Test-board prototyping definitions

PROTOTYPING

Prototypes are models developed to simulate the appearance and functionality of a product in development.

Image 01: Some examples of prototypes, (left) a digital magazine (middle) a dream car (right) a manned drone.
Source: [Luxuo](), [Planet CarsZ]() *and* [EHang]() *websites*.

A prototype is a representation of the interface that the user can interact with and still provide information to come up with changes and improvements.

Getting started

When starting a test-board project, you need to gather all the necessary information, such as datasheets, mechanical drawing of components (footprints if available, otherwise you will need to create them) and/or 3D drawing for better visualization of your project.

Start with what you want to develop, based on the features that the Hana Electronics HTLRBL32L chip provides. Analyze the input and output resources (GPIOs, I^{2}C, SPIs) that are available on the chip and if they are enough for your project. After this initial verification we will start the process of prototyping the test-board.

Getting to Know the HTL RBL 32L (LoRa/BLE) Chip

Image 02: Hipertexto20_HTLRBL32L (LoRa/BLE) chip. Source: *Hana Electronics* <https://github.com/htmicron/htlrbl32l>.

HTLRBL32L (LoRa/BLE) Chip Specifications

- LoRaWAN® compliant
- Bluetooth® Low Energy 5.2 compliant
- 32-bit ARM Cortex M0+
- 256 KB flash
- 64 KB RAM
- 7 KB ROM

- STMicroelectronics transceiver BlueNRG-LP
- Semtech transceiver SX1262
- TX output power (LoRa): +21.5dBm
- RX sensitivity (LoRa): -137dBm
- TX output power (BLE): +6.5dBm
- RX sensitivity level (BLE): -96dBm @ 1Mbps, -97dBm @ 125kbps (long range)
- Low power consumption: 0.94uA (sleep with RAM retained)
- LoRaWAN© Frequency plans:
  - EU863-870
  - US902-928
  - AU915-928
  - AS923
  - KR920-923
  - IN865-867
  - RU864-870
  - EU433
  - CN470-510
- Modulation schemes (SX1262): LoRa, FSK, GFSK, MSK, GMSK
- Single power supply: 2.7 V to 3.6 V
- Operating temperature range: -20°C to +75°C
- External antenna
- 13x13x1.5 LGA – 32 pads package

INTERFACES

- 1x DMA controller with 8 channels supporting ADC, SPI, I^2C, USART and LPUART
- 2x SPI
- 1x I2S

- 1x I²C (SMBus/PMBus)
- 1x PDM (digital microphone interface)
- 1x LPUART
- 1x USART (ISO 7816 smartcard mode, IrDA, SPI Master and Modbus)
- 1x independent WDG
- 1x real time clock (RTC)
- 1x independent SysTick
- 1x 16-bit, 6 channel advanced timer
- Up to 19 fast I/Os: 17 of them with wake-up capability; 18 of them 5 V tolerant
- 12-bit ADC with 5 input channels, up to 16 bits with a decimation filter
- Battery monitoring
- Analog watchdog
- Analog Mic I/F with PGA

Applications

- Smart home
- Wireless alarm systems
- Manufacturing
- Agriculture
- Buildings automation
- Intelligent metering
- Intelligent lighting systems

Datasheet, symbol and footprint for HTLRBL32L chip

Hana Electronics has made available the datasheet, symbol, and footprint needed to develop the design for their test-board in a repository on GitLab. Check out the websites below:

- Hana electronics HTLRBL32L chip datasheet:
  https://gitlab.com/ht_advtech/design_team/htlrbl_design

- Hana electronics HTLRBL32L chip symbol to be used in the schematic:
  https://gitlab.com/ht_advtech/design_team/htlrbl_design

- Hana electronics HTLRBL32L chip Footprints:
  https://gitlab.com/ht_advtech/design_team/htlrbl_design

Test Board Prototyping Feature Definitions

Internal Architecture

Hana Electronics' HTLRBL32L chip contains a LoRa and BLE module in its internal architecture that combines the easy access of BLUETOOTH Low Energy (BLE) with the long range capabilities of LoRaWAN® in a 13 mm x 13 mm chip.

Image 03: Chip internal architecture.
Source: *LoRaWAN logo by The Things Network* <https://www.thethingsnetwork.org/docs/lorawan/> *and Bluetooth logo by Bluetooth* <https://www.bluetooth.com/>.

Describing the Circuit

Based on the internal architecture we can then use the HTLRBL32L chip's features using its LoRa module and develop a point-to-point communication. Take a look at the image below.

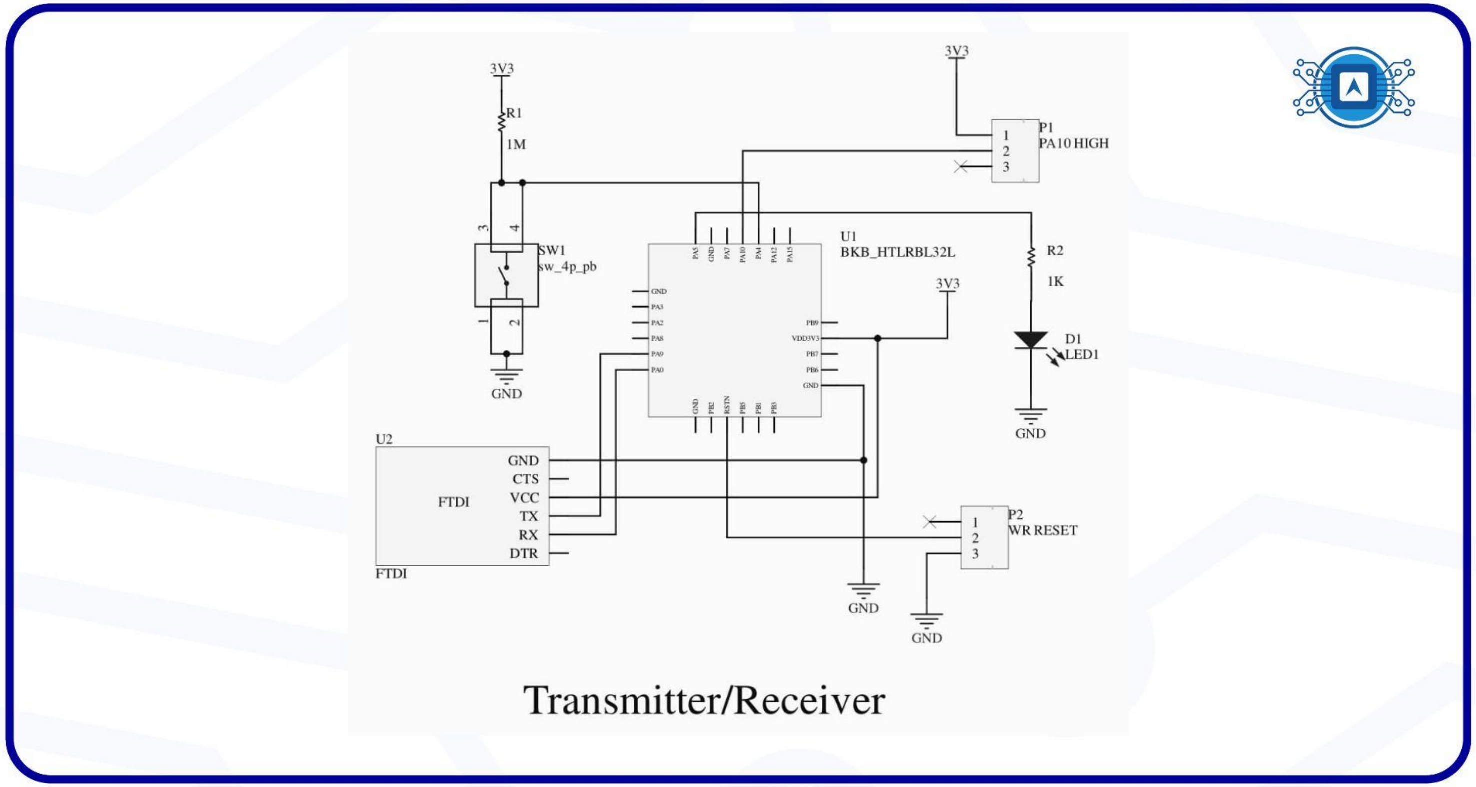

Image 04: Transmitter and Receiver circuit example using Hana chip. Source: *The author*.

The illustration above shows a basic application using the HTLRBL32L chip that contains a transmitter circuit and a receiver circuit, using a LoRa protocol.

The application described is for a circuit that transmits a pulse through switch S1 and the receiver gets this pulse through LoRa communication and displays the incoming signal on LED1 on the receiver.

A circuit to write U1 and U3 is added for each circuit as the test firmware will be written.

The test firmware for this circuit is available at:

https://gitlab.com/ht_advtech/design_team/htlrbl_design

Remember that the HTLRBL32L power supply is 3,3Vdc.

External Supplier Recommendations

Some component suppliers suggest to designers or CAD designers some details in their datasheets regarding the typical circuit to be used for the board layout, in order to provide better performance. An example of this can be seen in the image below:

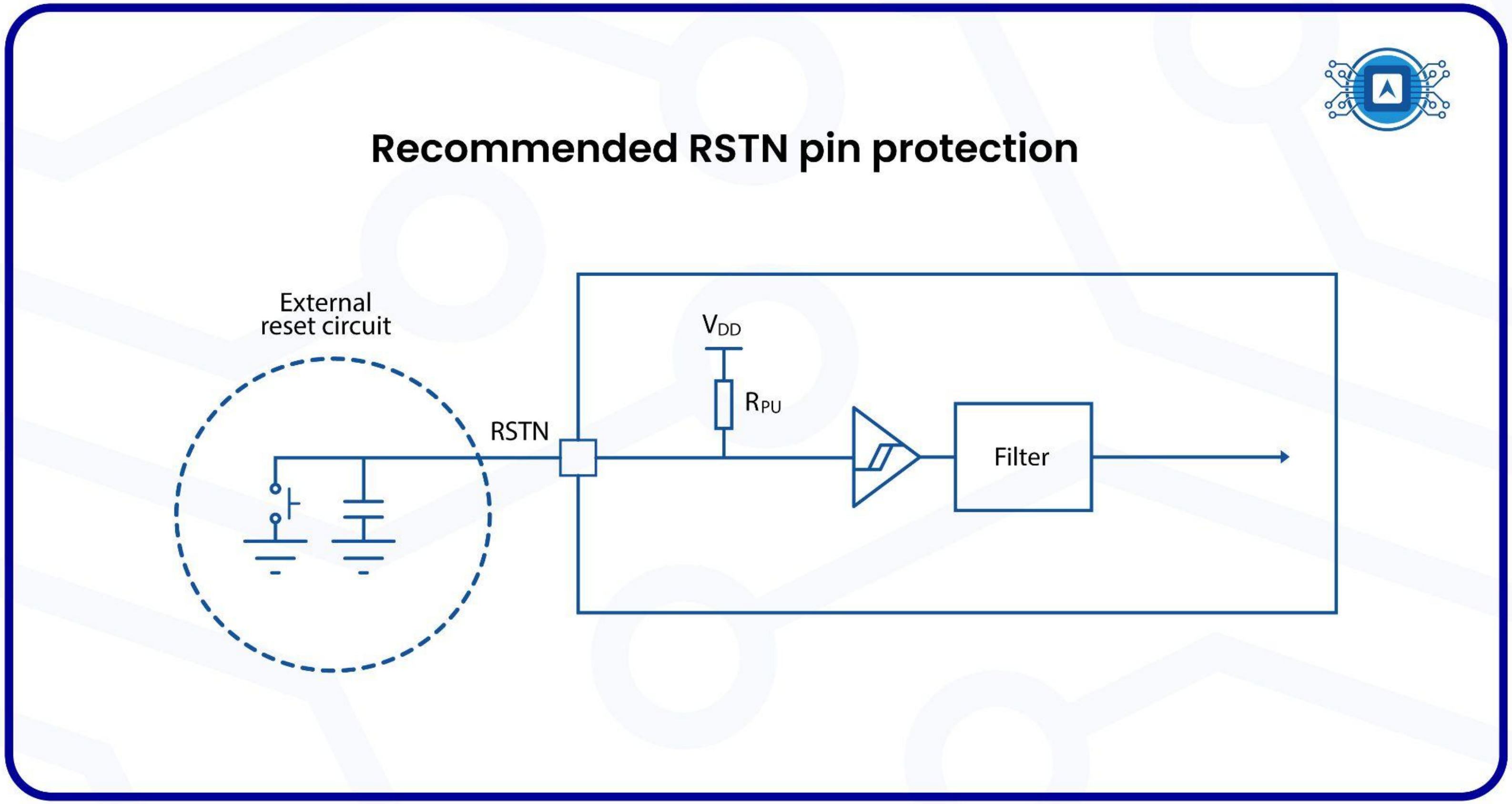

Image 05: Recommended RSTN pin protection.
Source: *STMicroelectronics, 2022, p.51 <https://www.st.com/resource/en/datasheet/bluenrg-lp.pdf>*.

Observe that in image 05 the use of a protection circuit is recommended and the reasons why they should be used are explained.

Review all manufacturers' recommendations before finishing your project.

- Layout recommendations for local prototype without a weld mask

For a prototype circuit that is in development and will go through a validation and testing process, it is recommended to use a simple model without a weld mask. For this we will use only the copper masks (TOP and/or BOTTOM).

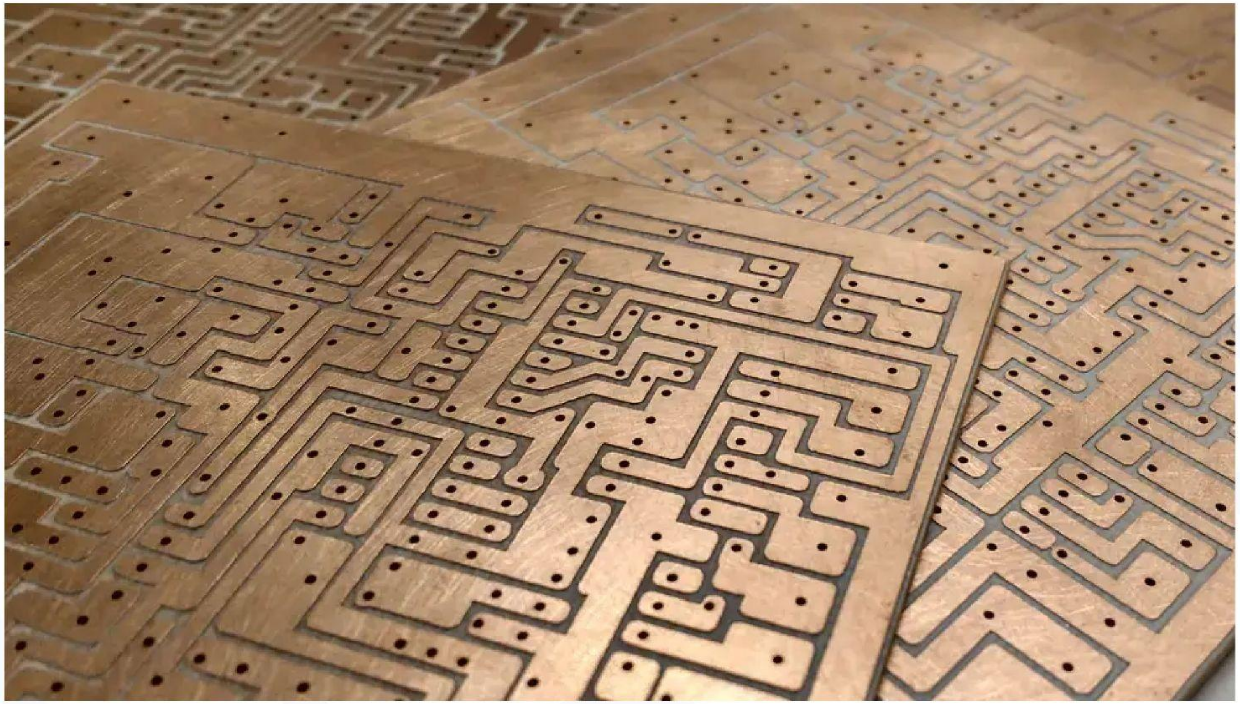

Image 06: PCB without weld mask. Source: *Mazzo <https://www.mazzo.me/en/posts/pcb-milling-mpcnc/>*.

- Recommendations on PCB layout.

The following recommendations were suggested by designer Zachariah Peterson in a post on Altium Design's website, in this article he highlights 5 main rules of good practice for PCB layout [1]

1 - Determine your PCI design rules before layout

When starting a new PCB project, we shouldn't forget the important design rules for both schematics and PCBs. There are some constraints that you can place in your design editor that, if determined early in the project, will eliminate many component offsets, as well as constraints from your PCB manufacturer.

1. The first step is to talk to your PCB manufacturer. A good manufacturer will usually publish their resources online or provide this information in a document. If it is not in an obvious place on their website, send them an email and ask for their specifications. It is recommended that you do this before you start putting in your components.
2. The second is to go over the rules and constraints in your design editor. As you move through the layout process, your design rules will help eliminate most of the design errors that will lead to manufacturing and assembly problems. Once you have defined your design rules, you can begin the arrangement process.

2 - Adjusting the Component's Positioning

The PCB layout process is an art and a science, requiring strategic thinking about the major components available on your board. The goal in component arrangement is to create a board that can be routed easily, ideally with as few layer transitions as possible. In addition, the design must match up to design rules and satisfy mandatory component arrangement. These points can be difficult to balance, but a simple process can help a designer place components that meet these requirements.

a. **Put the required components first**

Often there are components that must be placed in specific locations, sometimes due to mechanical constraints or because of their size. It is recommended to place these components first and lock their position before proceeding to the rest of the layout.

b. **Place large processors and ICs**

Components such as ICs or processors with multiple pins often need to make connections to various components in the design. The central location of these components facilitates trace routing in the PCB layout.

c. **Try to avoid crossroads.**

When components are placed on the PCB layout, the unrouted lines are usually visible. It is best to try to minimize the number of intersections. Each track intersection will require a layer transition across paths. If you can eliminate track intersections with creative component placement, it will be easier to implement the best routing for a PCB layout.

d. **SMD PCB design rules.**

It is recommended to place all surface mount device (SMD) components on the same side of the board. The main reason for this appears during assembly. Each side of the board will require its own passage through the SMD weld line. Therefore, placing all SMDs on one side will help avoid some extra mounting costs.

It is OK to rotate the components to try to eliminate crossover intersections. Try to orient the connected pads so that they face each other, this can help simplify your routing.

If you follow steps (a) and (b), it is much easier to layout the rest of the board without much crosstalk between routes. Also, your board will look modern in layout, where a central processor supplies data to all the other components around the perimeter of a board.

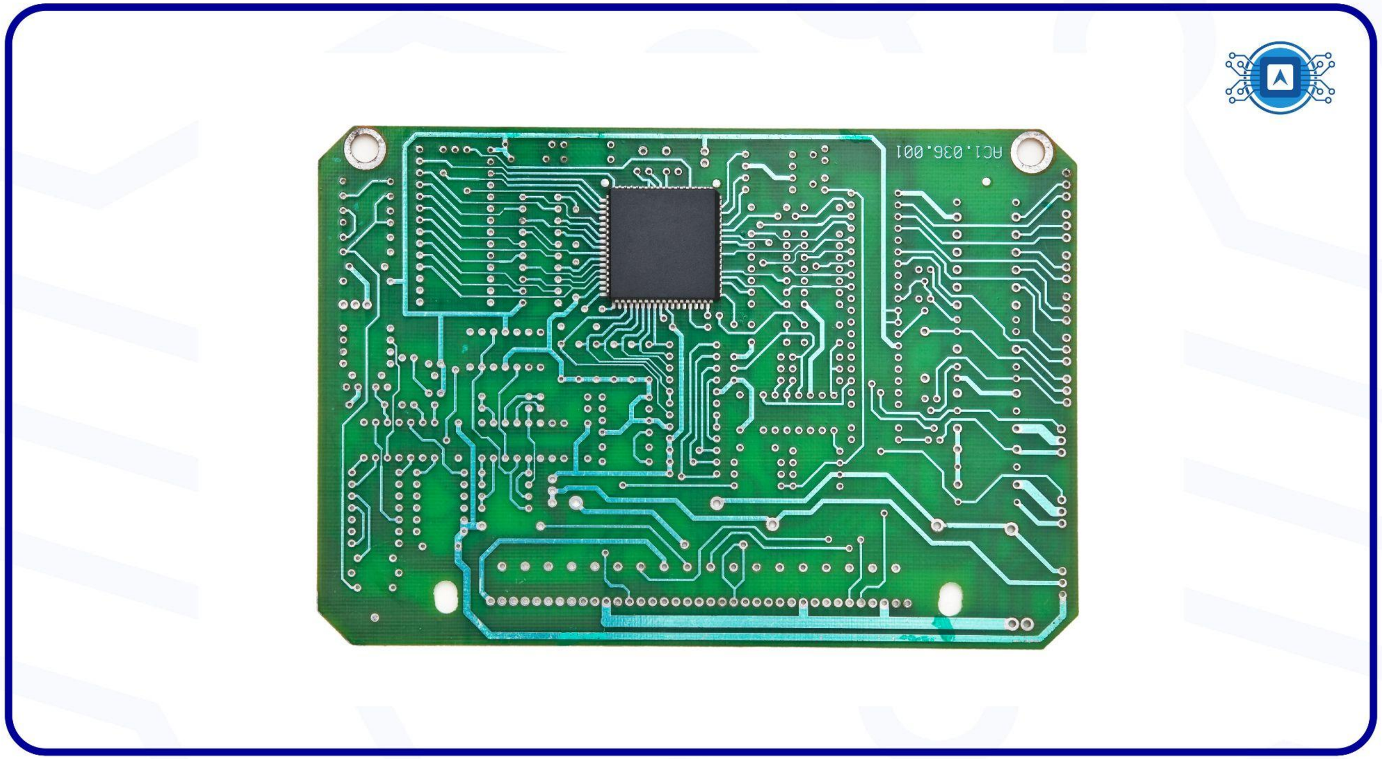

Image 07: PCB with tracks coming out of a microcontroller in the center of the board. Source: *Shutterstock.*

3 - Placing the voltage, ground, and signal terminals

After the component arrangements have been made, it is now time to route the power, ground, and signal traces to ensure that the signals have a clean and smooth routing. Here are some guidelines to remember for this step in your layout process:

1. Where to place the voltage and ground planes?

Usually, voltage and ground are placed on two internal layers. For a 2-layer board, this may not be so easy, so you must put a large ground plane on one layer and then route signals and power traces on the other layer.

With 4-layer circuit boards as well as more layers, you should use grounding planes rather than trying to route grounding traces. For components that need direct connections to power, it is recommended to use common traces for each source if you are not using a voltage plane; make sure you have wide enough traces (100 mils or 2.54mm is good for 5 to 10 A), and avoid sequential voltage lines from one part to another.

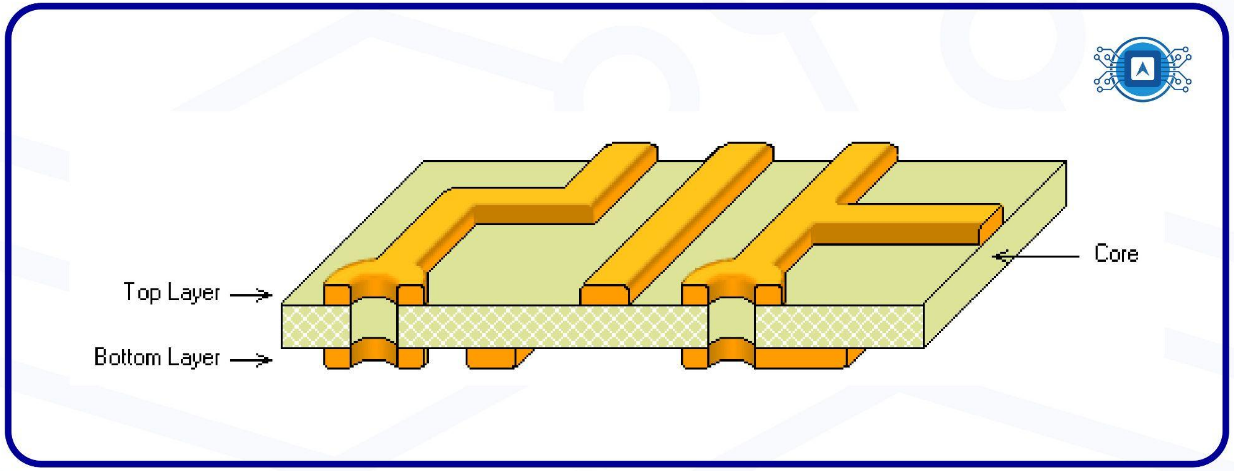

Image 08: Two-layer PCB board, with conductive tracks on the top and bottom sides of the board. Source: Altium <https://www.altium.com/documentation/altium-designer/understanding-printed-circuit-board-makeup?version=17.1>.

Some recommendations declare that the placement of the flat layer should be symmetrical, but this is not strictly necessary for manufacturing. On large boards, this may be necessary to reduce the chances of rupture, but this is not a concern on smaller boards. Focus on power access and grounding, plus make sure all traces have a strong return path coupling to the nearest ground plane first, then worry about perfect symmetry in PCB stacking.

2. Routing Guidelines for PCI Layout Designs

Connect your signal tracks to match the networks in your schematic. PCB layout best practices recommend that you always place short, straight tracks between components when possible, although this is not always practical on larger boards. If your component arrangement forces horizontal trace routing on one side of the board, always route traces vertically on the opposite side. This is one of the many important rules of 2-layer PCB design.

PCB design rules and PCB layout guidelines become more complex as the number of layers in your stack increases. Your routing strategy will require alternating horizontal and vertical traces on alternating layers, unless you separate each signal layer with a reference plane. On very complex boards for specialized applications, many of the most common PCI best practices may no longer apply, and you will need to follow the design guidelines specific to your application.

3. Defining Trace Width

PCB layout designs use tracks to connect components, but how wide should these tracks be? The required track width for different networks depends on three possible factors:

1. **Manufacturing:** The tracks can't be too thin, otherwise they can't be manufactured reliably. In most cases, you will work with track widths that are much wider than the minimum value that your manufacturer can produce.
2. **Current:** The current flowing through a track will determine the minimum width needed to prevent it from overheating. When the current is higher, the track needs to be wider.
3. **Impedance**: High-speed digital signals or RF signals will need to have a specific path width to achieve a required impedance value. This doesn't apply to all signals or nets, so you don't need to enforce impedance control on all nets in your design rules.

For tracks that do not need specific impedance or high current, a track width of 10 mills (0.254 mm) is good for the great majority of low-current analog and digital signals. PCB tracks carrying more than 0.3A may need to be wider. To verify this, you can use the IPC-2152 standard to determine the track width of your PCB for a required current capacity and temperature rise limit.

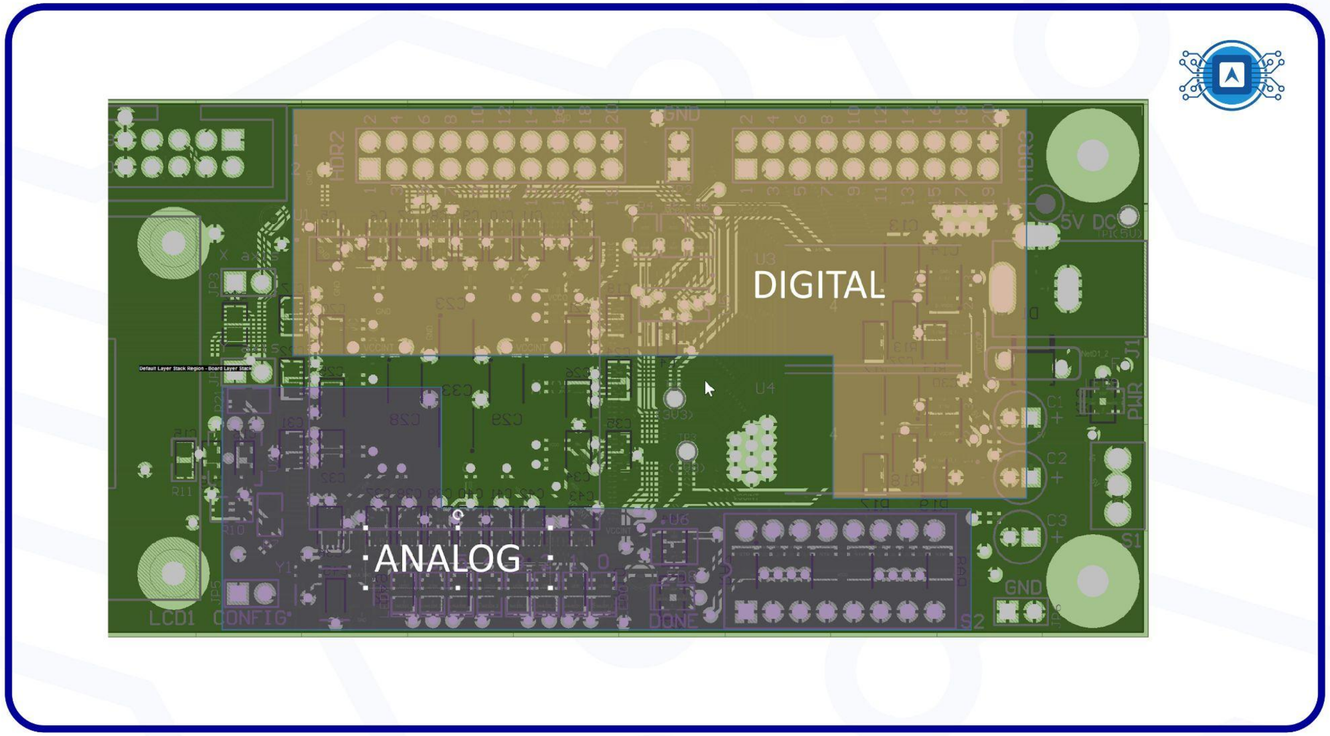

Image 09: Example of Analog and digital area separation on a PCB.
Source: *Embedded computer design <https://embeddedcomputing.com/technology/processing/guidelines-for-designing-a-printed-circuit-board>*.

Thermal relief fittings for Through-Hole components

The ground plane can act as a large heatsink that carries heat evenly throughout the board. Therefore, if a specific path is connected to a ground plane, not using this thermal relief on that path will allow heat to be conducted into the ground plane. This is preferable to keeping the heat trapped near the surface. However, this can create a problem if TH components are assembled on the board using wave soldering, because you need to keep the heat trapped close to the surface.

Thermal Reliefs are a PCB layout design feature that may be necessary to ensure that a board is manufactured in a wave soldering process, or in other words, for TH components connected directly to planes. Since it can be difficult to maintain the process temperatures when a through hole is a solder point directly to a plane, the use of thermal reliefs is recommended to ensure that the soldering temperature can be maintained. The idea behind thermal relief is simple: it decreases the rate at which heat is dissipated in the plane during welding, which will help prevent cold welds.

Typical Thermal Relief Pattern

Some designers will tell you to use a thermal relief pattern for any track or hole connected to an internal ground or voltage plane, even if it is only a small polygon. This advice is often widespread. The need for a thermal path in any through hoje component will depend on the size of the copper plane or polygon that will make a connection in the inner layer, and is something you should request from your manufacturer before putting your board into production.

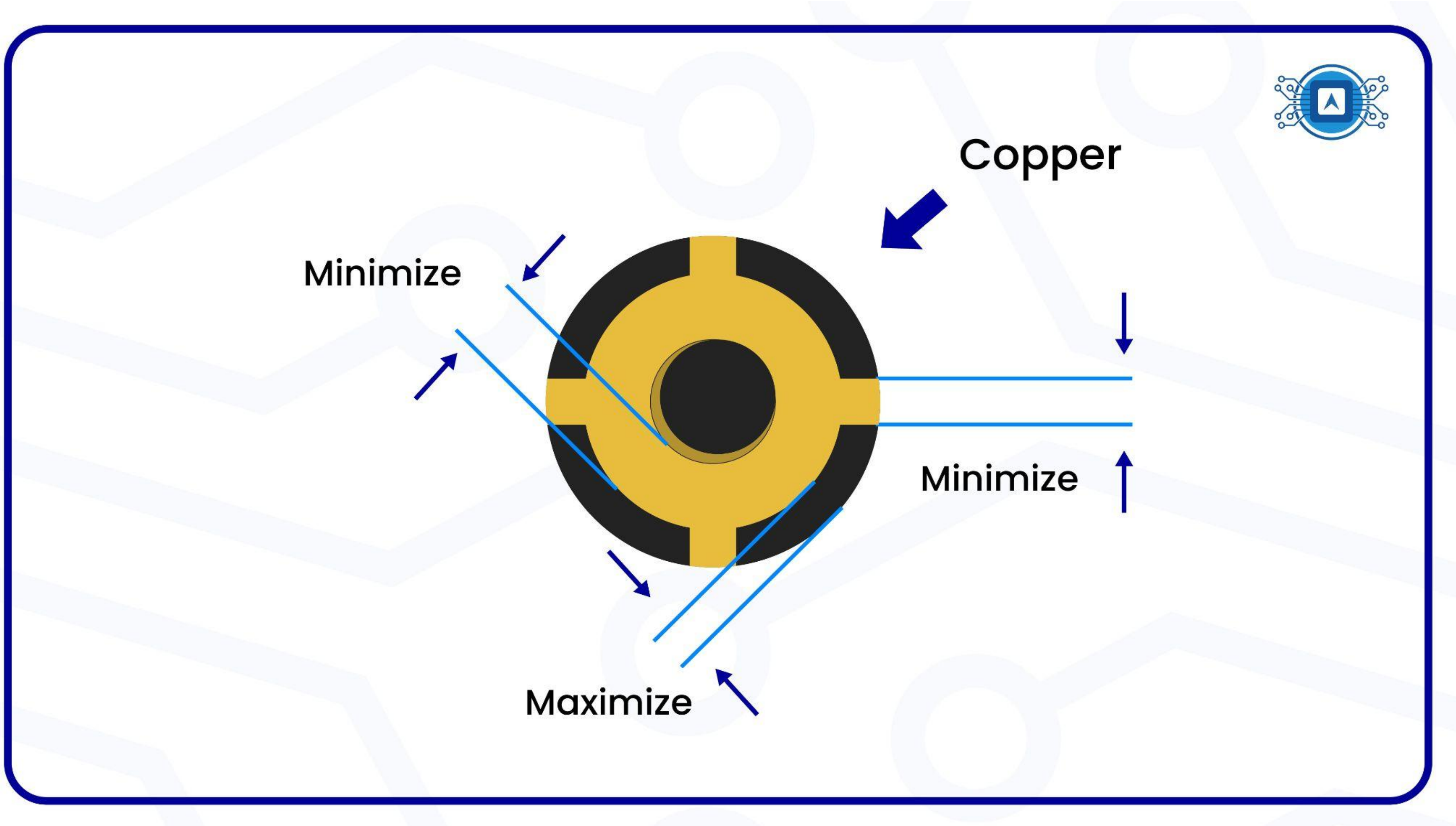

Image 10: Thermal Relief.
Source: *Altium <https://resources.altium.com/p/thermal-relief-design-proper-impedance>*.

4 - Keeping things apart

There are some routing guidelines for PCB on how to group and separate components and traces to ensure easy routing while avoiding electrical interference. These grouping guidelines can also help with thermal management, as you may need to separate high-power components.

Clustering Components

Some components are best placed in the PCB layout design by placing them together in one area. The reason is that they can be part of a circuit and can only connect to each other, so there would be no need to place the components on different sides or areas of the board. PCB layout then becomes an exercise in designing and arranging individual groups of circuits so that they can be easily connected with the tracks.

In many layouts, you will have some analog and digital components, and you must prevent the digital components from interfering with the analog components. The way this was done decades ago was to divide the ground and voltage planes into different regions, but this is not a valid design choice in modern designs. Unfortunately, this is still documented in many board layout guidelines and leads to many bad routing practices that create EMI.

Try instead to use a complete ground plane below your components and don't physically divide the ground plane into sections. Keep all the analog components operating on the same frequency. Also, keep digital components together. You can visualize this as having each component type filling a different region above the ground plane in the PCB layout design, but the ground plane should remain consistent in most designs.

Splitting high-power components

It is appropriate to separate the components that will dissipate a lot of heat on the board into different areas. The idea behind splitting these high-power components is to balance the temperature around the PCB layout, rather than creating large hotspots in the layout where the high-temperature components are clustered. This can be done first by finding the "thermal resistance" ratings in your component datasheets and calculating the temperature rise from the estimated heat dissipation. Heatsinks and coolers can be added to keep component temperatures down. You may have to carefully balance the placement of these components to maintain short rail lengths when devising a routing strategy, which can be challenging.

5 – Completing Your PCI Design and Layout

It is easy to get overwhelmed at the end of your design project as you struggle to squeeze in the remaining pieces for fabrication. Double and triple checking your work for errors at this stage can mean the difference between manufacturing success or failure.

To help with this quality control process, we always recommend starting with Electrical Rules Checking (ERC) and Design Rules Checking (DRC) to verify that you have met all the constraints you have set. With these two systems, you can easily define gap widths, trace widths, common manufacturing requirements, high-speed electrical requirements, and other physical requirements for your specific application. This automates the PCB layout review guidelines to validate your layout.

Be aware that many design processes claim that you should run design rule checks at the end of the design phase as you prepare for manufacturing. If you use the right design software, you can run checks throughout the design process, which allows you to identify potential design problems early and correct them quickly. When the final ERC and DRC deliver error-free results, it is recommended to check the routing of each signal and confirm that you have not missed anything by running the layout one wire at a time.

These were some key PCB layout guidelines that apply to most circuit board designs! Although the list of recommendations is short, these guidelines can help you quickly develop a functional and manufacturable board. These PCB design guidelines are only a fraction of rules, but form a foundation to build and solidify a practice of continuous improvement in all your design practices.

References

ALTIUM LLC. In: **Top 5 PCB Layout Basics Every Designer Needs to Know.** Ultima atualização: 25/10/2021. Karlsruhe, Germany, 21 fev. 2017. Available at: **<** https://resources.altium.com/p/top-5-pcb-design-guidelines-every-pcb-designer-needs-know#determine-your-pcb-design-rules-before-layout > . Accessed on may 23th 2022.

Semtech Corporation. **SX1262: Long Range, Low Power, sub-GHz RF Transceiver** , datasheet:. Rev. 1.1. Camarillo, CA, 2017. Available at: **<** https://semtech.my.salesforce.com/sfc/p/#E0000000JelG/a/2R000000Un7F/yT.fKdAr9ZAo3cJLc4F2cBdUsMftpT2vsOICP7NmvMo >. Accessed on may 23th 2022.

STMicroelectronics. **BLUENRG-LP,** dataseet:. Rev.5. ed. Genebra, Suíça, 5 abr. 2022. Available at: **<** https://www.st.com/resource/en/datasheet/bluenrg-lp.pdf >. Accessed on may 23th 2022.

Additional Reading

GLOSSARY

TEST-BOARD

A board designed to test an electronic component or circuit.

Project

Usually defined as an initiative (whether collaborative or not), often involving research or design, that aims to achieve a unique outcome.

Datasheet

It is a document that summarizes the performance and other technical specifications of a product, machine, component, material, subsystem, or software in sufficient detail that it can be used by a design engineer to integrate the component into a system.

PCB

Printed Circuit Board (PCB) is a board with copper-plated, electronically designed faces to insert electronic circuit components.

Symbol

An electronic symbol or electronic symbology is a pictogram used to describe a variety of electrical and electronic devices or functions, such as wires, batteries, resistors, and transistors, in a schematic diagram of a circuit.

Footprint

Geometric description of an electronic component, used to design PCB.

Internal Architecture

Internal architecture or internal structure of a microcontroller or a chip are a set of electronically interconnected blocks that makes up a circuit or part of a circuit for a particular purpose.

Circuit

Circuits or electronic circuits differ from electrical circuits because they have interconnections between several electronic components, whereas electrical circuits only have connections between electrical components.

Firmware

It is a specific kind of computer software that provides low-level control for specific hardware devices.

Copper masks (TOP and/or BOTTOM).

They are mechanical drawings originally from an electronic circuit with interconnections of electronic components that will be passed to a board with copper plated surfaces to generate a printed circuit board.

EMI

Electromagnetic interference is a generic term commonly used in electronics to express noise that interferes with the transmission of information or the operation of a circuit.

SMD

Surface mount technology is an electronic circuit assembly method in which the components or SMC are built directly onto the surface of the printed circuit board, allowing both sides to be used.

PCB Layout

A layout is a digitized form of an electronic circuit, focused on the perspective of how it would look on a printed circuit board (PCB)

ICs

Integrated circuit (or simply IC) is an electronic circuit that incorporates miniatures of several components (mainly transistors, diodes, resistors and capacitors), " printed" on a small silicon chip.

Zachariah Peterson

He was a Professor at Portland in the United States and led research on random laser theory, materials and stability has an extensive technical background in both academy and industry. He currently provides research, design, and marketing services to companies in the electronics industry. Before working in the PCI industry, his scientific research background covers topics in nanoparticle lasers, semiconductor electronic and optoelectronic devices, environmental sensors, and stochastics. His work has been published in over a dozen peer-reviewed journals and conference proceedings, and he has written over 1,000 technical blogs on PCB design for several companies. He is a member of the IEEE Photonics Society, IEEE Electronics Packaging Society, American Physical Society, and the Printed Circuit Engineering Association (PCEA), and was a former member of the INCITS Quantum Computing Technical Advisory Committee.