This document contains a series of specifications and tasks related to the PCB design and manufacturing for our project, which is intended for production at JLCPCB.
The tasks listed below are crucial to ensure that the project meets quality standards and manufacturing requirements, as well as to avoid errors that could lead to delays or unforeseen changes. Each development phase, from electronic design verification to netlist creation and layout specification management, has been meticulously planned to ensure perfect component integration and compliance with the limitations imposed by JLCPCB's manufacturing process. In addition, it is essential that the tasks related to change documentation, layer management, power and connection analysis, and signal integrity verification are carefully followed. The tasks have been divided into specific tasks, each of which aims to solve critical issues related to the PCB design. It is important that each task is completed strictly according to the specifications, in order to avoid the risk of introducing new issues that could compromise the realization of the project.

We ask that you pay special attention to managing print limits and separating non-printable information on the appropriate layers. Additionally, generating PDF files for the factory, design analysis, and component review (with particular attention to supplier compatibility) are essential to the success of the project.

Overview

The board is designed to support essential peripherals such as eMMC storage, LPDDR4X RAM, WiFi/Bluetooth, camera module, and display. The design is optimized for performance and ease of assembly, with a compact form factor.

🎯 The goal was to make it accessible to everyone—hobbyists, makers, robotics enthusiasts—**without minimum order limitations**. This project follows the workflow of creating an SBC (Single Board Computer), robot board, and wearable device.

📄Technical information

Dimensions
W: 34mm | **H:** 30,946mm | **T**: 1mm

Primary Components
Back Side
* *NXP* **MIMX8MN5DVPIZAA** (I.MX 8M NANO)
* PMIC *NXP* **PCA9450BHNY**
* Antenna **M310220** *Kyocera Avx*
* **ADG824BCPZ-REEL7** or **QPC8020QTR13** (For Antenna Diversity)
* Camera (Minimum 1MP or 2MP 16/24Pins around 100 FOV) we only found this product
* Connector *JST* **B4B-ZR-SM4-TF** **1.5mm, 1 Rows, 4 Contacts** (Bodyboard connector and communication: We

must have the same 4-pin header on the new board. This is where power comes in and where the serial communication pins are**)**
*** Pads usb** for SDM (Serial download mode for flashing firmware)
***** Speaker Pads and external speaker, we only found this product (if don't is compatible change with another one)
Combo chip:
  * *Kingston* **32EP16-M4FTC32-GA68**. eMMC 5,1 HS400 + LPDDR4X 144-FBGA
  * *Realtek* **RTL8723DS** (Wifi e Bluetooth, 6x6mm) but I want to use **CYW43012** **(106-ball WLBGA (3.76 x 4.43mm; 0.35mm pitch)**
Front Side
* IMU Bosch **BMI160**
* Display **ST0103A2W-WSNLW-F**
* NC7SP125P5X
* MAX98357AETE+

Software Version: v24.6.1

Legends:
🟥 **High Priority (Urgent)** – Critical task, must be completed immediately.
🟧 **Medium-High Priority** – Important but can wait compared to urgent tasks.
🟨 **Medium Priority** – Should be completed within a reasonable time, but not urgent.
🟩 **Low Priority** – Non-urgent task, to be done if time allows.

Validation of electronic design

Pinmux's check with nxp official software: NXP Config Tool for Configuration to verify the Configuration of the SoC Pine (Link Software NXP MCUXEXPRESSO-CONFIG-TOOLS), export file and put inside folder project.

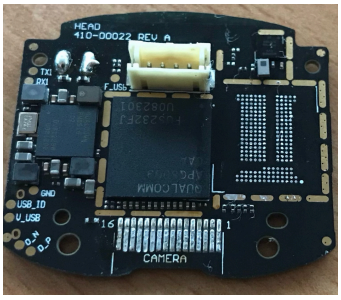
Before starting the project and tackling the related tasks, it is essential to perform the Pinmux configuration using the official NXP software, MCUXpresso Config Tools. This step is essential to verify the correct pin assignment of the SoC and prevent any hardware conflicts. After the configuration, the generated file must be exported and placed in the project folder for reference. It is important to note that the tasks listed in the document do not follow a strict chronological order, but each step must be completed carefully to ensure the integrity of the design and compliance with manufacturing requirements.

Task #1: Silkscreen 🟥

1. **Move Text Below JLPCB Minimum 1mm to Another Layer:**
   - [ ] Remove any text from the silkscreen that is smaller than 1mm (as required by JLPCB) **without deleting it**.
   - [ ] Move this text to a **Document Layer** or **Notes Layer** (or create a new one) to keep the information without affecting the silkscreen.
   - [ ] add it in a custom layer too and use a suitable color to see small shapes and writings like a neon purple.
2. **Use Comments for Additional Information:**
   - [ ] Add comments or annotations to a **Mechanical Layer** or **Layer** (e.g., "Notes Layer") to include any necessary details about the components or assembly that should be visible but not part of the silkscreen.
3. **Generate Complete Silkscreen in PDF:**
   - [ ] Export the **complete silkscreen** (including both **Top Overlay** and **Bottom Overlay**) as a PDF with all the component labels and markings.
4. **Generate PDF with Only Labels Not Included in Gerber:**
   - [ ] Create a **PDF** showing only the **labels** (component designators) that are present in the silkscreen but **not included in the Gerber files**.
   - [ ] Ensure this PDF includes labels for both **Top and Bottom** layers.

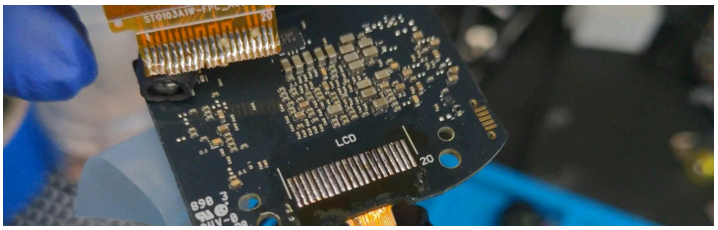
TP8C0 As you can also see in the Top layer (which in reality would be the BackSide, because the bottom layer is actually our front because there is the display). there are many writings that need to be fixed, in any case all those that cannot be printed below the minimum limits of jlpcb must be separated and documented

for everything. in essence the final visual result is as in the photo "Ref Board Backside" but in reality it will be necessary to document everything on the notes layer, mechanical layer for the factory, and 2 pdfs for each side, 1 complete pdf for each side, 1 pdf for the non-gerber labels so that we have everything for both the community and the factory.



”Ref Board Backside”

Recap this task: there must be no texts and things that JLPCB can't print, so move to a non-gerber layer to keep the silkscreen documented without texts



"Ref Front Side"

So it should look like this, **clean**, the texts that can be printed are few like "LCD", sdm pads labels, and other few big ones at minimum, everything else is removed and moved to a documented layer, in the layer notes for factory production and in the pdf. And so it's not just move to a level and who cares and go to the next task, if this is not done well we can't even produce the card.

Task 2: # Schematic 🟥

- ☐ **Multi-sheet & Hierarchical Designs:** Create and structure the schematic pages, breaking the work into multiple pages I think it is about 15/20 schematic pages, each with its own **SUB-PAGES**. Each page should include connection details for the main components such as Block Diagram, CPU (**MIMX8MN5DVPIZAA - now this page missing)**, PMIC, WiFi, BLE, display, power, I/O, memory, and other interfaces. There should be complete management of power connections, pinouts, communication interfaces, and RF layout, with attention to test points and clock management. Make sure everything is properly organized so that each part of the system is well represented and interconnected.
- ☐ Remove old references we don't use in the project like ddr3, microsd, and more things, we do not have an SOM but a fully integrated circuit (purple block in block diagram and if they present in other parts around project).
- ☐ Improve the schematic cpu symbol on schematic and above all carefully check that the selected cpu is documented in the electrical diagram and do a check on the official datasheet that is consistent with the project in general on everything.
- ☐ PCB/SCH fully synchronized.

Task #3: Netlist 🟧

To confirm correct connections

- ☐ Export Netlist file (and put the file inside project folder)

Task #4: Fix Display LCD Footprint and Pads

- ☐ Separate from 3D body, remove circles and unnecessary stuff near, leave display body only in 3D view and must not be joined as footprint at all, display pads must be

separated with near LCD as label, like in "Ref Front Side" picture.

Task #5: Remove/adjust/Fix This. 🟥

- [ ] Ruined shape, these two holes squared and one hole circle are not there in our shape.

- [ ] This no need and appear in gerber and in preview.

Task #6: USB Pads 🟥

- [ ] Download firmware mode with usb pads
- [ ] Place them in approximately the same positions and tilt them in this way D_N and D_P by **approximately 25 degrees**.
- [ ] the serial mode download labels should be printed in the gerber so you have to increase the size to the minimum of jlpcb like the other few labels that will be printed in the factory.

Task #7: Wifi and Bluetooth 🟥

- [ ] Ensure the RF track has the correct impedance as required by the antenna specifications.
- [ ] **Properly trace the copper line for the antenna ("traced antenna") following RF design guidelines.**
- [ ] Confirm that the module reference on the PCB matches the official datasheet.
- [ ] Ensure proper wiring for communication interfaces.
- [ ] Correctly position the antenna to minimize interference.
- [ ] Verify proper grounding and isolation from other signal traces.
- [ ] Check the power supply and filtering for the module.
- [ ] Verify the footprint and pad connections for the **CYW43012** **(106-ball WLBGA (3.76 x 4.43mm; 0.35mm pitch)**.

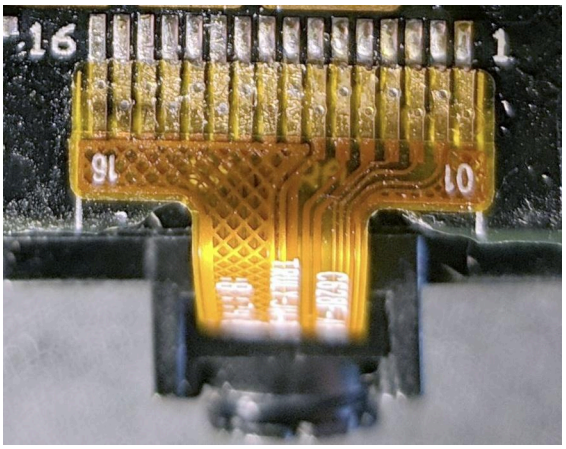
Antenna Diversity Implementation

- [ ] Implement an antenna diversity system to improve WiFi/Bluetooth performance.
- [ ] **Trace the RF line for the secondary antenna to ensure proper diversity, following impedance and RF layout rules.**
- [ ] Verify compatibility of **ADG824BCPZ-REEL7** with the design (if not compatible, find a smaller and more efficient alternative).
- [ ] Simulate the antenna diversity behavior to ensure efficient switching between antennas.

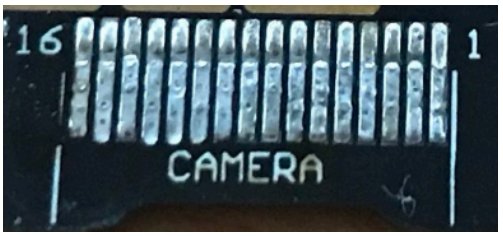
- [ ] Ensure the integration of the RF switch does not introduce significant signal loss.

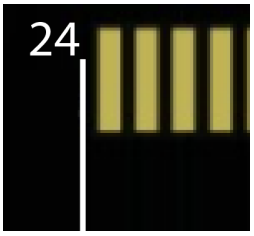
Task #8: Camera and Display 🟥

- [ ] the camera pads must be the other way round so 24 1 because the camera is flexed on the front side.



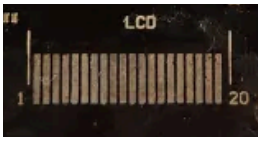
- [ ] CAMERA label in capital letters, same for LCD.





As we can see in the photo on the left (old board), there are two white vertical guidelines with the number of pins above them, extremely important

- [ ] There are two vertical reference lines under the pin numbers 1 line under 24 and one under 1 as in the old PCB. Same for display.



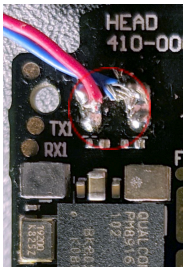
- [ ]  Remove all tiny circle white around the project, don’t need.
- [ ] Adjust display body 3d in front side and with flex connector and same for camera (camera is in back side soldered but is flexed on frontside, see images reference). Missing camera 3d body. the display component placed at the bottom is a bit annoying, make sure it is flexed towards the front with the camera. 🟩

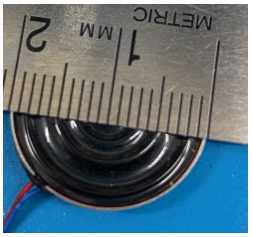
Task #9: Speaker pads 🟥

(**Original**: 20mm 8ohm 0.5 W speaker).

Pads are already present in pcb but i want inside schematic and speaker component selected in bom. We have 2 pin for speaker externally connected speaker connect with NXP board (with I2S protocole?). Add DAC (MAX98357AETE+) inside the board(front-side, bottom-left just fine) then we can out 2 pins for speaker.

- [ ] Working properly in the circuit layout and schematic. In the same position like old board.



near top left, near the corner.


Task #11: Current power supply analysis (Power Integrity - PI) 🟥

Tensions distribution simulation (PDN - Power Distribution Network). Use the Altium PDN Analyzer to check:

- [ ] Voltage falls on the power tracks
- [ ] Impedance of the Power Planes to avoid instability
- [ ] Current distribution (to avoid hot spots)

Control of the Ripple and Noise

- [ ] It simulates tensions on the power lines to make sure that there are no excessive oscillations.

verification of the sequencing of the power supply

- [ ] Some members require that tensions are activated in a certain order (for example, first the DDR, then the CPU, but this must be carefully verified otherwise you risk doing a useless job, The NXP I.MX 8M requires a precise order in activating tensions).
- [ ] Check with the Spice Simulator of Altium that the PMIC regulators activate correctly.

Task #12: Fixing the 6 holes (For the 3D Display/Camera enclosure). 🟧

Screw M1.4


- [ ] The two holes at the bottom are for two M1.4 screws while the other 4 are slightly lower for the plastic of the plastic enclosure that holds the display and the camera, one of these is slightly horizontally oval. So fix diameters, dimensions and perfect them, currently they are made without precision.

Task #13: Drc/Erc Rules 🟥

- [ ] Import and Export Rules for JLPCB Stack-up file (.RUL)
  https://github.com/gsuberland/jlcpcb_autogenerated_stackups (and put the file inside project folder)
- [ ] Pass test 0 errors drc/erc on JLCDFM (official JLPCB tool)
- [ ] Report pdf (and put the file inside project folder)

Task #14: PRESET FILE VIEW 3D AND 2D 🟩

For import inside the configuration panel in Altium.

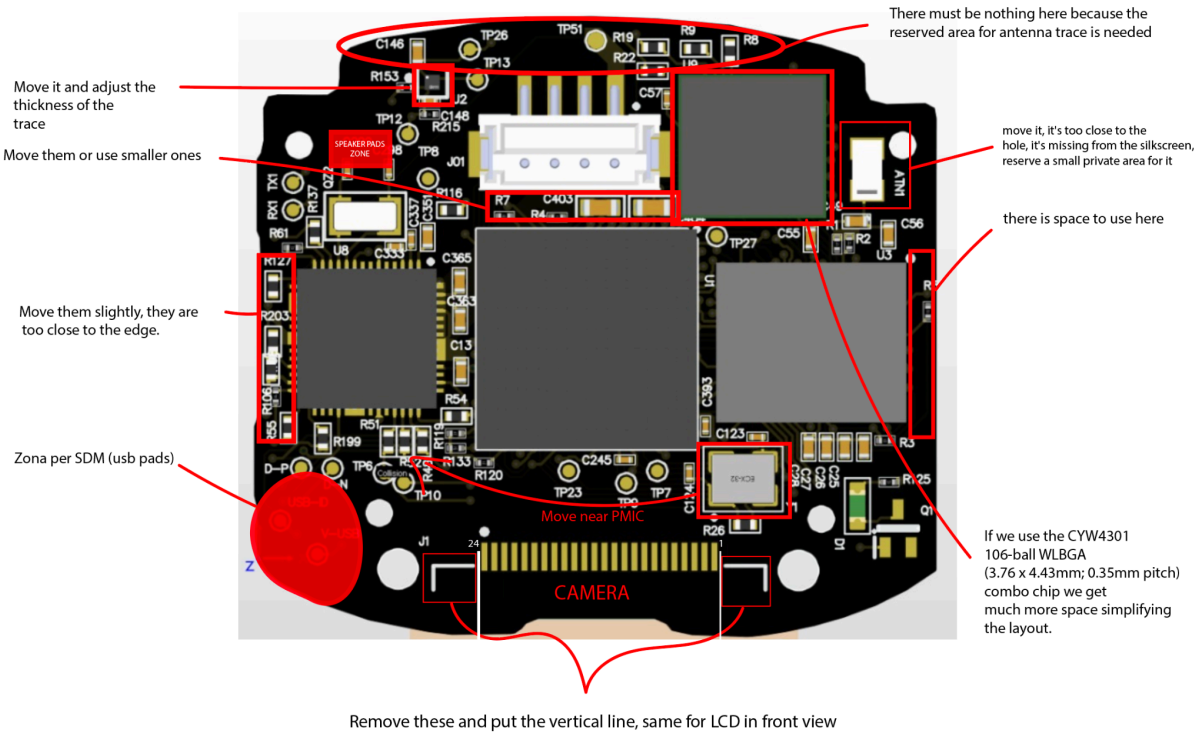
- [ ] Save file inside project folder

Task #15: Pin Mapping 🟥

- [ ] Compare the pins of all components with the datasheets to avoid connection errors.

Task #16: High-Speed Design, Layout Circuit and Signal Integrity and Power Testing 🟥

- [ ] Important improvements:



- [ ] Make sure there are no components too close to the mounting holes.
- [ ] Simulation of the LPDDR4X to check for impedances and reflections.
- [ ] Analysis of power lines with Altium PDN Analyzer.
- [ ] Verification of Emi/EMC and reduction of any mass rings.

Task #17: JST Connector **B4B-ZR-SM4-TF 1.5mm, 1 Rows, 4 Contacts** 🟥

1 - VCC
2 - GND
3 - TX (headboard) → **RX (bodyboard, pin 3)**
4 - RX (headboard) ← **TX (bodyboard, pin 4)**

- [ ] **Update the schematic to reflect the new pinout and generate the updated netlist.**
- [ ] **Re-route the traces in Altium if necessary.**
- [ ] **Do a DRC check (Design Rule Check) to confirm that the layout is correct.**
- [ ] **Simulate and verify the serial connection (UART) to avoid accidental reversals.**
- [ ] 

Add 4,1 in silkscreen(*among the few in the gerber*).

Task #18: Finalize BOM 🟨

- [ ] Add Links and Datasheet and complete all fields inside bom.
  (yes some components are fine separate links 1pc from aliexpress/alibaba like imu, display, controller combo chip realtek)

Task #19: JLCDFM Analysis 🟥

- [ ] Pass all the tests on jlcdfm
- [ ] Report pdf (and put the file inside project folder).

Task #20: Production data 🟩

- [ ] **Screenshots of JLPCB simulated order Complete + Stencyl.**
- [ ] Fabrication drawing.
- [ ] Assembly drawing.
- [ ] Bill of materials (BOM)
- [ ] IPC netlist.
- [ ] Gerber files.

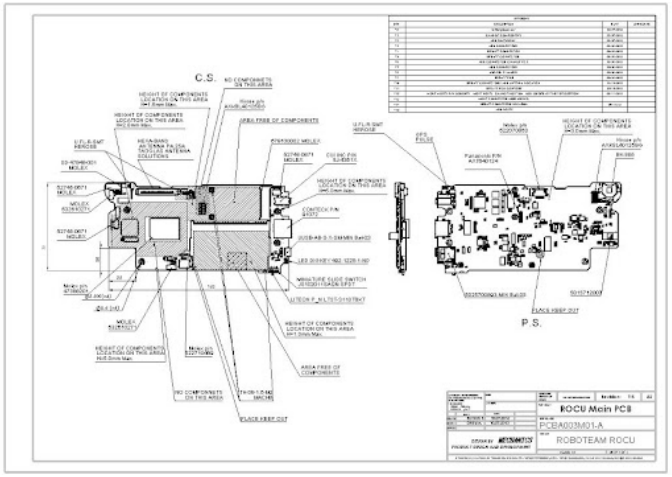
---

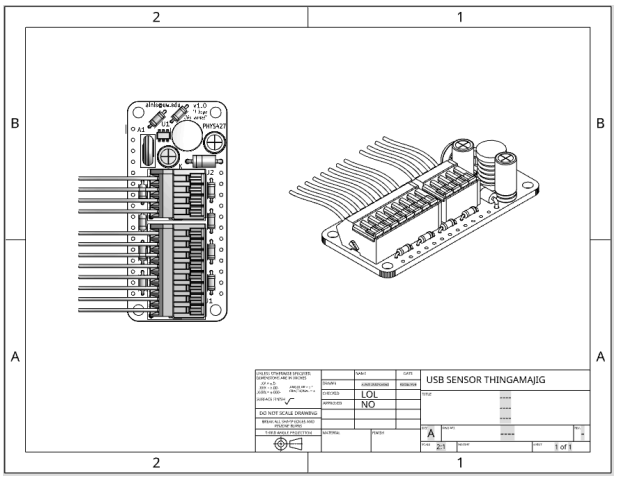
only the components we choose and JLPCB owns them, do not replace them

- [ ] NC drill files.
- [ ] Pick and place files.
- [ ] ODB++

Task# 21: Technical Drawing 🟧

The images are just examples





- [ ] Complete Technical drawing with all information, Nothing should be missing, including hole information.

  Tips:

  - How to Add Fabrication and Drill Drawings Views in Draftsman Documents
  - Easy PCB Assembly Drawing with Draftsman in Altium Designer

Task #22: Thermal Simulation and Analysis 🟥

- [ ] Perform a thermal simulation in Altium.

- [ ] Generate a report with hotspot analysis and heat dissipation data.
- [ ] Video MP4 (Screen recording).
- [ ] Ensure critical components stay within safe operating temperatures.

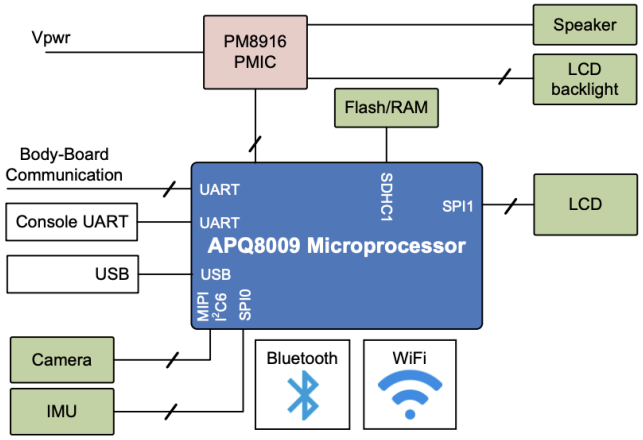
Task #23: Electromagnetic Interference (EMI) Simulation 🟥

- [ ] Run EMI analysis to detect potential interference issues.
- [ ] Optimize PCB layout to minimize EMI.
- [ ] Provide a report with recommended fixes.

Important information about old board

Full Document: https://randym32.github.io/Vector-TRM.pdf

The head-board handles the display, playing sounds, communication, and all of robot real processing. It is powered by a quad-core Arm-A7 Qualcomm APQ8009 microprocessor. The processor also connects to Bluetooth LE and WiFi transceivers, an HD camera, LCD display, speakers and an IMU.



The head-board's functional elements are:

Bluetooth LE transceiver A Bluetooth LE transceiver is built into the package

Camera uses a 1280x720 wide-angle camera (MIPI) camera to visualize his environment and recognize his human companions.

Flash/RAM (eMMC) Flash and RAM are provided by single external package, a Kingston 04EMCP04-NL3DM627 mixed memory chip with 4 GB flash and 512MB RAM.

Inertial measurement unit (IMU) The headboard includes a 6-axis IMU – gyroscope and accelerometer – used for navigation and motion control.

LCD display An IPS LCD, with an active area is 23.2mm x 12.1mm. It has a resolution of 184 x 96 pixels, with RGB565 color.

Microprocessor The head-board is based on a Qualcomm APQ8009 (Snapdragon 212). The processor is a quad-core Arm A7 (32-bit) CPU.

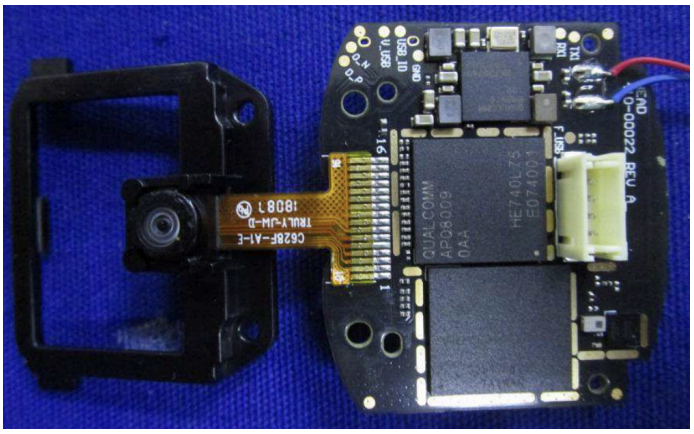
Power management IC (PMIC) The PM8916 power management IC provides voltage regulation for the processor, flash/RAM and other parts; it also provides audio out to the speaker and controls the LCD backlight.

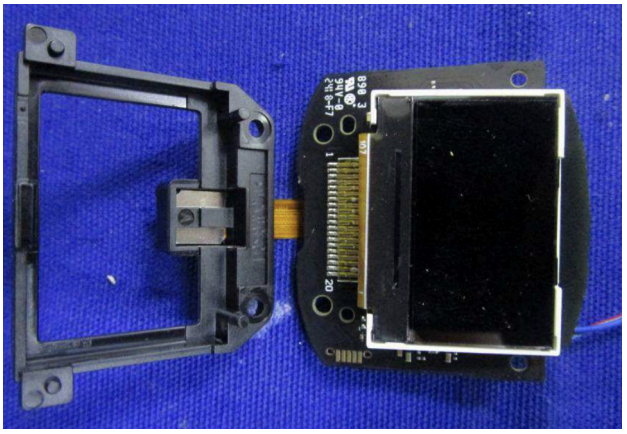
Speaker is used to play sounds, and for speech synthesis.

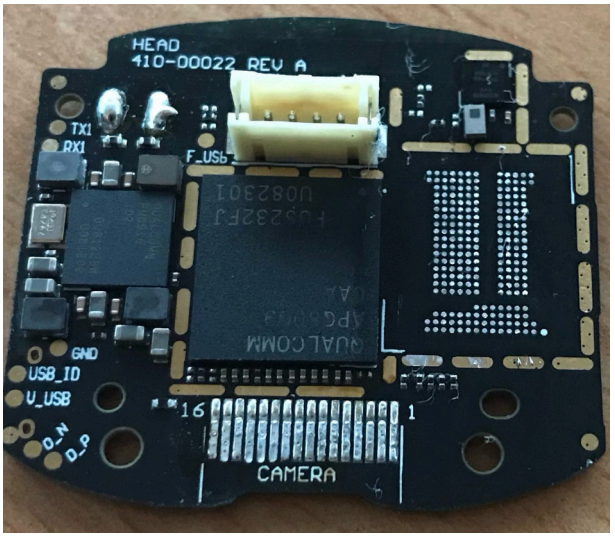
WiFi transceiver An 802.11AC WiFi transceiver is built into the processor package

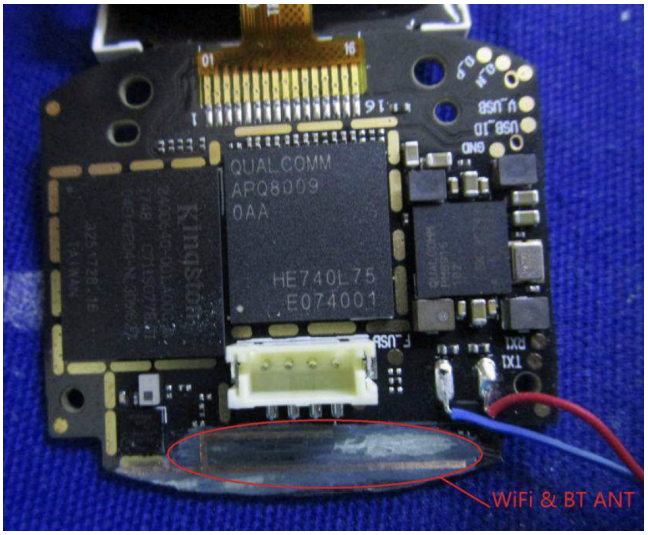
BodyBoard Connector: This is where power comes in and where the serial communication pins are.

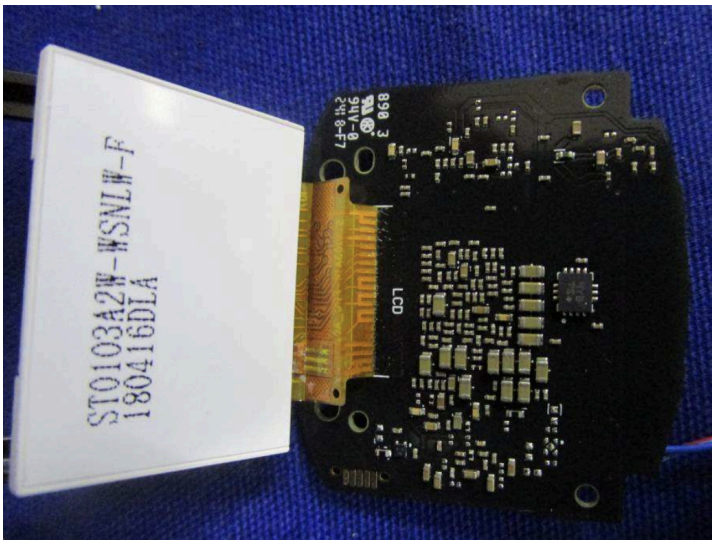
Final Result











Maximum attention is required in the execution of the assigned tasks to avoid the creation of new problems or complications that may generate additional tasks. Any errors or changes made necessary due to negligence, carelessness or lack of checks will be borne by the freelancer at no additional cost. It is essential to scrupulously adhere to the specifications provided, verify each phase of the work and promptly report any critical issues before proceeding with implementations that could negatively impact the project.