

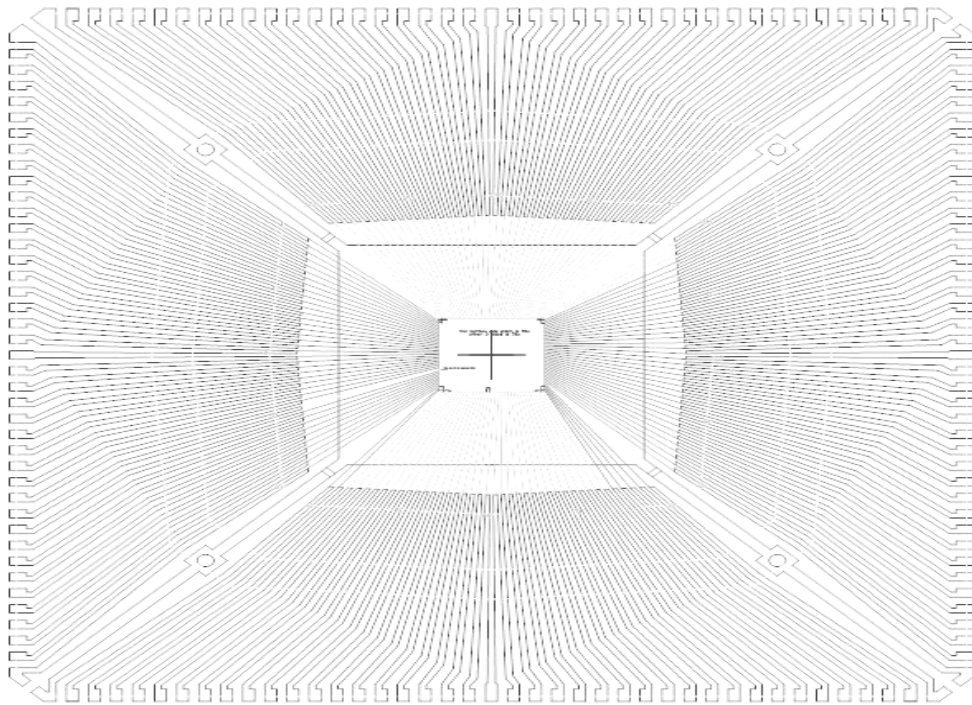


LONTIUM SEMICONDUCTOR CORPORATION

ClearedEdge™ Technology

Layout Guide

龙迅半导体



We produce mixed-signal products for a better digital world!



1.1 PCB层规划

1.1 PCB layer setup

- PCB 的层数越多，性能越容易控制，但是成本也越高。综合考虑，推荐使用4层板，PCB 的叠层结构为S-P-G-S，或者S-G-P-S，其具体定义如下：

第一层(顶层)	->	走线和地
第二层(内层)	->	完整的地层
第三层(内层)	->	走线和电源层
第四层(底层)	->	走线和地

其中地层保持完整，电源层尽可能少走线，或不走线。完整的地层应与高速信号走线层或元件密度较大层相邻，尽可能避免差分等重要信号跨参考平面。

The more layers of PCB, the easier it is to control, but the higher the cost. For comprehensive consideration, it is recommended to use a 4-layer board. The laminated structure of the PCB is S-P-G-S, or S-G-P-S, which is defined as follows:

The first layer(top layer)	->	signal line and ground
The second layer(inner layer)	->	complete ground
The third layer(inner layer)	->	signal line and power
The fourth layer(bottom layer)	->	signal line and ground

the ground layer should be adjacent to the high-speed signal trace layer or a larger component density layer, avoiding important signals such as differences across the reference plane as much as possible.

- 对于两层板，器件布局和走线尽量走在同一面，另外一面尽量保持完整。

For 2 layer board, put component and signal on the same layer if possible, another layer can have complete ground.

1.2 PCB板布局

1.2 PCB layout

- PCB 板的布局按功能模块将模拟信号部分，数字信号部分尽量合理地分开，使相互间的信号耦合最小，尽量减少相互之间的交叉，不干扰其他模块。

Try to separate analog from digital module, to reduce mutual interference and coupling between different signals.

- 电源转换模块尽量远离其他模块，避免转换器件发热或者DCDC 的干扰对其他信号产生影响。音频转换模块或者模拟视频模块应尽量远离高速信号和DCDC 的电感。在DCDC 电源模块布局时，需注意使输入输出环路面积小。

Let power module far away other module, to avoid self-heating and switch-frequency affect other signals. Analog audio and video module should be layout far away high speed digital signal and DC-DC module. And DCDC power supply layout should pay attention to make the input and output loop area small.



- 晶振既是干扰源，也是易受干扰的模块，尽量靠近芯片，远离其他数字信号源和易受干扰的信号。

Crystal is both a source of interference and an easy interfered module, so try to put the crystal close to IC chip and far away digital signal and analog module.

- 去耦电容要尽量靠近芯片电源管脚。

Decoupling capacitors should be close to the IC chip power pin.

- 芯片周围不同电源需用磁珠隔开，且磁珠靠近芯片放置。

Different power supplies on the chip need to be separated by beads, and the magnetic beads are placed close to the chip.

- 特征电阻 (REXT_1%) 需靠近芯片管脚放置，走线适当加粗，该PIN对应芯片内部的BandGap电路。

The characteristic resistor (REXT_1%) needs to be placed close to the chip pin, and the trace is appropriately thickened. The PIN corresponds to the BandGap circuit inside the chip.

- 对于容易遭受到静电的接口，尽量预留出ESD 的位置，ESD 靠近接口布局。

For the port, such as HDMI\DP\USB port, try to set aside the position of ESD component, which close to the interface as possible.

- 在设计许可的条件下，元器件的布局尽可能做到同类元器件按相同的方向排列，相同功能的模块集中在一起布置；相同封装的元器件等距离放置，以便元件贴装、焊接和检测。

If possible, similar components are arranged in the same direction and placed equidistant to each other, and same module components are arranged together for easy soldering and testing.

1.3 PCB走线

1.3 PCB route

- 高速信号线要走差分，并且要做阻抗匹配。HDMI、DVI、MIPI、LVDS、DP等差分阻抗控制为100 ohm，USB2.0及TYPE C控制90 ohm。对于不同板厂来说，因为制作工艺不同，线宽线距可能会有差异。使用板厂推荐的线宽线距。差分对之间、差分与其他信号之间控制3倍线宽间距 (3W)。

High-speed signal lines must be differential and impedance matched. Differential impedance control such as HDMI, DVI, MIPI, LVDS, DP, USB 3.0 is 100 ohm, USB2.0 and TYPE C control 90 ohm. For different board manufacturers, line width and line spacing may vary due to different manufacturing processes. Use the line width recommended by the board factory. Control 3 times line width between differential pairs, between differential and other signals.



- 差分最好不要过孔，差分线需要打过孔时，不要超过两个过孔，最好在过孔处打几个伴随地孔，增加回流路径。

It is best not to use a via for the difference. When the differential line needs to be punched, do not exceed two vias. It is best to make several associated holes at the via to increase the return path.

- 若采用AC模式，阻容件需要靠近芯片放置，且阻容焊盘对应的参考层挖空处理以补偿阻抗，AC阻容件采用0402封装或0201封装。

If the AC mode is used, the RC needs to be placed close to the chip, and the reference layer corresponding to the RC pad is hollowed out to compensate the impedance. The AC RC is packaged in 0402 package or 0201 package.

- 模拟VGA的三根R\G\B信号需要包地处理，注意在包地线上均匀的添加过孔。RGB信号上的75R下拉电阻需靠近芯片放置，RGB信号需控制75欧姆阻抗匹配。

The three R\G\B signals of the analog VGA need to be packaged. Pay attention to the uniform addition of vias on the ground. The 75R pull-down resistor on the RGB signal needs to be placed close to the chip, and the RGB signal needs to control 75 ohm impedance matching.

- 音频信号和模拟视频信号尽量走粗一些，不小于10mil，可以的话，最好包地处理。

Analog audio and video signal should be routed thick if possible, not less than 10mil, and be packaged ground signal as better.

- TTL信号布线尽量短，需要等长处理，CLK信号注意屏蔽抗干扰。

The TTL signal wiring should be as short as possible, and it should be treated with equal length. The CLK signal should be shielded against interference.

- 易受静电干扰的信号如RESET、CBUS等走线时不要太靠近板边，避免静电干扰造成信号突变，引起系统工作不正常。

For the signal of reset\cbus and other clock signal, which is susceptible to interfered by ESD, should be routed far away pcb board edge, to avoid the signal mutation.

- 电源线和地线要遵循环路最小原则，尽量粗且短，大于100mA电流的电源走线不小于20mil。需要打过孔时，最好打两个过孔。

Power and ground loops should follow the principle of minimum, and let the trace thick and short as possible, recommend the power trace width larger than 20mil when it have 100mA current. If need via, let two and more via as better.

- 从降压芯片出来的电源要经过电容后再供下游使用，所以要注意布局及布线路径，尤其是DCDC类电源。

The power supply from the buck chip should pass through the capacitor and then be used downstream. Therefore, pay attention to the layout and wiring path, especially the DCDC



power supply.

- 内电层敷铜时，注意不要出现空白未敷铜区域，避免板子受热时出现变形损坏。
When applying copper to the inner layer, be careful not to blank the uncoated copper area to avoid deformation damage when the board is heated.

- EPAD上建议每隔1mm间距打一个孔，打满整个EPAD。

It is recommended to make a hole every 1mm spacing on the EPAD to fill the entire EPAD.

- PCB上空白区域均匀分布一些地孔，尤其是音、视频信号、晶振这些重要信号，尽量在板边均匀的打些地孔，改善EMI。

Add some ground holes evenly in the blank area of the PCB, especially the important signals such as sound, video signal and crystal oscillator. Try to make some holes on the edge of the board to improve EMI.

**Lontium Semiconductor Proprietary & Confidential**

This document and the information it contains belong to Lontium Semiconductor. Any review, use, dissemination, distribution or copying of this document or its information outside the scope of a signed agreement with Lontium is strictly prohibited.

LONTIUM DISCLAIMS ALL WARRANTIES, EXPRESSED OR IMPLIED, INCLUDING THOSE OF NONINFRINGEMENT, MERCHANTABILITY, TITLE AND FITNESS FOR A PARTICULAR PURPOSE. CUSTOMERS EXPRESSLY ASSUME THEIR OWN RISK IN RELYING ON THIS DOCUMENT.

LONTIUM PRODUCTS ARE NOT DESIGNED OR INTENDED FOR USE IN LIFE SUPPORT APPLIANCES, DEVICES OR SYSTEMS WHERE A MALFUNCTION OF A LONTIUM DEVICE COULD RESULT IN A PERSONAL INJURY OR LOSS OF LIFE.

Lontium assumes no responsibility for any errors in this document, and makes no commitment to update the information contained herein. Lontium reserves the right to change or discontinue this document and the products it describes at any time, without notice. Other than as set forth in a separate, signed, written agreement, Lontium grants the user of this document no right, title or interest in the document, the information it contains or the intellectual property it embodies.

Trademarks

Lontium™ 龙迅™ and ClearedEdge™ is a registered trademark of Lontium Semiconductor. All Other brand names, product names, trademarks, and registered trademarks contained herein are the property of their respective owners.

Website: www.lontiumsemi.com