Application Note:
PCB Layout Guideline

AN-PCBL-GE1

Ver 1.0

2013/08/19

TELINK SEMICONDUCTOR

Brief:

This application note is aimed at offering guide and suggestions about layout and design to develop a good PCB.

The document mainly introduces structure design, antenna layout and design, design of power supply and ground, as well as design of audio, MIC, crystal oscillator and RF.

Published by
Telink Semiconductor

.fŘƎ oΣ мрллλ ½dzŎK2yƎÎKʎ wŘΣ

½KI·yƎ2ʎI·yƎ I ʎπ¢ŜŎK tI·ℕ]Σ Shanghai, China

Legal Disclaimer

Telink Semiconductor reserves the right to make changes without further notice to any products herein to improve reliability, function or design. Telink Semiconductor disclaims any and all liability for any errors, inaccuracies or incompleteness contained herein or in any other disclosure relating to any product.

Telink Semiconductor does not assume any liability arising out of the application or use of any product or circuit described herein; neither does it convey any license under its patent rights, nor the rights of others

The products shown herein are not designed for use in medical, life-saving, or life-sustaining applications. Customers using or selling Telink Semiconductor products not expressly indicated for use in such applications do so entirely at their own risk and agree to fully indemnify Telink Semiconductor for any damages arising or resulting from such use or sale.

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:

ŭŜfλy]ŎyãI·fŜă X ŭŜfλy]πăŜYʎΦŎ2Y

ŭŜfλy]ŎyãdzLJLJ2ℕŭ X ŭŜfλy]πăŜYʎΦŎ2Y

Revision History

Version	Major Changes	Date	Author
1.0	Initial version: AN-PCBL-GE1	2013/8	W.W.X., L.X, Cynthia

1 Table of contents

2 Table of Figures

3 Table of Tables

1 Brief introduction

The design of PCB is the basis of product design, for a well-designed PCB will be good for later debugging and production.

To offer guide and suggestions for users to develop a good PCB, this application note introduces structure design, antenna layout and design, design of power supply and ground, as well as design of audio, MIC, crystal oscillator and RF.

2 Product structure

For PCB design, details about structure should be considered as an emphasis, such as wiring keepout, components keepout, height limited area, etc. Without considering the details, even a PCB design with good performance may conflict with the structure after assembling components, so rework does occur.

2.1 Route Keepin

The cut PCB size will be slightly bigger than needed if there is no route keeepin or not enough shrink.

To obtain suitable PCB size, it's recommended to establish a route keepin area with above 0.3mm shrink for board frame edge.

2.2 Package Keepout

It's noted that location holes for location columns, screw caps or connectors should be at least 1~2mm far away from other components.

2.3 Height limited area

Height of components must satisfy the requirement of height limited area.

3 Antenna

3.1 Layout and design principles

Communication distance of a wireless product is closely linked with antenna performance. Antenna placement is very important for it directly influence its performance.

3.1.1 Metal components

As a radiation unit, antenna will be blocked from transmitting and receiving signals by metal components nearby.

Generally antenna placement should be above 5mm away from any metal componet, such as metal screw, plate frame, etc.

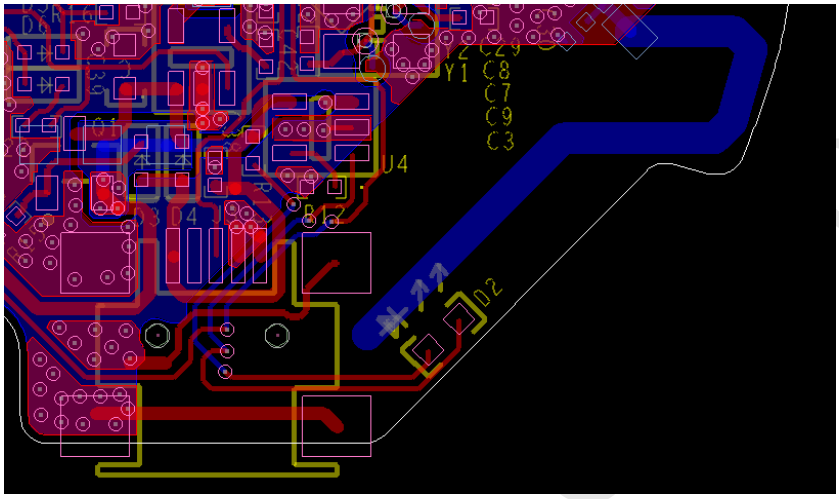

Figure 1 Bad design example: antenna placement is close to USB socket

3.1.2 Antenna orientation

Do not place antenna towards speaker, head or metal shell in order not to influence antenna transmission and reception.

It's suggested to adopt sky-oriented placement, as shown in .

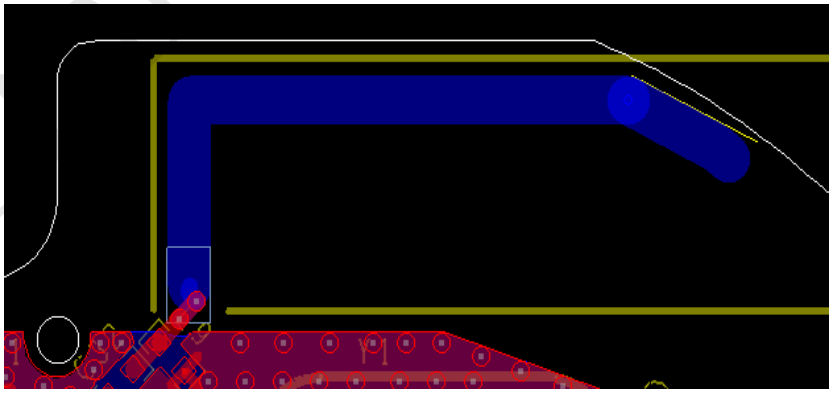

Figure 2 Good design example: sky-oriented antenna placement

3.1.3 Placing angle of dual antenna

In order to decrease dead angle of the receiving end, the headphone end generally adopts dual antenna design for wireless headphone application. It's noted that the dual antenna should be placed vertically.

If dual PCB antenna design is unfeasible due to product structure, consider employing the design of single PCB antenna plus single-wire antenna. But keep in mind that vertical placing is also required.

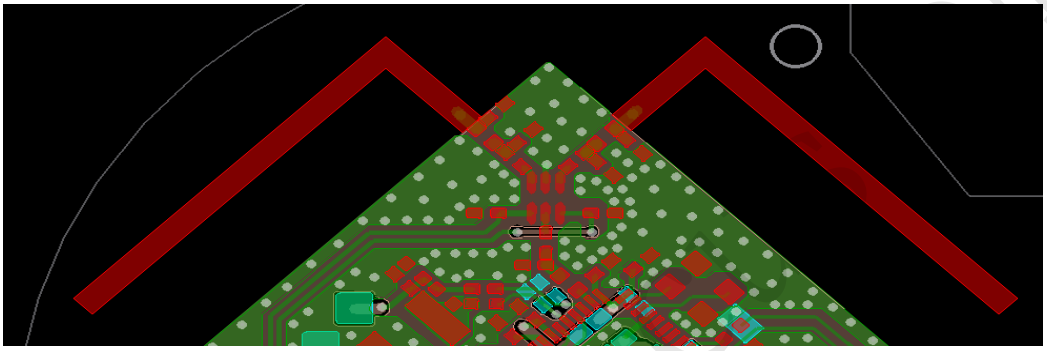

Figure 3 Good design example: dual antenna with vertical placement

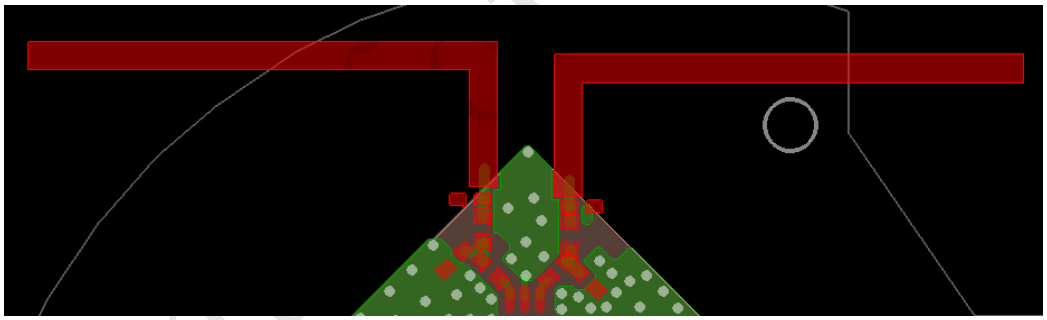

Figure 4 Bad design example: dual antenna with non-vertical placement

3.1.4 Notes of antenna layout

It’s prohibited to directly expose route and pad to antenna.

Do drill a row of vias along antenna surface parallel to GND.

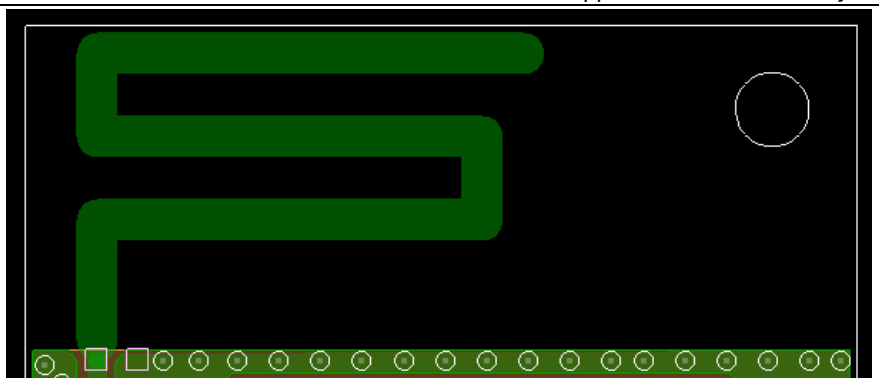

Figure 5 Good design example: uniform VIA placement

Figure 6 Bad design example: route exposed to antenna

3.1.5 Battery

For battery also influences antenna performance, antenna location should be 3~5mm away from battery at least.

3.2 PCB antenna design

3.2.1 Route width

For route width less than 0.5mm will cause much loss, it's generally recommended to adopt 0.5mm~2mm route width: 0.5mm for dongle application, and 1mm for applications including headphone, mouse, keyboard, etc.

3.2.2 Route spacing

It's recommended to adopt route spacing that is integral multiple of 0.5mm to avoid electromagnetic coupling and ensure effective length: 0.5mm or 1mm for dongle application, and 1mm for applications including headphone, mouse, keyboard,

etc.

3.2.3 Spacing from GND

It’s recommended to keep antenna away from GND for 3mm at least with 0.5mm step.

Due to size limitation, the spacing should be as far as possible for dongle application.

3.2.4 Antenna area

Neither front side nor back side of antenna area should be covered by GND, and GND should be away from antenna area as far as possible.

3.2.5 GND

It's noted to keep the integrity of back GND for the PCB design in order to enhance antenna radiation efficiency.

3.2.6 Copper thickness

It’s recommended to adopt 1OZ thickness for antenna copper coat.

3.2.7 Antenna length

It’s recommended to adopt 20~40mm antenna length: about 30mm for dongle application; about 20mm (less than 30mm) for applications including headphone, mouse, keyboard, etc.

It’s noted that the smaller GND is, the longer antenna should be.

3.2.8 Location of matching components

Matching components should be placed in the boundary of antenna and GND, and neither shrink nor expansion is permitted.

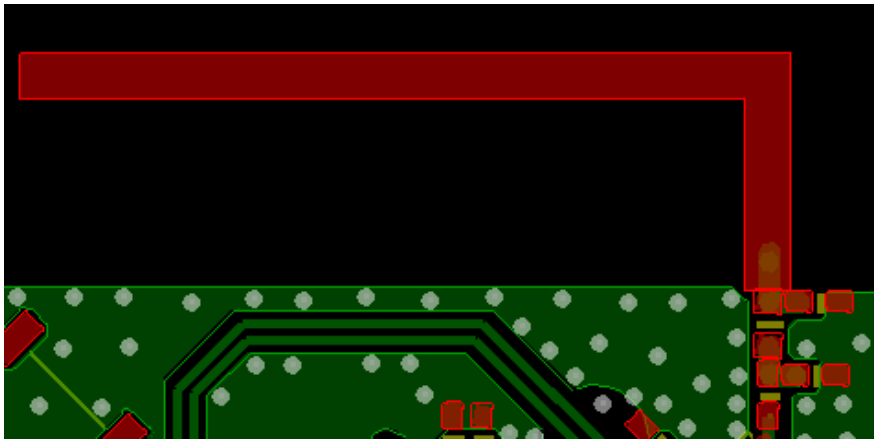

Figure 7 Good design example: matching components located in the boundary

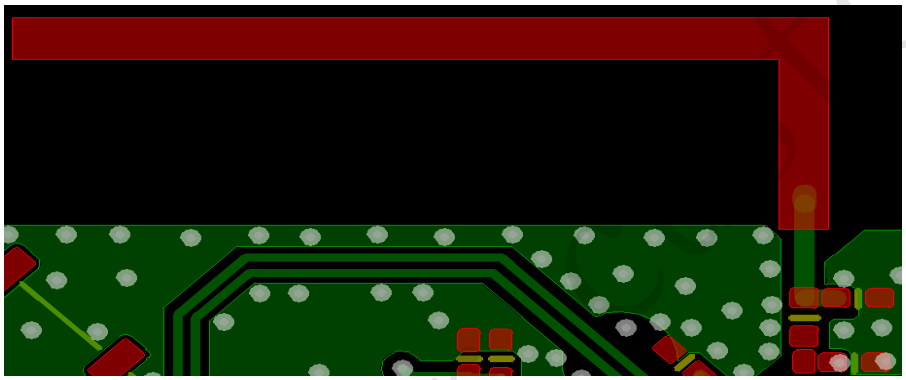

Figure 8 Bad design example: matching components shrink into GND

3.2.9 Common antennas

Common PCB antennas include mono ant, ILA ant, IFA ant and PIFA ant, for which there is no essential difference.

It's recommended to adopt high-efficiency mono ant for debugging.

4 Power and ground

4.1 Brief introduction

Well-designed power is very important for PCB design for it can decrease the probability of system instability. While bad power design may brings many unexpected problems, such as system instability, audio noise, local overheating, system crash, etc.

Generally, power design of serial connection is largely adopted due to PCB size and route. This solution will cause severe mutual interference, which can only satisfy

those non-demanding systems.

RF SoC system developed by Telink Semiconductor integrates modules which are demanding of power, such as RF, audio, etc, and low-cost dual layer structure is also adopted for PCB. For audio applications, power design of star connection must be adopted, meanwhile try to decrease crosstalk and noise via current loopback path design.

Ideal ground design should be equipotential with one layer integral ground plane at least; otherwise voltage difference would lead to noise interference. For most PCB of Telink is 2-layer board, it's noted to keep an integral and clean plane for ground. Ground current loopback path should be kept unimpeded to avoid ground current from influencing modules sensitive to noise (such as audio, mic, crystal oscillator, etc).

4.2 Design principles

Power and ground design can't be separated as individual element, and the balance is very important.

Please refer to the principles listed as follows for power and ground design:

1) Decoupling caps should be placed near pins of the chip.

2) Power cord should be thick and short; it's recommended to adopt power cord with 0.5mm~0.6mm diameter normally. Consider adopting diameter above 0.8mm if power cord is long.

3) Ground current loopback path should be wide and ground current must be filtered by LDO capacitor before flowing back to the battery.

4) Power should pass by RF, analog part and digital part in turn, so that digital part won't interfere in RF and analog part.

5) Input of power should be away from its output as far as possible, so that unfiltered input power won't interfere in filtered output power.

6) To avoid mutual interference, modules such as RF, crystal oscillator, audio, mic, USB, clock and flash should be isolated with GND and protected by vias as much as possible.

7) As for audio related modules including audio, Mic, Line-in and line-out, it's noted to keep the integrity of ground plane.

4.3 TLSR8870 analysis

Table 1 Power supply pins

Product Name	TLSR8870ET64	
Package Type	QFN8X8-64PIN-0.4MMPITCH	
Pin Name	**Pin No.**	**Pin description**
DVDD3	17	I/O supply
AVDD3	27	Analog supply for audio DAC
U_VDD3	30	3.3V USB PHY supply
AVDD3	46	Analog supply for analog inputs
AVDD3	49	Analog supply
RFVDD3	50	Analog supply for RF
AVDD3	57	Analog supply for audio ADC

Integrating audio and mic module, the TLSR8870 system is demanding of power supply. Except for basic principles in section 4.2, requirements as follows should be noted:

(1) PIN27 and PIN57 are power supply pins for audio and mic respectively, with direct route from power IC output capacitor. Other pins can be connected in turn according to point 4) in section 4.2.

(2) If it's difficult to route, PIN57 should be ensured first.

(3) External audio PA or codec is needed for high requirement of audio quality; audio PA or codec should be directly routed from power IC output capacitor alone. PIN27 can be connected with other power supply pins in turn according to point 4) in section 4.2.

4.4 TLSR8868 analysis

Table 2 TLSR8868 power supply pins

Product Name	TLSR8868ET32		TLSR8868ET48	
Package Type	QFN5X5-32PIN-0.5MMPITCH		QFN7X7-48PIN-0.5MMPITCH	
Pin Name	**Pin No.**	**Pin description**	**Pin No.**	**Pin description**

DVDD3	8	I/O supply	13	I/O supply
AVDD3	15	Analog supply for audio DAC	23	Analog supply for audio DAC
AVDD3			33	Analog supply for analog inputs
AVDD3	24	Analog supply	36	Analog supply
RFVDD3	25	Analog supply for RF	37	Analog supply for RF
AVDD3	31	Analog supply for audio ADC	43	Analog supply for audio ADC

Integrating RF, audio and mic module, the TLSR8868 is mainly applied to audio receiving end, which is demanding of power supply. Except for basic principles in section 4.2, requirements as follows should be noted:

(1) PIN15 (TLSR8868et32) and PIN23 (TLSR8868et48) are power supply pins for audio; while PIN31 (TLSR8868et32) and PIN43 (TLSR8868et48) are power supply pins for mic. All of the four pins should be routed directly from power IC output capacitor, and other pins can be connected in turn according to point 4) in section 4.2.

(2) If it's difficult to route, PIN31 (TLSR8868et32) and PIN43 (TLSR8868et48) should be ensured first.

(3) External audio PA or codec is needed for high requirement of audio quality; audio PA or codec should be supplied with power alone with direct route from power IC output capacitor. PIN15 (TLSR8868et32)/PIN23 (TLSR8868et48) can be connected with other power supply pins in turn according to point 4) in section 4.2.

4.5 TLSR8869 analysis

Table 3 TLSR8869 power supply pins

Product Name	TlSR8869ET32	
Package Name	QFN5X5-32PIN-0.5MMPITCH	
Pin Name	**Pin No.**	**Pin description**
DVDD3	9	I/O supply

U_VDD3	15	3.3V USB PHY supply
AVDD3	21	Analog supply for line-in I/O
AVDD3	24	Analog supply
RFVDD3	25	Analog supply for RF
AVDD3	31	analog supply for audio ADC

The TLSR8869 is mainly applied to dongle and line-in.

For dongle application, the TLSR8869 is used to transmit RF signal, without relevance to audio. When basic principles are followed strictly, power design of serial connection type is feasible; each power supply pin is connected in turn.

Integrating RF and line-in, the TLSR8869 is demanding of power supply. Except for basic principles in section 4.2, requirements as follows should be noted:

(1) PIN21 and PIN31 are power supply pins for line-in. The two pins and Line-in bias voltage point should be routed directly from power IC output capacitor and supplied with power respectively. Other pins can be connected in turn according to point 4) in section 4.2. This application adopts 4-layer PCB design with star type route for power layer. Protection of GND should be noted.

(2) External codec is needed for high requirement of audio quality; codec should be supplied with power alone with direct route from power IC output capacitor. PIN21 and PIN31 can be connected in turn with other power supply pins according to point 4) in section 4.2. This application adopts dual-layer PCB design, and protection of GND should be noted for power route.

5 Audio

Telink audio application includes two types: LC output; external audio PA or codec.

For audio application is demanding of power, power supply and ground current loopback path must be noted.

For LC output type, L/R channel inductors should be away from each other as far as possible; try not to place inductors in parallel to avoid mutual coupling.

6 Mic

Telink Mic application includes analog Mic and digital Mic; both are demanding of power and ground. So power supply and ground current loopback path must be noted.

7 Crystal oscillator

As the basic clock reference for the entire system, crystal oscillator must be isolated from other interference signals by GND. Vias should be drilled as much as possible, and try to keep the integrity of ground plane.

8 RF

RF route must be short; it must be designed into 50Ω micro strip, even if it's a little longer. Vias should be drilled evenly along the RF route, and the integrity of ground plane must be noted.