided into single-layer boards or double-layer boards.

2.1 Single-layer board

In single-layer board design, make sure that all components and keys can be placed on the same side and there should be enough space for the PCB antenna. This is suitable for boards with a small number of components and routings.

The remote control board shown below can be designed as a single-layer board.

Figure 2-1 Single-layer board



Telink Semiconductor

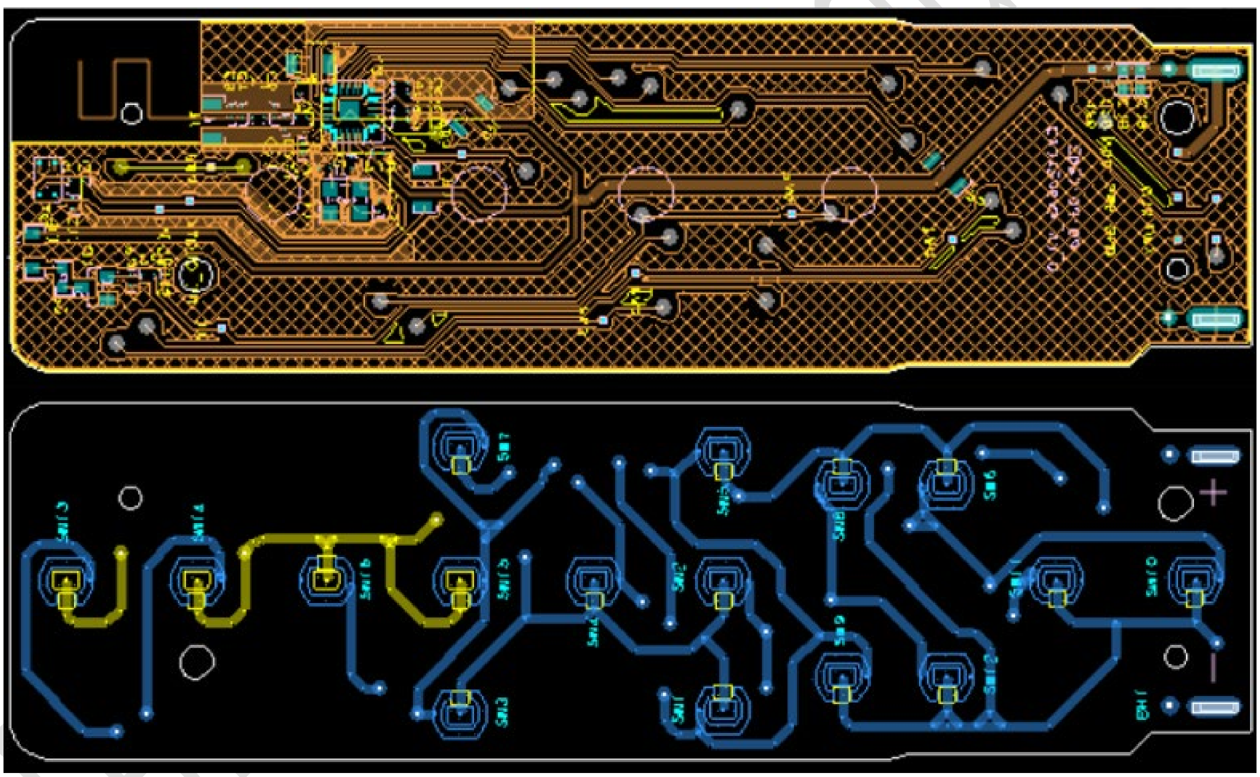
2.2 Doulbe-layer board

2.2.1 Component and copper wire layer + carbon film routing layer

One layer of this double-layer board is used to place components and route copper wire, and the other layer is for carbon film routing. For example, in a remote control design, we place the components on one layer and the keys on the other layer. The keys need to be designed as carbon film keys and the keys routing is connected to the component layer via carbon film via holes. Note that carbon film vias are chosen for cost concerns.

The remote control board shown below can be designed as this type of double-layer board.

Figure 2-2 Component layer + carbon film layer



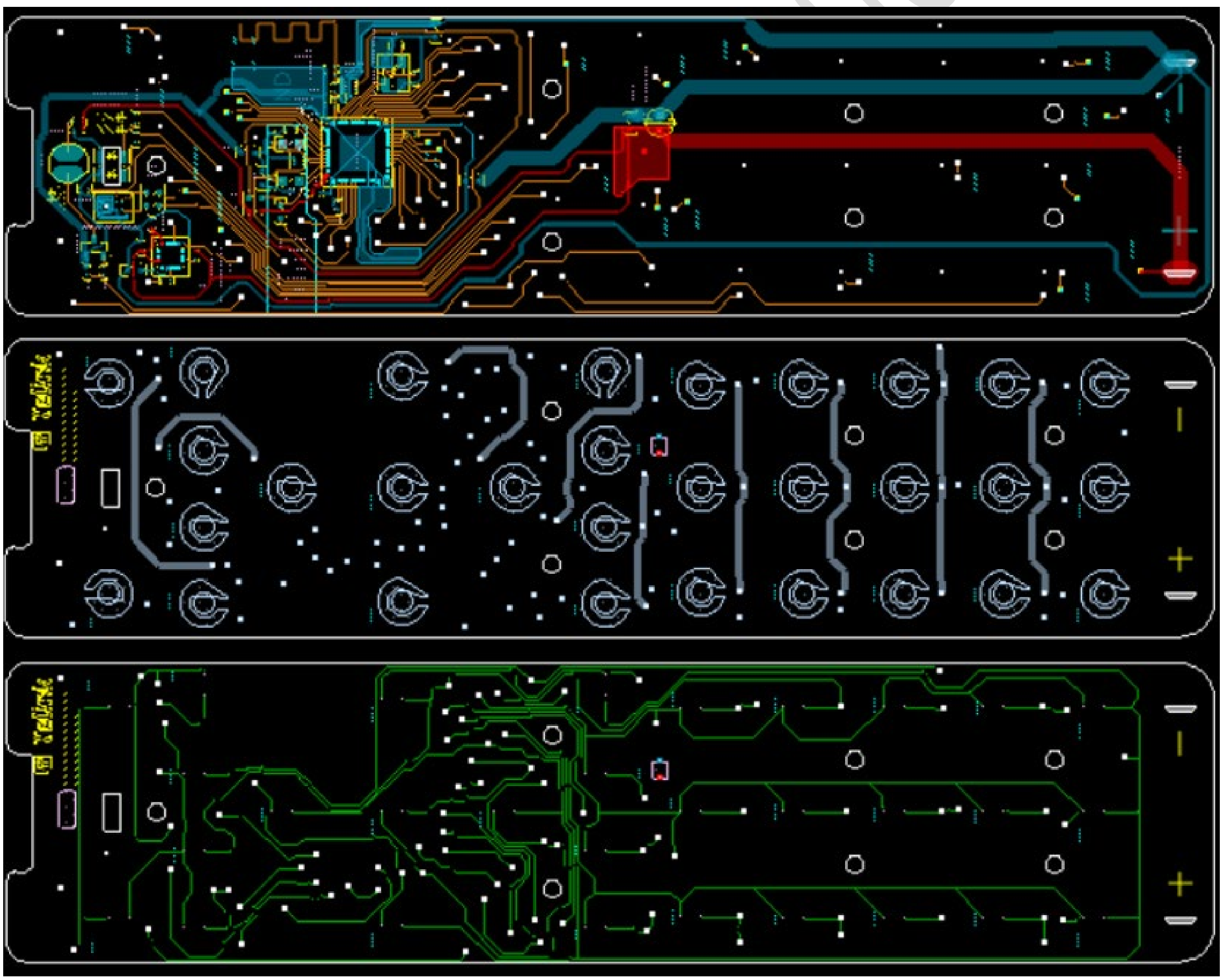
2.2.2 Component and copper wire layer + carbon film and copper wire layer

One layer of this double-layer board is used to place components and copper wire routing, and the other layer is for carbon film routing + copper wire routing. For example, in a remote control design, we place the components on one layer and the keys on the other layer. The keys need to be designed as carbon film keys and the keys routing is connected to the component layer via carbon film via holes. When there are many components and the routing is complex, if the design shown in 2.2.1 cannot be completed routing, then in addition to the carbon film routing on another layer, it is necessary to add copper routing and connect the component side routing through the carbon film via holes. Note that carbon film vias are chosen for cost concerns.

The board shown below can be designed as this type of double-layer board.

ductor

Figure 2-3 Double layer routing + carbon film routing



3. Key Points of FR1 Board Design

3.1 Board layer

3.1.1 Board thickness selection

In order to reduce cost, FR1 or CEM-1 boards are generally used to produce PCB boards.

- The thickness of FR1 board is recommended to be 1.6mm.
- The thickness of CEM-1 board is recommended to be 1.2mm or 1.0mm.

Note:

1) *CEM-1 is more suitable for making thinner boards than FR1, and CEM-1 is less likely to warp boards than FR1 over wave soldering.*
2) *Whether FR1 or CEM-1 is used, the rules and notes for PCB design are the same.*

3.1.2 Introduction of board structure

In general, FR1 circuit board is single surface board, however, we need create another layer in addition to the Top layer and Bottom layer, called the carbon film layer. As shown in the figure below, the Key layer is the carbon film layer.

Figure 3-1 Stack structure

Telink Semiconductor

3.2 Carbon film routing

Carbon film routing is generally 1mm width routing, and the process of carbon film via hole is shown in the table below.

Table 3-1 Design of carbon film via hole

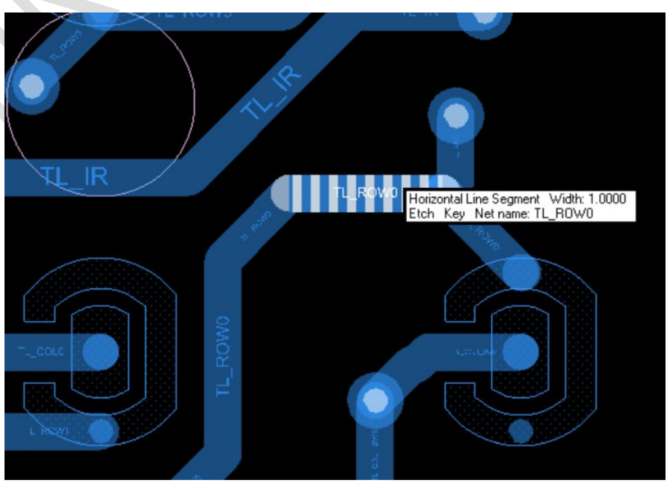
No.	Item	Requirement and description	Process limit capabilities, which can lead to reduced yields or increased production difficulties
109	Carbon grouting board	Preferred FR1 thickness 1.6mm board	When using 1.2mm or 1.0mm thickness for card type remote control, it is recommended to use CEM-1 board to ensure the strength and avoid board warping.
110	Drilling diameter of carbon grouting hole	0.7mm	
111	Diameter of copper plate for carbon grouting hole	1.4mm	Copper plates at least 0.35mm larger than the hole on one side, design as 15mm or 1.6mm if there is enough space.
112	Diameter of weld-resistant openings	1.3mm	Copper plate with green oil on one side 0.05mm
113	Diameter of insulated opening	1.6mm	0.1mm greater than copper plate on one side when insulated
114	Diameter of the carbon film on the surface of the grouting hole	1.8mm	When there is no protective oil, in principle the carbon film is required to be 0.2mm larger on one side of the copper plate to avoid revealing copper. When it is not possible to meet the carbon film single-side is 0.2mm larger than copper, set the carbon film and copper have the same width or single-side is 0.05mm larger, but in this case we need to print protective oil.
115	Diameter of the carbon film on the skimmer surface	1.8mm	Minimum 1.6mm; when there is no protective oil, in principle, it requires that one side carbon film is 0.3mm larger than copper plate to avoid revealing copper
116	Diameter of carbon hole protection oil	2.3mm	Minimum 2.0mm, one side is larger than copper 0.2mm at least
117	Width of back carbon grouting hole edge from carbon bridge edge	⩾2.3mm	Optimum 2.5 mm. The back carbon coverage width is at least 2.1 mm due to oil spillage of 0.65 mm on one side of the hole.

Telink Semiconductor

No.	Item	Requirement and description	Process limit capabilities, which can lead to reduced yields or increased production difficulties
118	Distance between the edges of adjacent copper plates on different networks	≥1.0mm	At least 0.9mm, distance from the edge of the carbon plate to the edge of the carbon plate is at least 0.5mm.
119	Distance between the edges of adjacent copper plates on the same network	≥0.5mm	At least 0.4mm: the distance between the copper wire and the adjacent non-connecting carbon surface is 0.4mm or more (if the joint is a carbon point to be filled through, it must be kept at 0.7mm or more).
120	Distance between the edge of the carbon plate and the edge of the green oil covered wire	≥0.5mm	The principle is to avoid short circuits caused by carbon oil seepage in case of pinholes in the single layer of green oil; the edge of the carbon plate is at least 0.1mm from the insulation layer covering the wire when there is an insulation layer underneath the carbon film.
121	Carbon hole resistance	<100 ohms/hole	
122	Carbon hole reliability design	Designed for parallel connection of two holes when space is available	Reduce open circuit scrap and reduce circuit resistance.

Carbon film wire and carbon film via hole impedance are relatively high, so the carbon film wire should not be less than 1mm. Due to the manufacturing process for carbon film via hole, the carbon film wires are easy to short circuit with other holes and wires, so the routing around the carbon film hole should avoid the hole more than 1mm, as shown in the figure below.

Figure 3-2 Carbon film routing



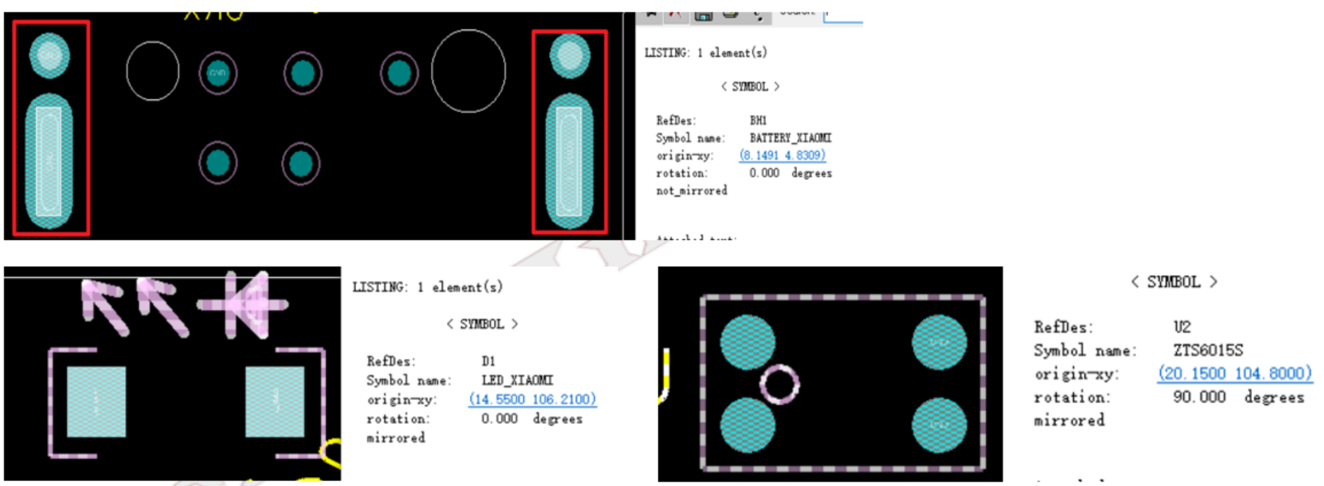
Telink Semiconductor

4. Layout Regulations

4.1 Package

The package forms of battery, LED, and MIC are shown in the figure below.

Figure 4-1 Package forms



4.2 Solder pads and via holes

The PCB package design of Telink IC is as shown in Figure 4-2.

The carbon film via hole is shown in Figure 4-3, the hole diameter is greater than 0.7mm, outer diameter 1.4mm. The air gap between carbon film hole and copper wire is 2.5mm or more.

Figure 4-2 Package design for Telink IC

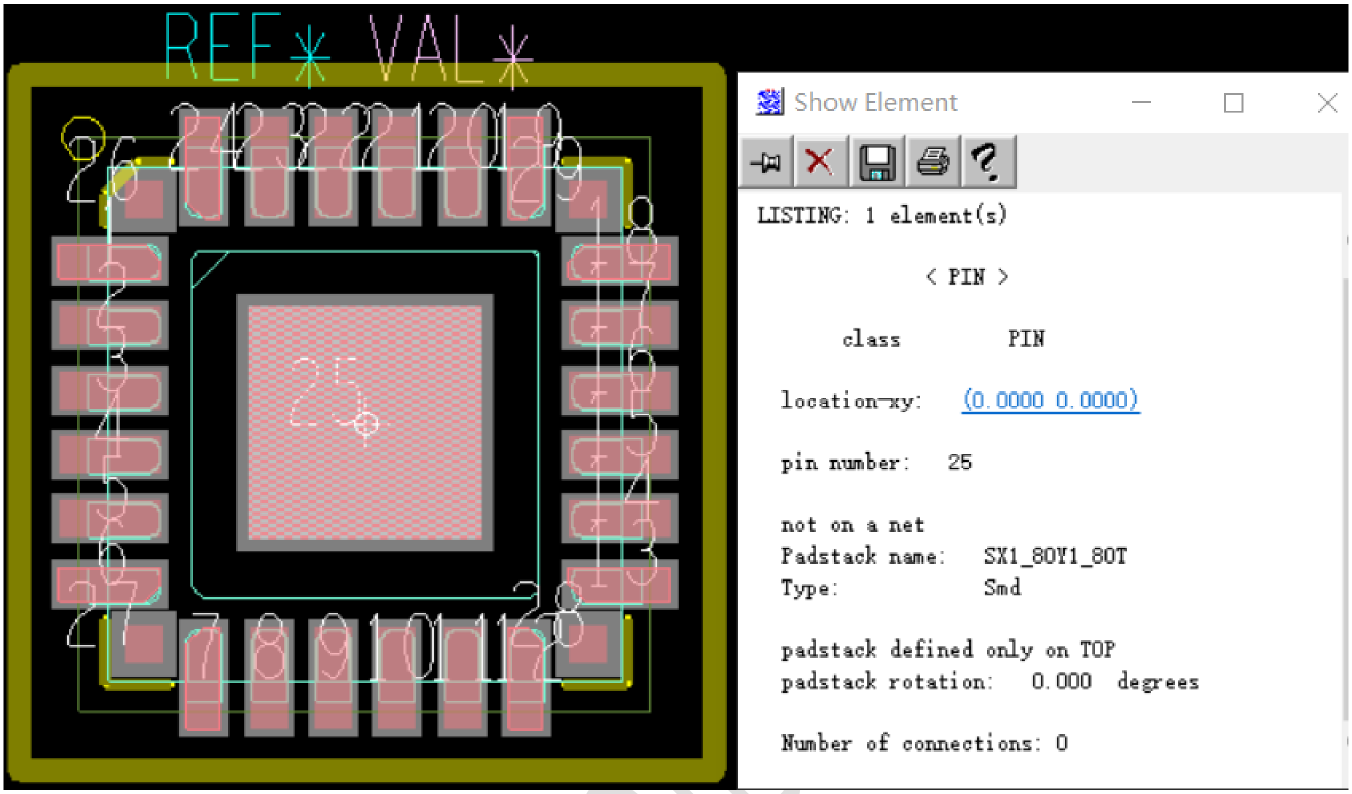
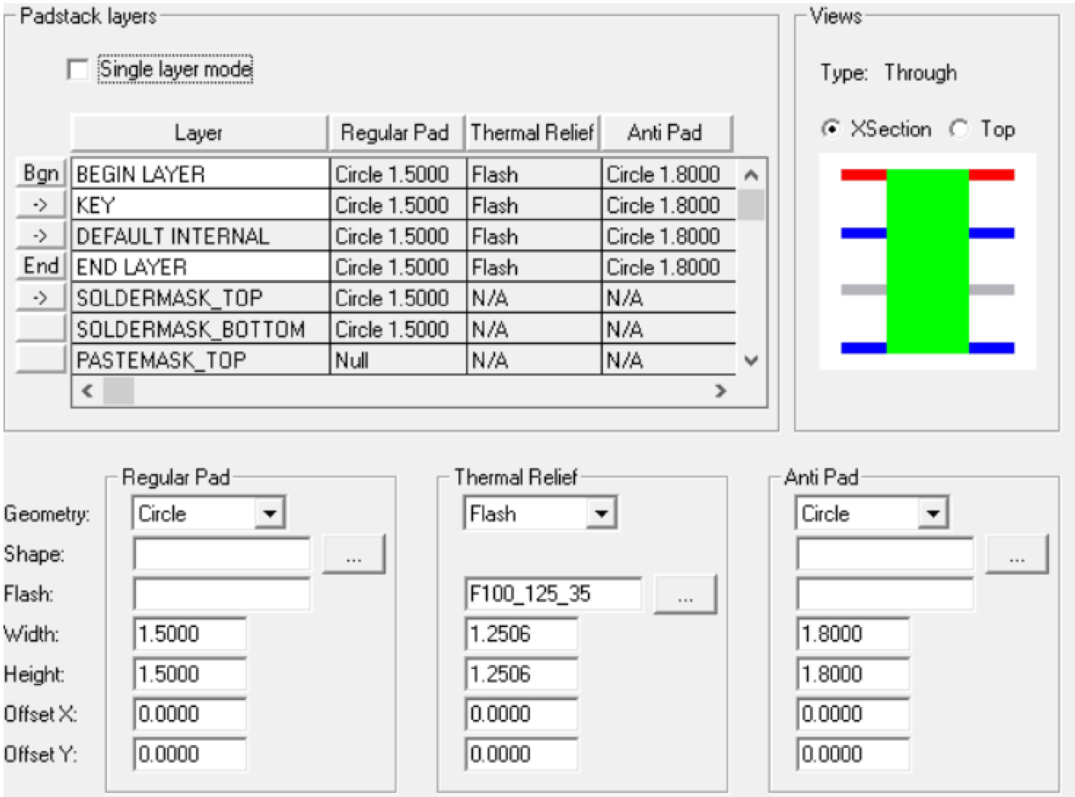


Figure 4-3 Via hole on carbon film

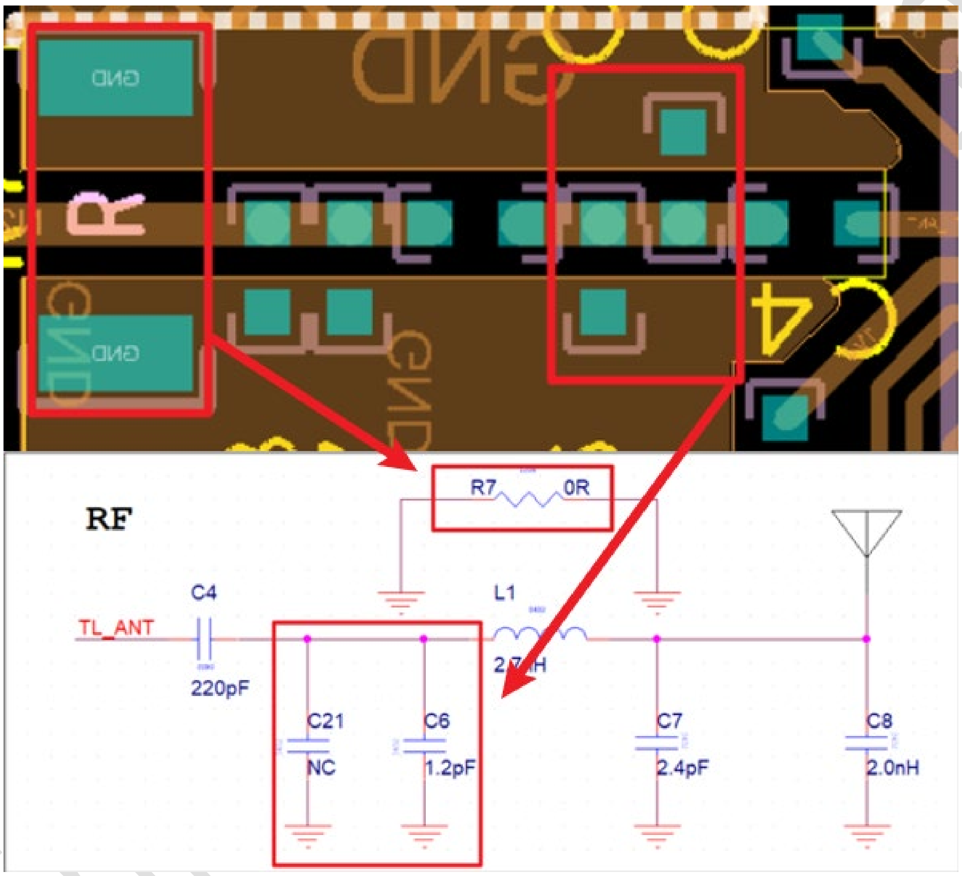


4.3 Notes

- First of all, the layout should consider the placement of the chip, based on the antenna, and reserve enough space for the antenna first, so as to determine the location of the chip.
- According to the board structure requirement, place the components (battery holder, keys, MIC, and etc., while paying attention that do not place components in the location column area).
- The components used for RF matching are close to the RF pins. The ANT circuit is isolated from the chip and other circuits by ground to avoid signal crosstalk. The capacitors of the LC filter are placed on both sides of the RF routing as shown in Figure 4-4. The purpose of placing on both sides of the RF routing is to allow better harmonic regulation. It is best to place a 0R (0 ohm) resistor at its RF end to improve ground return on both sides of the RF circuit, as shown in Figure 4-4.
- The placement of the various capacitors needs to be arranged as suggested below, as shown in Figure 4-5.

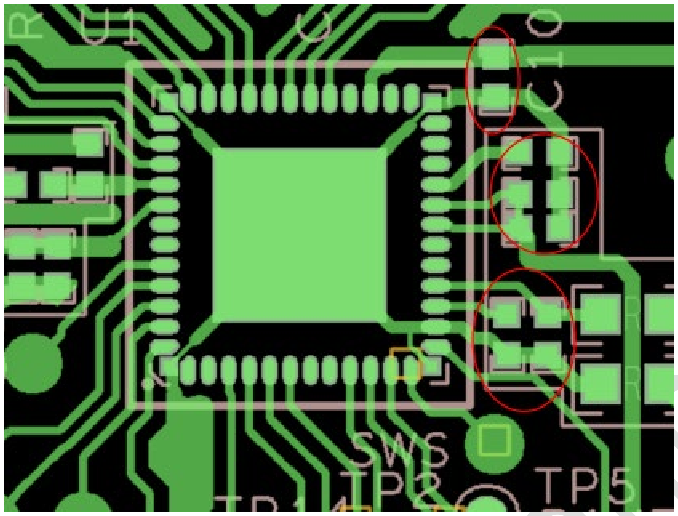
- Before supplying power to the chip, the power must be filtered through a capacitor. The power supply route is: the positive end of the battery spring tab → the filter capacitor → the power pin of the chip.
- The filter capacitor for the chip's power pin should be placed as close as possible to the corresponding pin of the chip.

Figure 4-4 Layout for RF circuit



Telink

Figure 4-5 Layout for power capacitors



Telink Semiconductor

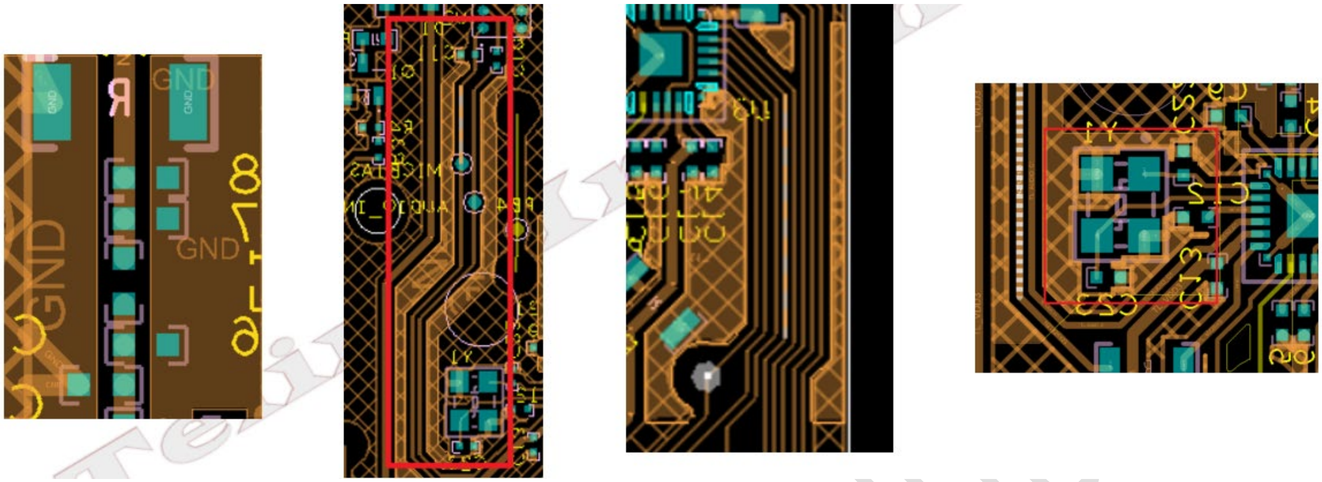
5. Routing Notes

Please be noted of the following routing points.

- Prioritize the power and RF circuit routings, power wires should be as thick as possible (0.5mm ~ 1mm), as close as possible to the power supply components or chip end in the use of star alignment respectively power supply.
- Then comes the normal routing, prioritize to route audio lines, and audio lines should have complete copper on both sides, as shown in Figure 5-1.
- For via hole, carbon film hole is needed, pay attention that put the carbon film hole as far as possible with the routing and other carbon film hole, at least 1mm, to prevent short circuit.
- Keep the width of the carbon film routing at 1mm or more to reduce carbon film impedance.
- The ground is the main focus for FR1 board design.
  - Due to the high current jumps during RF operation, the smaller the GND impedance, the better, to help reduce RF noise. For single-layer board, we should connect each component's ground with wires after the power routing is completed, giving priority to ensuring that the ground line can go through, and adding the 0R cross-line resistor at locations where it has blocks.
  - Ground wire should be as thick as 1mm and above, and routed in a ground shape.
  - Routing does not require excessive division of the ground plane, that is, the same direction routing is together, and the routing distance should not be too long to affect the ground backflow. Priority to ensure that the chip center ground and the power supply negative ground has a good ground connection.
- The crystal should be grounded as much as possible, and the RF components should be completely grounded and covered with solid copper, as shown in Figure 5-1.
- Avoid large areas of solid copper, in the wave soldering process, abnormal heat dissipation, the board is easy to bulge. When encountering large areas of copper coverage, choose a mesh structure to facilitate heat dissipation. For key circuits, such as RF, crystal, audio lines, and etc., the complete copper coverage can be used directly. In addition, for some areas of narrow space, the complete copper should be laid out to strengthen the ground reflow, as shown in Figure 5-1.

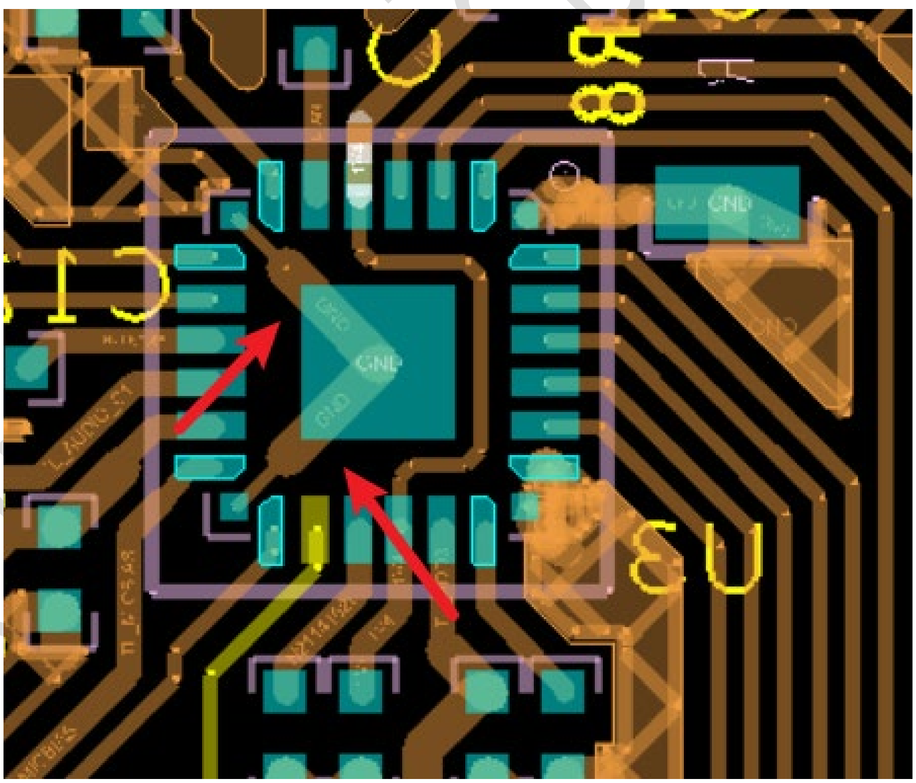
Telink Semiconductor

Figure 5-1 Routing example 1



- In order to make a good connection between the chip and the ground and improve the RF performance, it can be connected to the system ground through the chip's four corner ground, as shown in the location of the red arrow in Figure 5-2.

Figure 5-2 Routing example 2



- After the overall routing is complete, for the copper plates where the ground plane is disconnected from the chip ground or power ground due to the routing, we should use 1206 package resistor to across the line to make it connected with the chip ground or power ground copper, as shown in Figure 5-3.

Figure 5-3 Routing example 3