# Checking the recommended pad design for the chip and sizing information in the datasheet

When you begin a layout of an electronic circuit you must have all the circuit's electrical, electronic, and mechanical information.

Electrical - What is the operating voltage of your circuit, what is the consumption current? Also, in some cases, the required power.

Electronics - What electronic components will be used? Are there specific features? map the layout constraints such as tracks, spacing, component proximity.

Mechanical - You must have the component mechanical drawings (footprints). If not, it will be necessary to draw in the CAD Editor with the correct dimensions verified in the component datasheets.

## Download the datasheet

For the design and details such as pad sizes of the Hana Electronics chip (HTLRBL32L - LoRa/BLE), you will find this and other information at the website: https://gitlab.com/ht_advtech/design_team/htlrbl_design.

# Important information

Check all the necessary components for your circuit, analyze the current consumption, if you will use SMD components (Surface Mounted Device) or PTH components (Pin Through Hole), size of the board to be used, connectors, that is, all the necessary components.

# What is the relevance of a schematic diagram?

The schematic diagram or electrical schematic is one of the important aspects of designing a printed circuit board or PCB. A good schematic shows a well-structured circuit diagram, clearly showing the electrical connections between multiple electronic components. It should also be noted that a technically correct but crowded schematic is still bad, as it can confuse designers. According to Amit (2020), schematics can be an extremely valuable troubleshooting tool because they trace the connections in the circuit.

# Here are some guidelines for drawing a PCB schematic diagram

To achieve a successful project, follow these standard schematic guidelines:

### 1. Page Size

Most design tools offer different page sizes and it is recommended that designers make their selection based on the size of the circuit project. Generally, the tools already default to A4 page size, however, it should be noted that several other page sizes are also available.

## 2. Page Nomenclature

The logic blocks in the schematic should be separated by pages, named using the letters A, B, C, and so on. In doing so, we can put the pages in alphabetical order, as shown below (AMIT, 2020)

A. Block Diagram
B. Power Supply
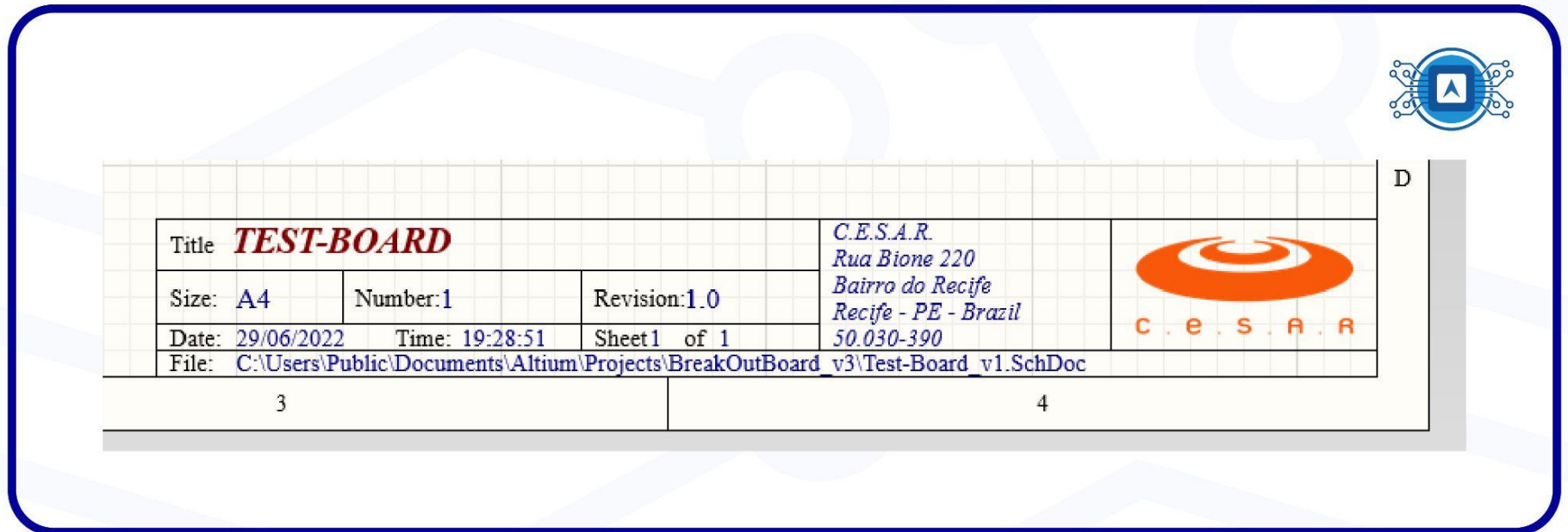C. MCU interface
D. Memory interface
E. Revision history

Block diagrams and revision history are often ignored by most designers to save time. However, they can be very helpful to other designers trying to understand the schematic. Most product-based organizations require all of these protocols and regulations.

## 3. Grid Configuration

Although it is not a direct requirement from the designer, the CAD tool that we will be using needs to have some references. Therefore, the grid system is followed. Having grids helps the designer to reference the parts correctly and make their connection. Circuit components and connections should always be on the grid, this helps in probing the grids during analysis.

## 4. The Template with the Page Title Block

The Template with the page title block, is present at the bottom of the schematic page. It is good practice to fill in all the necessary details, such as page size, update date, revision, document number, circuit name/function, and company disclaimer, as shown in image 1.



Image 01: Page Title Block. Source: *Screenshot by the author*.

## 5. Notes/Comments

Designers must write the necessary comments about the circuits. The notes can be written in separate documents or pages in the schematics. Generally, notes are provided on a separate page for complex designs. Examples of notes can be jumper status, PCB layout constraints/guidelines, and so on. A schematic with notes can be seen in image 02.
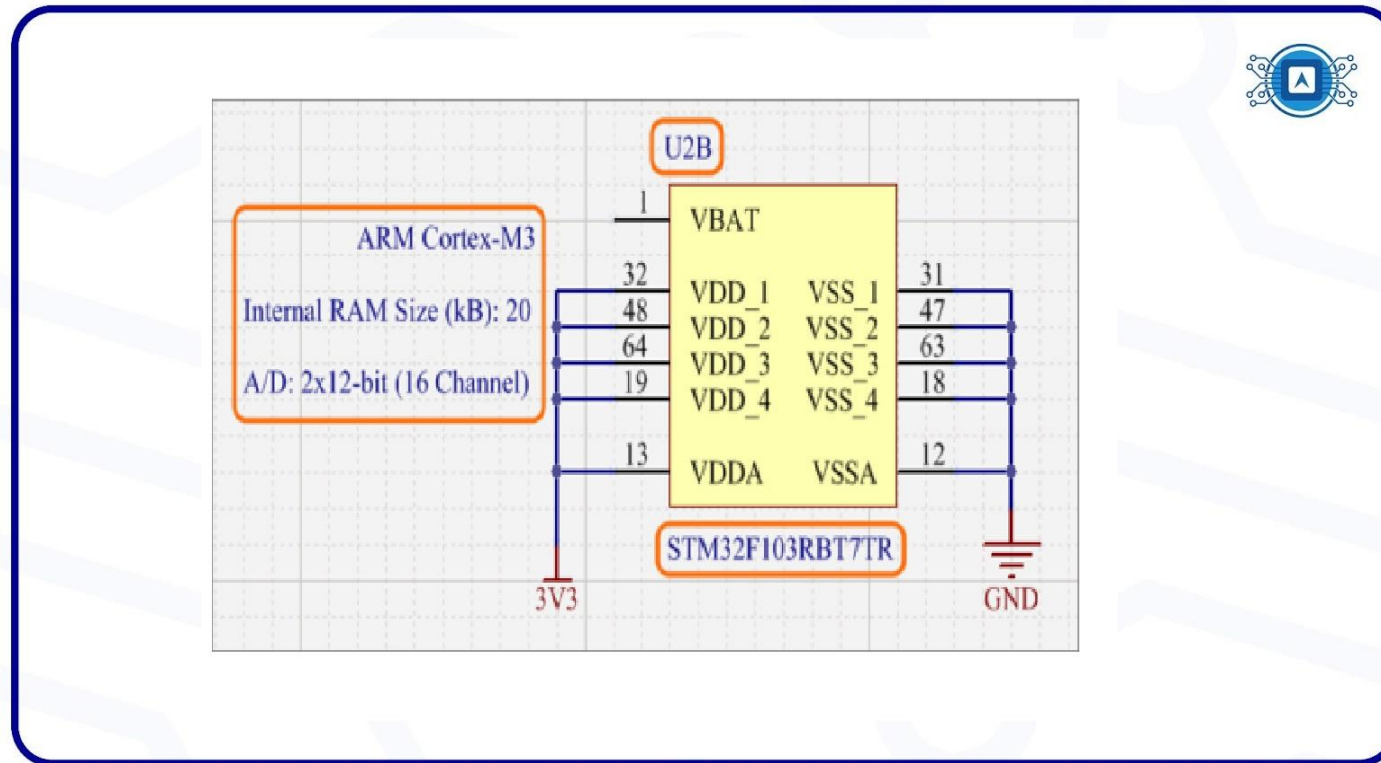


Image 02: Notes/comments. Source: *Altium <https://www.altium.com/documentation/altium-designer/schematic-parameter?version=18.1>*.

## 6. Revision History

The revision history is where you will find the changes that have been made to the project. This document provides information such as the date and description of the changes made, the name of the author and the reviewer and revision comments, if any, as shown in image 3. The revision history is usually placed on the first or last page of the schematic. 03.



| Rev | Nature Of Changes from Second Gerber Release | Date | Approved By |
|---|---|---|---|
| 3.0 | LAYOUT CHANGES<br><br>C192 TH cap is added<br>U44 audio amplifier is added<br>DB9 Connector is removed and it is changed to 3 pin relimate connector<br>Debug UART is also connected to RJ45 power connector<br>Added new circuit for Tricolout LED<br>Mic connector is changed to 3 pin relimate connector<br>Hole size of the relimate connector is increased<br>Mic connector is changed to 3 pin relimate connector<br><br><br>BOM CHANGES<br><br>C153,C149,C155,C152,C18,C88 made as DNP<br>R182,R177,R199,R350,R176,R198,R215 made as DNP<br>U34,U35,U32,U6,U10, J19 made as DNP<br>C9,C10 made as DNP<br>Audio Amplifier circuit is added<br>R18,R137 changed to 100R<br>R14,R16,R156,R163 made as DNP<br>Tricolour LED circuit is added | 21-03-2013 | ANIL K S |

Image 03: Revision history of a schematic. Source: *Sierra Circuits <https://www.protoexpress.com/blog/how-to-draw-design-pcb-schematic/>*.

# Download footprint templates from GitHub

Based on the information acquired through datasheets made available by Hana Electronics, we will have the drawing of the Hana chip symbol, as well as the footprint of the chip with the size of the pads for better solder flow when assembling the PCB, other necessary components will be determined as needed, as shown in the image below:
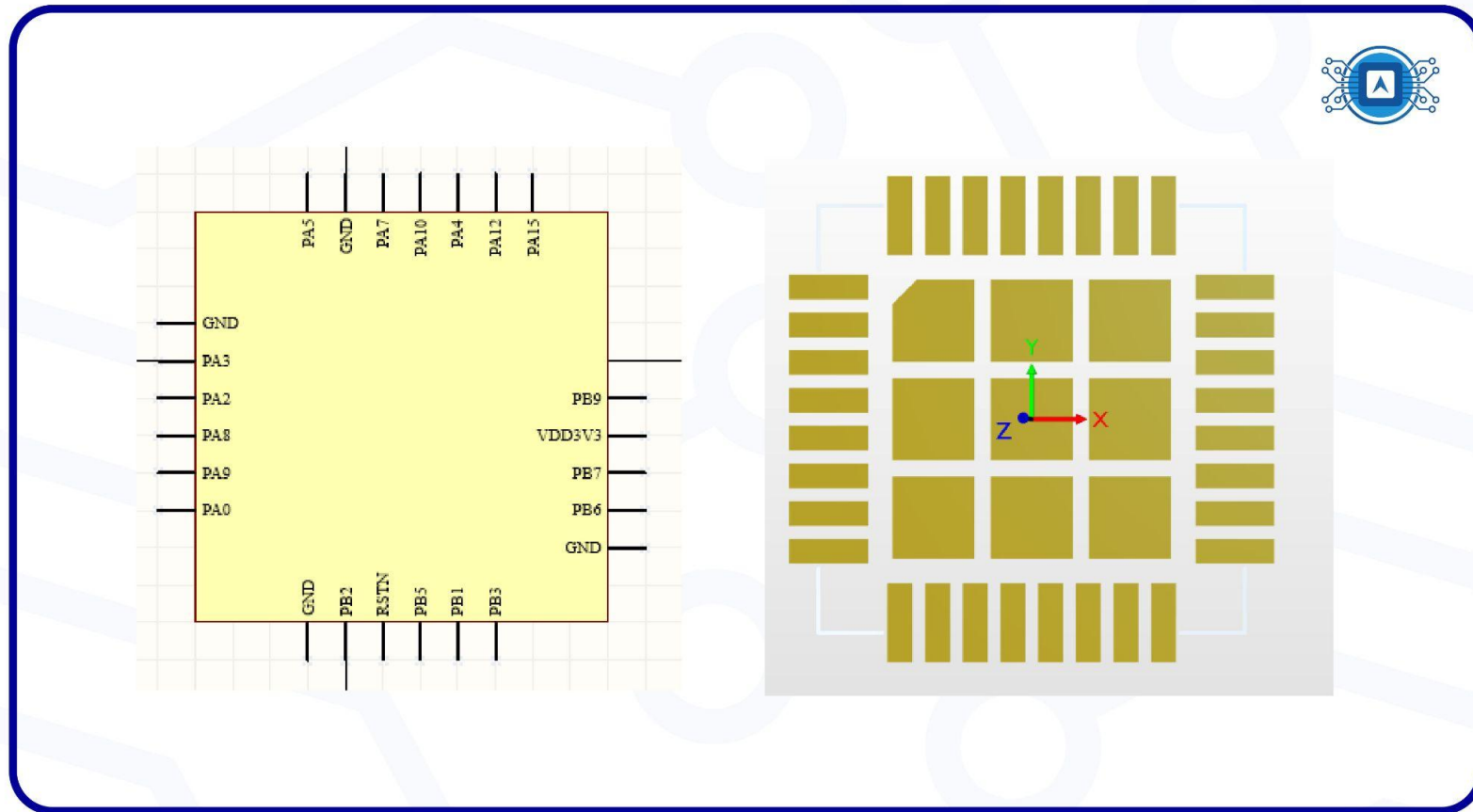


Image 04: (Left) Hana chip symbol, (Right) Hana chip footprint. Source: *The author*.

# Github

It is a web hosting platform for software development and version control using Git. It offers Git's distributed version control and source code management (SCM) functionality, plus its own features. It provides access control and several collaboration features such as bug tracking, feature requests, task management, continuous integration, and wikis for each project.

The link to the Github repository containing the symbols and footprint of the Hana HTLRBL32L - LoRa/BLE chip:

https://github.com/htmicron/htlrbl32l

We will follow below a procedure to download this information:

1 - On the Github screen go to the "Code" button;

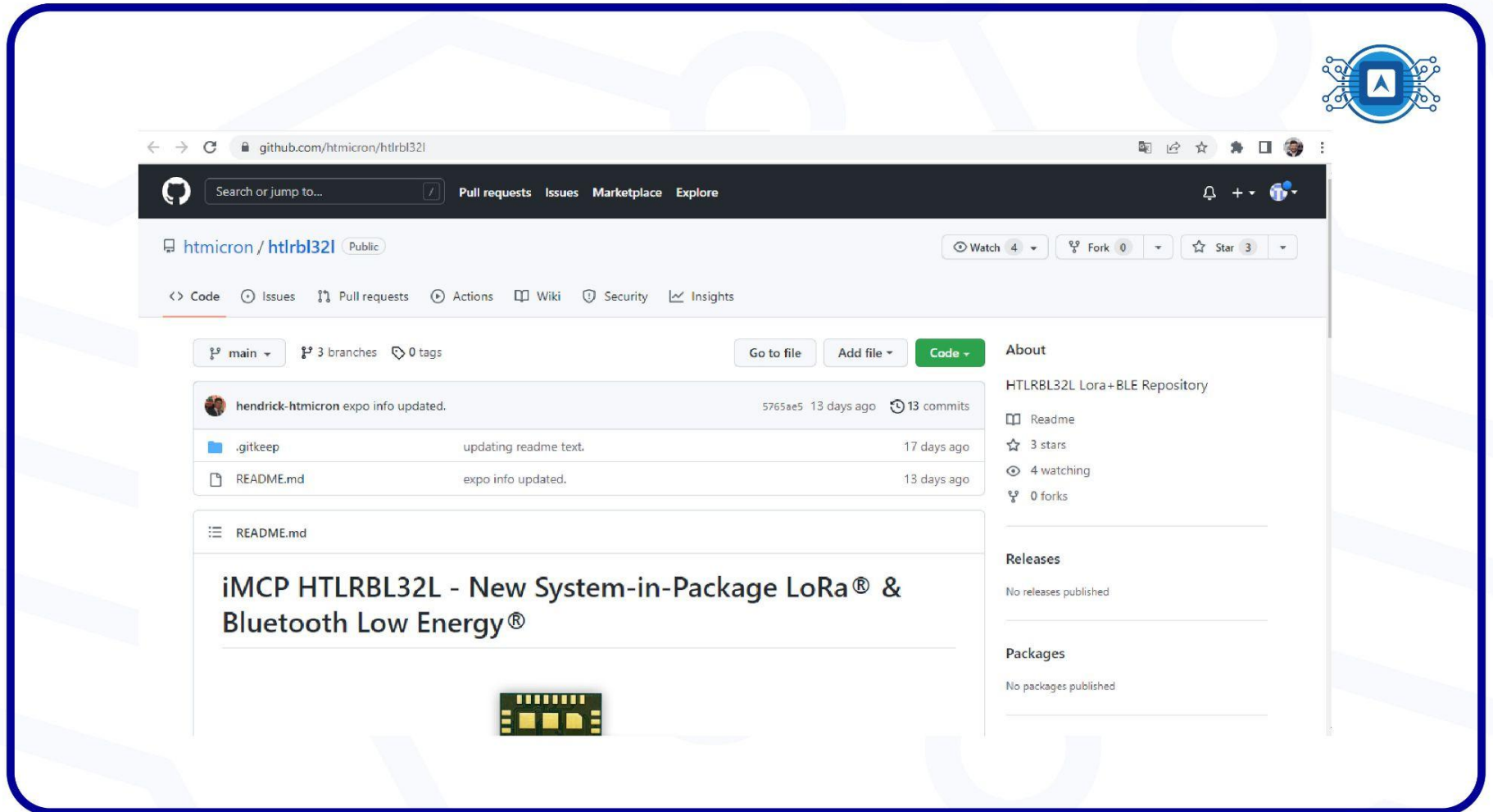

Image 05: Github Tele. Source: GitHub HTLRBL32L Lora+BLE Repository <https://github.com/htmicron/htlrbl32l#readme>.

2 - In the tab, go to the "Download ZIP" option.


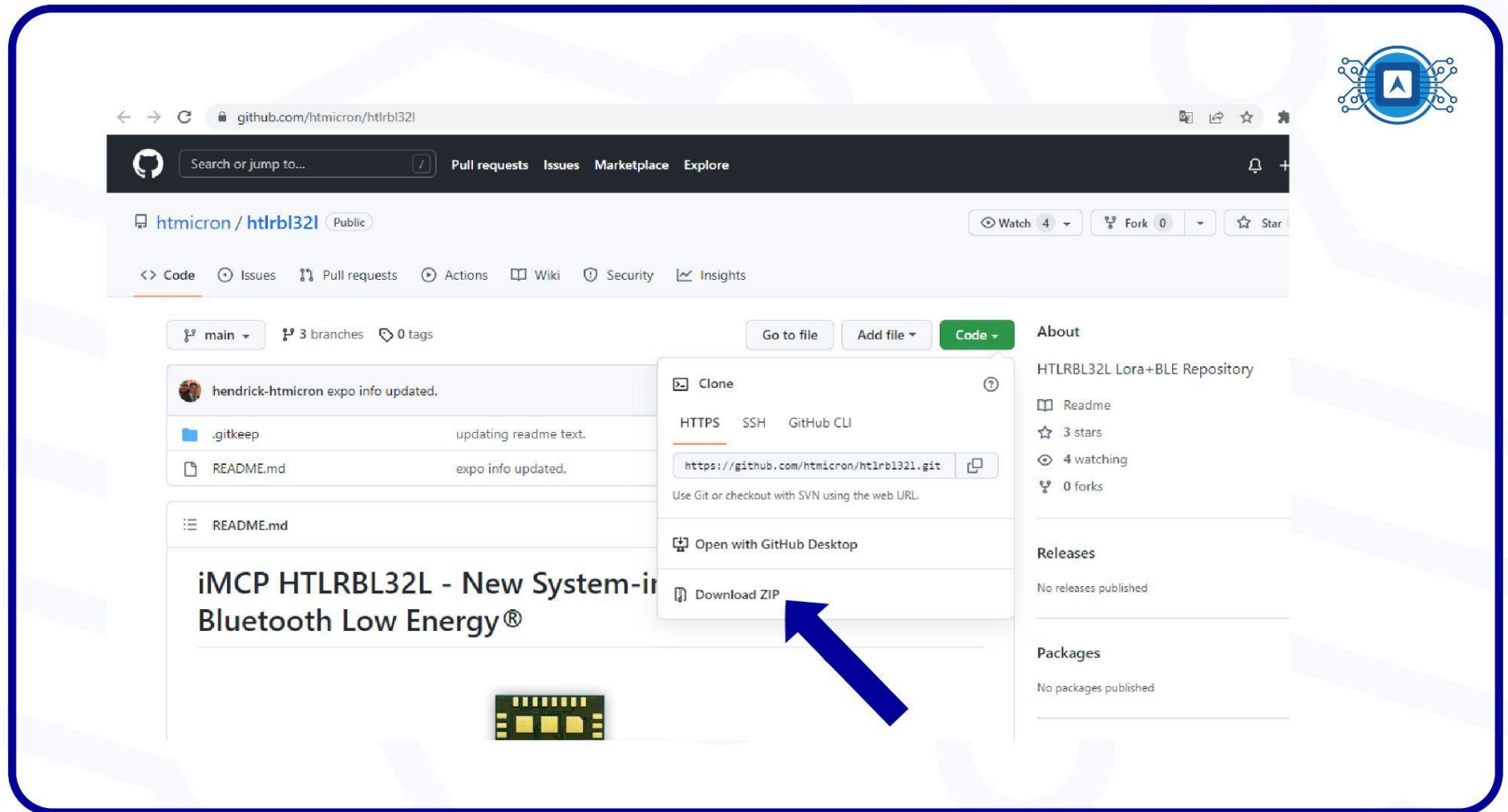
Image 06: Github cloning process. Source: GitHub HTLRBL32L Lora+BLE Repository <https://github.com/htmicron/htlrbl32l#readme>.

3 - A compressed package will be sent to your computer's "downloads" folder, assign this package to a folder of your choice and unzip it.

# References

AMIT, Bahl. **High-Speed PCB Design Guide. Sunnyvale, CA: Copyright Sierra Circuits Inc**. 2020. Available at: **<** https://www.protoexpress.com/pcb-design-guides/high-speed-pcb/ >. Accessed on june 29th 2022.

GITHUB. **In: Wikipédia. [S. l.]**, 19 jan. 2009. Available at: **<** https://en.wikipedia.org/wiki/GitHub >. Accessed on june 29th 2022.

**1.3 Getting Started - What is Git?.** Available at: **<** https://git-scm.com/book/en/v2/Getting-Started-What-is-Git%3F > . Accessed on june 29th 2022.

# Additional Reading

# Glossary

## Datasheets

A datasheet, specification sheet, is a document that summarizes the performance and other technical features of a product, machine, component (for example, an electronic component), material, subsystem (for example, power supply), or software in sufficient detail that it can be used by a design engineer to integrate the component into a system.

## Schematic Diagram

Schematic or wiring diagram is the graphical representation of electrical and electronic circuits.

## Grid

The Grid technique consists of squaring the reference image and the sheet where the drawing will be made, and filling each square individually according to the reference.