PROJECT RED FIR

Advanced solutions crafted for today's needs

Directed by
AKKAOUI Youness

Under the supervision of
Mr. Tomasz Matusiak, PhD

Table of contents

Acknowledgments

I would like to express my sincere gratitude to **SEMIQA** for giving me the opportunity to participate in the Project Red Fir – Advanced solutions crafted for today's needs. I am especially thankful to **Mr. Tomasz Matusiak, PhD**, my internship supervisor, for his continuous guidance, valuable feedback, and technical advice throughout this project. Finally, I would also like to thank my academic institution for their support in allowing me to carry out this internship and gain valuable experience in the field of electronic system design.

Abstract

This report presents the design and development of a printed circuit board (PCB) integrating multiple high-speed communication interfaces including **PCIe (M.2 E-Key slot), HDMI input, QSPI, and RS485 (UART)**.
The objectives of the project were to:

- Create electronic schematics
- Select appropriate components and prepare a complete Bill of Materials (BOM)
- Design a compact 4-layer PCB with optimized placement and routing
- Provide 2D and 3D documentation for manufacturing

This work demonstrates practical expertise in PCB design, component selection, and documentation preparation for advanced embedded systems.

1. Introduction

The rapid evolution of embedded systems and high-speed data communication has increased the demand for compact, efficient, and multifunctional electronic solutions. Within this context, SEMIQA launched *Project Red Fir – Advanced solutions crafted for today's needs*, aiming to address modern requirements for data streaming, system control, and hardware integration.

The central objective of this project is to design a **PCIe-based expansion card** that integrates multiple communication interfaces, including HDMI, RS485 (UART), and QSPI. Such a versatile design can be applied in a wide range of domains, from industrial automation and signal processing to embedded vision and advanced computing applications.

The project primarily emphasizes **schematic design, PCB layout, and component selection**, ensuring that the proposed solution is not only functional but also optimized for size, reliability, and manufacturability. A particular focus is placed on integrating a well-grounded four-layer PCB structure, providing enhanced signal integrity and minimizing electromagnetic interference.

This internship does not cover the physical assembly or laboratory testing of the hardware but instead concentrates on delivering **comprehensive design documentation**, including 2D/3D models, a bill of materials (BOM), and cost estimations. The outcome will serve as a technical foundation for further prototyping and validation by SEMIQA's engineering teams.

Ultimately, the project represents a valuable opportunity to bridge academic knowledge with industrial practice, while contributing to the development of **flexible and future-ready electronic solutions**.

2. Design PCB

A. Project Requirements and Specifications

Main interfaces :

- PCIe via **M.2 E-Key slot**
- HDMI input
- QSPI interface
- RS485 (UART) communication

Control unit : STM32F103 microcontroller acting as an RS485 slave

Supporting elements:

- Power supply regulation (3.3V / 5V via AMS1117)
- Temperature sensor (TMP102)
- Fan control using MOSFET (AO3400A)

Mechanical constraints :

- 4 mounting holes (M2 screws, one in each corner)
- Compact PCB size, 4 layers
- Proper connector placement for usability (HDMI, power, fan, etc.)

B. Schematic Design

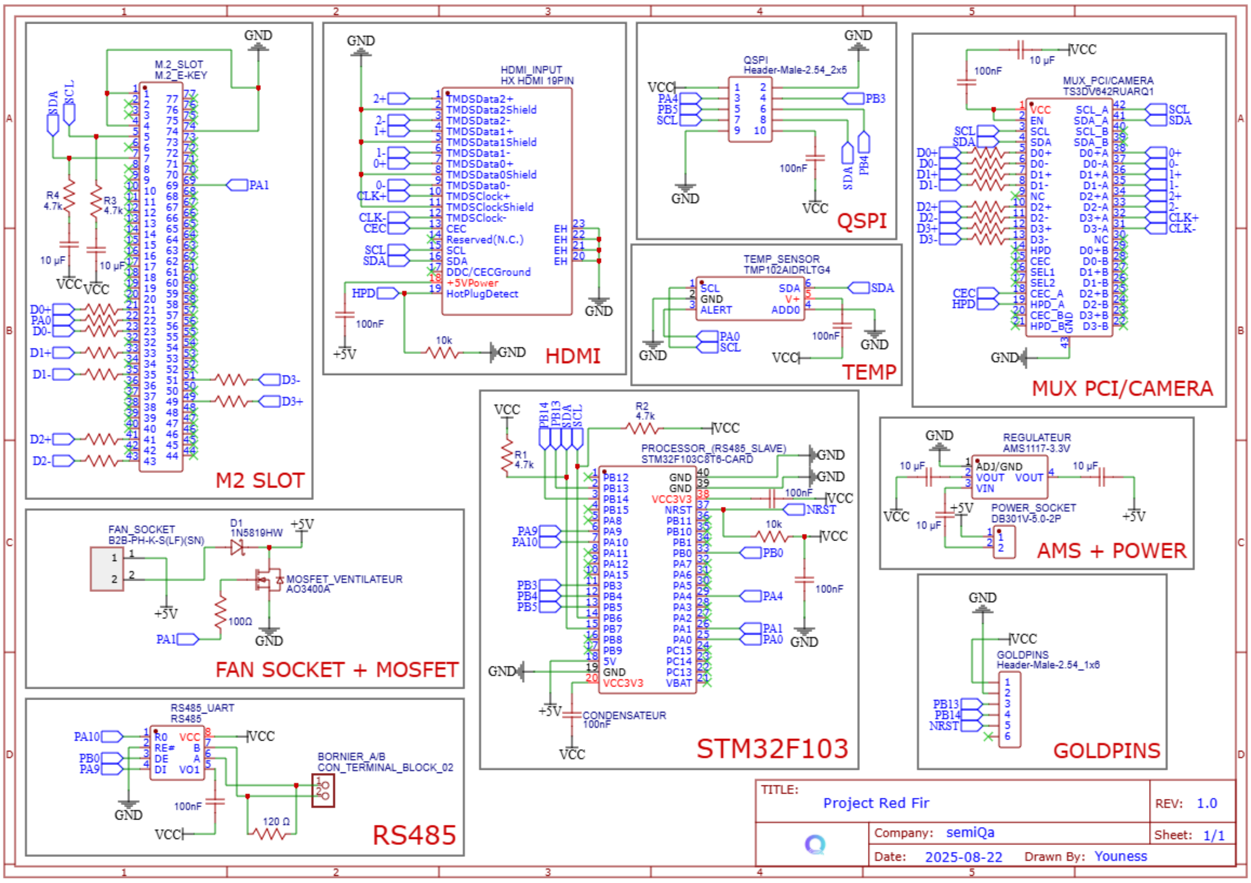
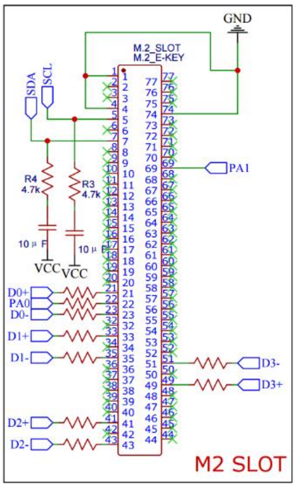


Figure 1 : Schematic Design

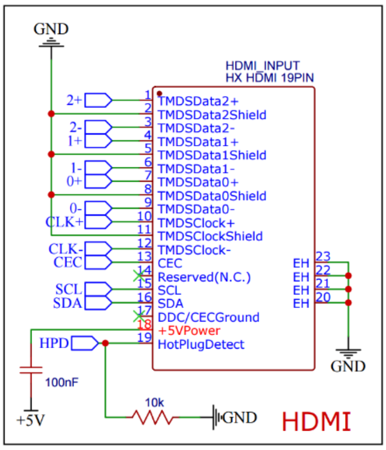
M2 SLOT (E-Key)



Interface to connect an extension module (camera ...) via the M.2 port.

- **SCL / SDA** : I2C communication lines for module configuration.
- **D0±, D1±, D2±, D3±** : High-speed differential lines (video or data).
- **PA0, PA1** : GPIO/control signals connected to the microcontroller.
- **VCC / GND** : Power supply for the module.

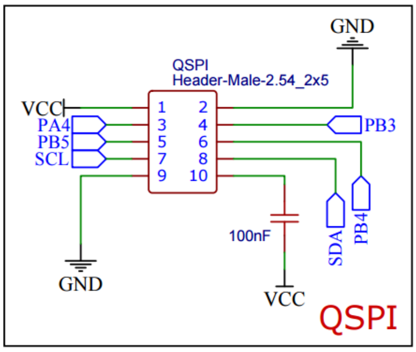
HDMI INPUT (HX-HDMI 19 PIN)



HDMI video input to capture an external video stream.

- **TMDS Data (D0±, D1±, D2±)** : Differential pairs for transmitting video data.
- **TMDS CLK±** : Differential clock associated with the data.
- **CEC (Consumer Electronics Control)** : HDMI control line.
- **SCL / SDA** : DDC (Display Data Channel) bus via I²C.
- **HPD (Hot Plug Detect)** : Indicates the presence of an HDMI device.
- **+5V / GND** : Standard HDMI port power supply.

QSPI Header (2x5)

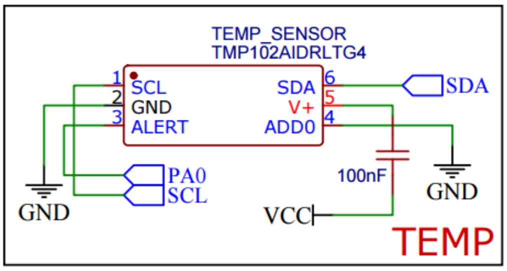


The main role is to debug/fast memory extension interface with QSPI.

- **VCC / GND** : Power supply.
- **SCL** : Serial clock.
- **PB3, PB9** : Data lines connected to STM32.

  ➡ Used to connect an external flash memory or a fast test module.

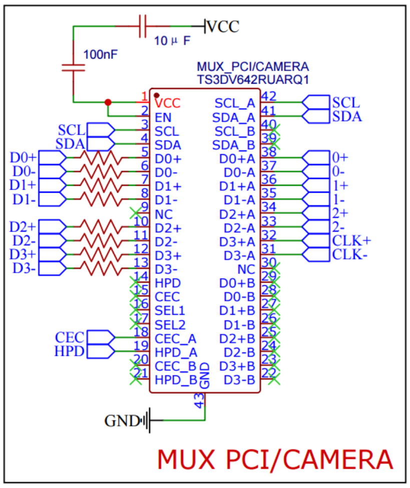
TEMP Sensor (TMP102)



Measure ambient/system temperature.

- **SCL / SDA** : I2C communication with the microcontroller.
- **ALERT** : Hardware interrupt sent to the STM32 when a threshold is exceeded.
- **ADDR0** : I2C address configuration.
- **VCC / GND** : Power supply.

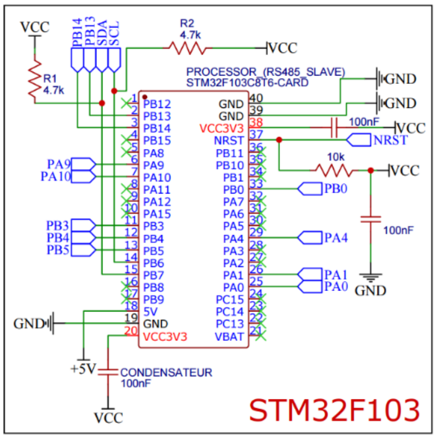
MUX PCI/Camera (TS3DV642)



High-speed video multiplexer to switch between two sources (HDMI, M.2, camera).

- **EN** : Enable/disable the multiplexer.
- **SEL0 / SEL1** : Input selection lines (A or B).
- **D0±, D1±, D2±, D3±, CLK± (A/B)** : Differential pairs for video channels.
- **CEC / HPD** : Auxiliary HDMI/DP signals also multiplexed.
- **SCL / SDA** : I²C for software configuration.
- **VCC / GND** : Circuit power supply.

STM32F103 (Microcontroller)

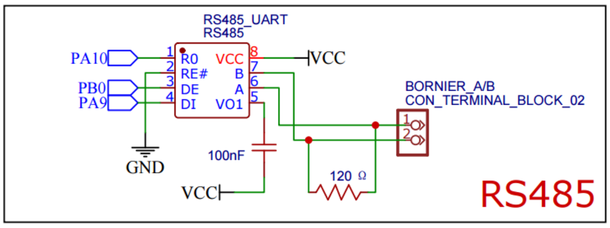


System core, managing logic and communication.

- **GPIO Ports (PAx, PBx, PCx)** : Configurable inputs/outputs for sensors, controls, etc.
- **NRST** : Reset pin.
- **VCC3V3 / GND** : 3.3V power supply.
- **PA9 / PA10** : UART for RS485 communication.
- **I2C (PA0, PA1, PB6, PB7)** : For TMP102 sensor and multiplexer.
- **QSPI (PB3, PB9)** : External memory interface.

➞ The STM32 coordinates the entire system: video selection, sensor reading, external communication.

RS485 UART + Connector

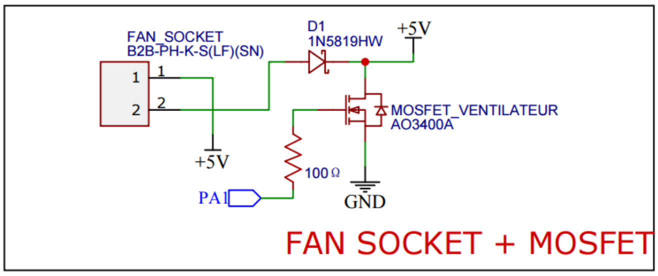


Industrial-grade serial communication via RS485.

- **DI / RO** : Data transmit/receive.
- **RE / DE** : RS485 driver control.
- **A / B** : Differential RS485 bus lines.
- **VCC / GND** : Power supply.

➞ Connected to the STM32 for exchanging data with external systems.

Fan Socket + MOSFET



Fan control for cooling.

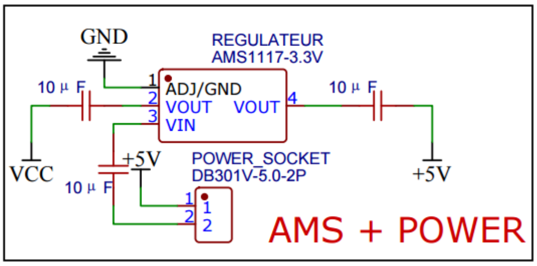
- **Fan Socket** : Power connector for the fan.
- **MOSFET AO3400A** : Electronic switch controlled by the STM32.

- **PA1** : PWM control signal to regulate fan speed.
- **+5V / GND** : Fan power supply.

AMS1117 + Power

Main Role : Power regulation.

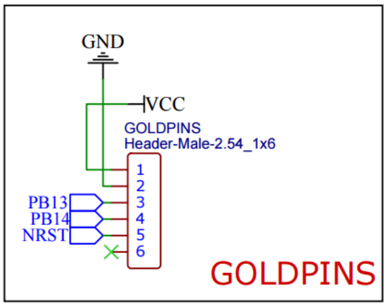
- **AMS1117-3.3V** : Linear regulator converting +5V into +3.3V for the MCU and peripherals.
- **Power Socket** : External power input (+5V).
- **100nF Capacitor** : Decoupling capacitor for voltage stabilization.



Goldpins (Header 1x6)

Main Role : Debugging or extension interface.

- Directly connected to STM32 (PB13, PB9, NRST, VCC, GND).
- Used for programming, debugging, or adding external modules.



<u>Conclusion</u>

The schematic can now be read as a **clear modular architecture** :

- **Video sources** (M.2 + HDMI) → pass through the **MUX** → sent to STM32.
- **Sensors & Communication** (Temperature, RS485, Fan) → managed by the **STM32**.
- **Stable power supply** → via **AMS1117 and Power socket**.

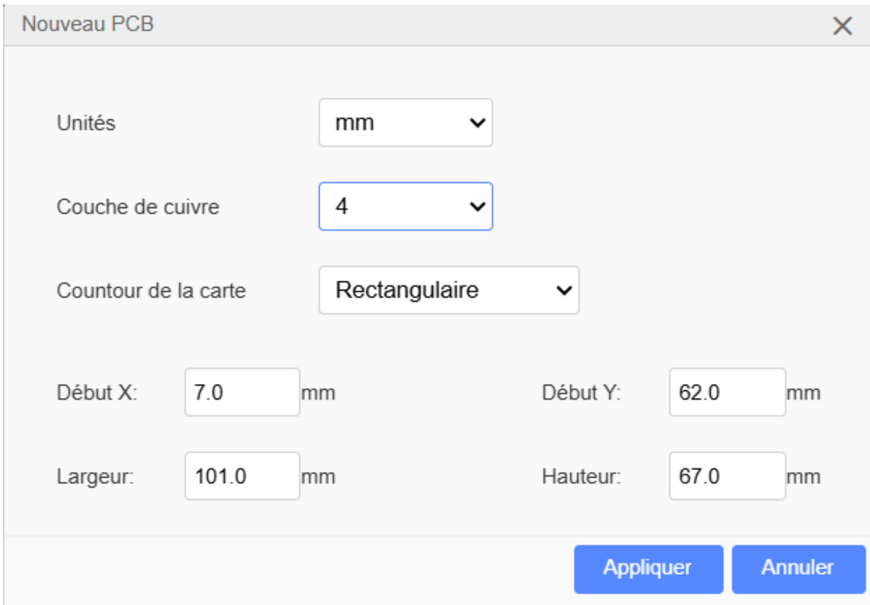
Resistors & capacitors

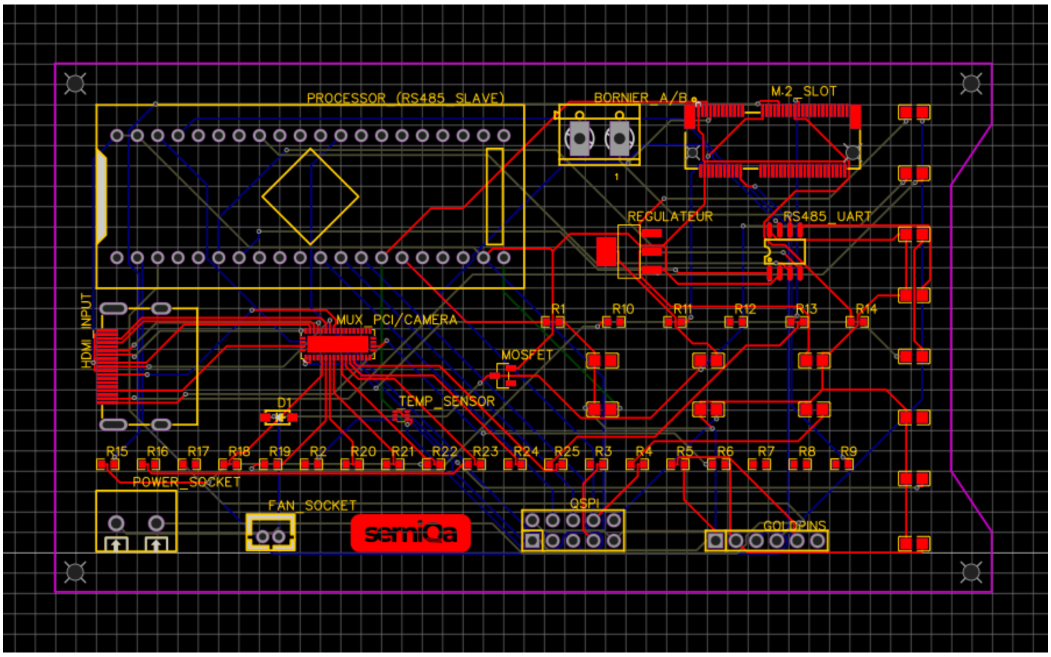
Component	Main Role	Detailed Description
Series resistors (typ. 33 Ω)	Termination / current limiting	Placed on high-speed signal lines (USB, HDMI, PCIe, M.2). Their role is to match PCB trace impedance to prevent reflections and slightly limit current to protect the drivers.
Pull-up / Pull-down resistors (10 kΩ, 4.7 kΩ, etc.)	Default logic state forcing	Keep a signal line at a defined logic state (high or low) when no active driver is present (RESET ENABLE, I2C SDA/SCL).

Decoupling capacitors	Power filtering and stabilization	Placed between VCC and GND as close as possible to IC power pins. They absorb current spikes and reduce high-frequency noise to stabilize the supply voltage.
Bulk capacitors (100 µF...)	Energy reservoir	Larger than decoupling capacitors. They stabilize the overall power rail (3.3 V, 5 V, etc.) and prevent voltage drops during high current transients.
Resistors on SDA / SCL (I2C)	I^2C bus pull-ups	Typically 4.7 kΩ. They pull SDA and SCL lines to VCC when no device drives them low. Without them, the I^2C bus would not function.
Filtering capacitors on USB / differential signals	Noise reduction	Sometimes placed in series or parallel with high-speed data lines (D+/D-). They filter high-frequency noise and protect against voltage spikes.
Capacitors on RESET	Reset delay	Provide a small RC delay to hold the system in reset for a few milliseconds at power-up, allowing the supply voltage to stabilize before releasing reset.
Series resistors on control signals (CLK, EN, etc.)	Current limiting / signal integrity tuning	They reduce overshoot and smooth transitions to avoid ringing and improve signal integrity on fast control lines.

C. PCB Layout & Routing

For the PCB routing phase, I generated an initial version of the trace layout while carefully respecting the design rules (DRC) and the required track widths for the different signal types. This first layout ensured the proper interconnection of all components across the board. I then refined the critical signals, particularly the HDMI and M.2 differential pairs, in order to guarantee impedance control and accurate length matching. This process allowed me to optimize both the electrical performance and the reliability of the final design.





After completing the routing stage, I generated the **2D model** of the PCB, which clearly illustrates the real placement of traces, power and ground planes, as well as the exact positioning of each component. This view helped me verify the consistency of the connections and confirm compliance with the routing rules.
Subsequently, I produced the **3D model** of the board, providing a realistic visualization of the final assembly. With this perspective, I was able to check the mechanical dimensions, the alignment of connectors (HDMI, M.2, power socket), and the compatibility with the mounting holes. This rendering is an essential step to anticipate integration constraints in an enclosure or hardware environment.

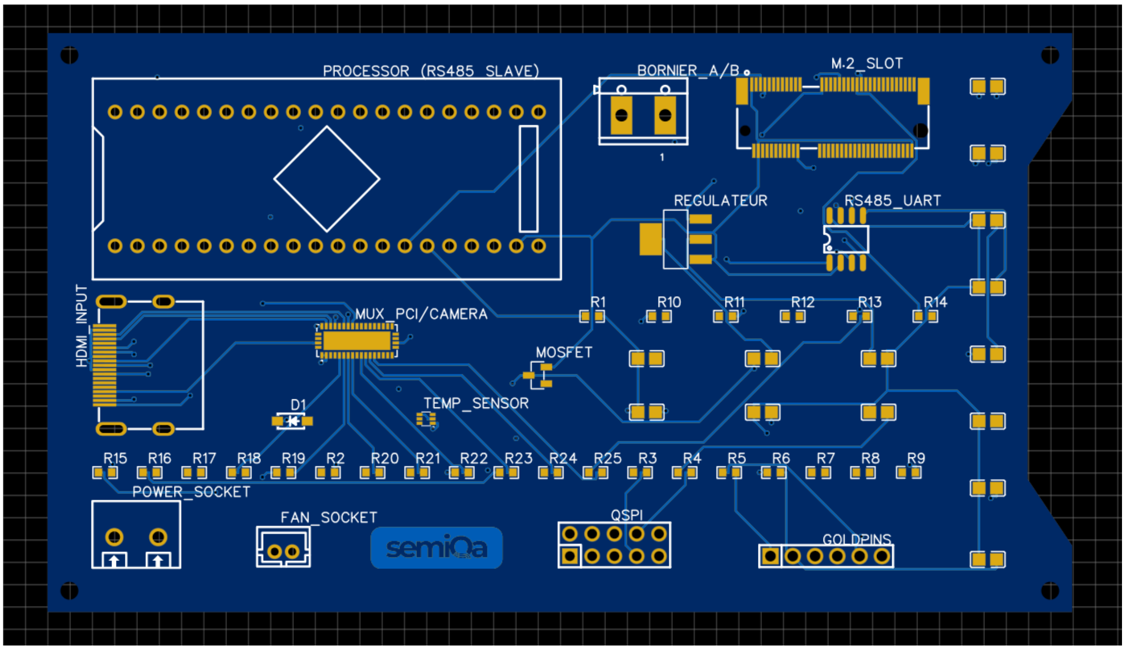


Figure 2 : 2D model

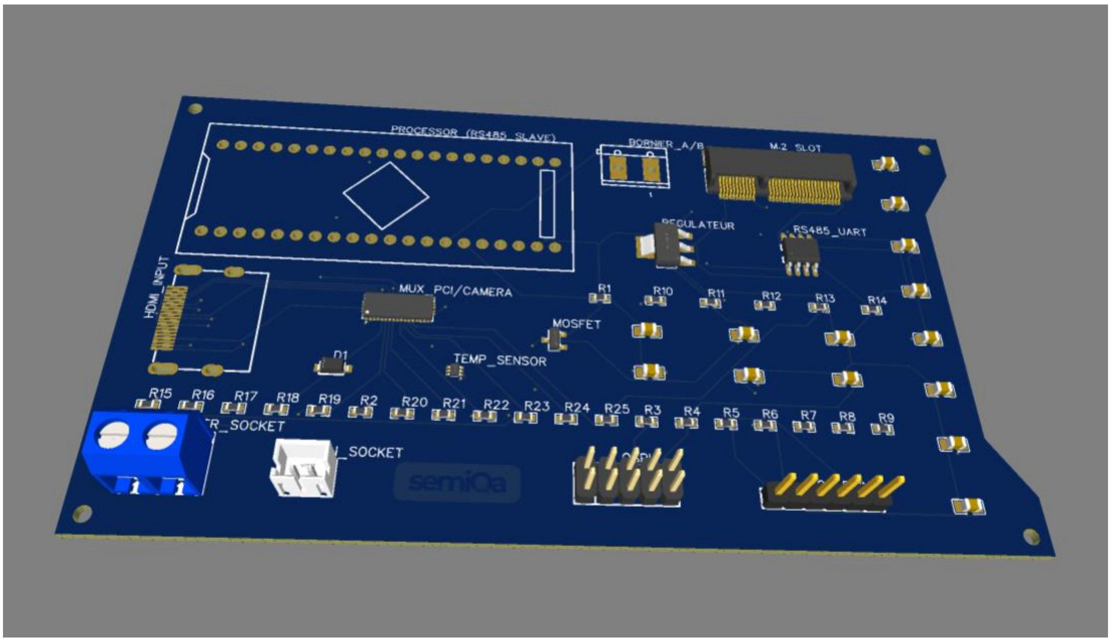


Figure 3 : 3D model

3. Case Design

After finalizing the PCB design using EasyEDA, I utilized the 3D model generated by the software to design a suitable enclosure. This step was crucial to ensure that the PCB would be properly protected and easy to handle in a real-world environment. The following steps were carried out to design the enclosure based on the EasyEDA 3D model :

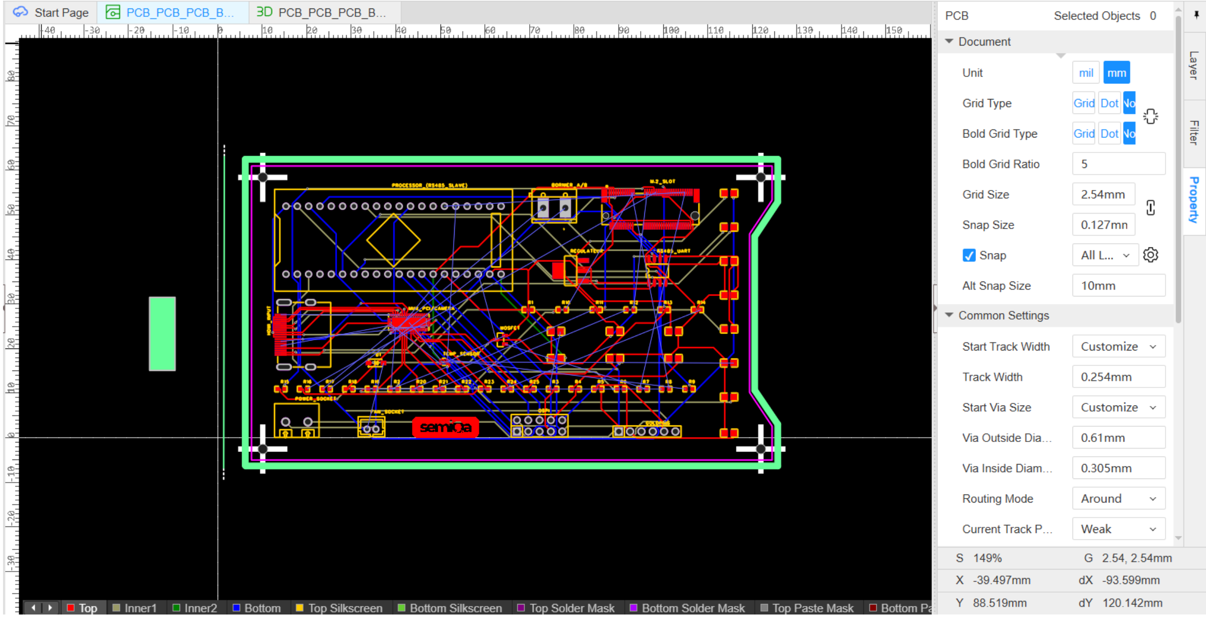
1. **Exporting the PCB 3D Model**

Once the PCB design was completed in EasyEDA, I exported the 3D model of the board in STEP or STL format, which is widely compatible with most CAD software. This 3D model includes all the components and their exact positions, allowing me to create an enclosure perfectly fitted to the board's dimensions.

2. Definition of Dimensions and Constraints

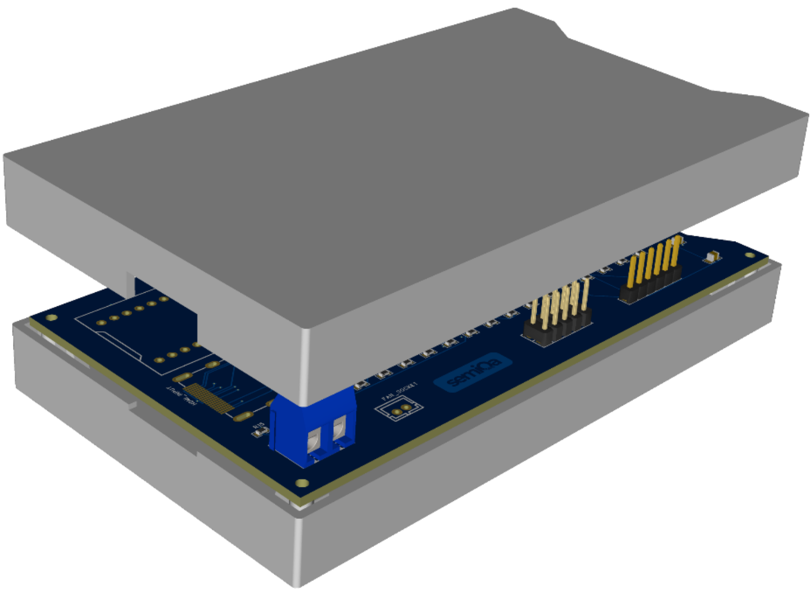
I added dimensional annotations to each view to indicate the exact dimensions of the enclosure. This included:

- **Required Clearance :** I measured the height of the tallest components (such as connectors or heat sinks) to ensure there was sufficient internal space for proper fit and to avoid mechanical stress.
- **Opening Alignment :** I positioned the openings for connectors, buttons, and cables according to the actual placement of these elements on the PCB to guarantee easy access and proper functionality.
- **Mounting Points :** I designed dedicated locations for screws or clips to securely hold the PCB inside the enclosure, ensuring stability during use and transportation.



3. Case 3D Design and Validation

After defining the dimensions and constraints, I created the 3D model of the enclosure. The base was designed with enough wall thickness and standoffs to hold the PCB securely, while the cover was modeled to fit precisely and leave space for the tallest components. Side openings were added for the HDMI connector and other sockets, and dedicated holes were included to match the PCB mounting points for screw fixation. Ventilation slots were also integrated to avoid overheating, and a simple closing system using screws or clips was designed to make the assembly practical. Finally, a virtual assembly was carried out to check alignment and accessibility, and renderings were generated to visualize the final design before prototyping.



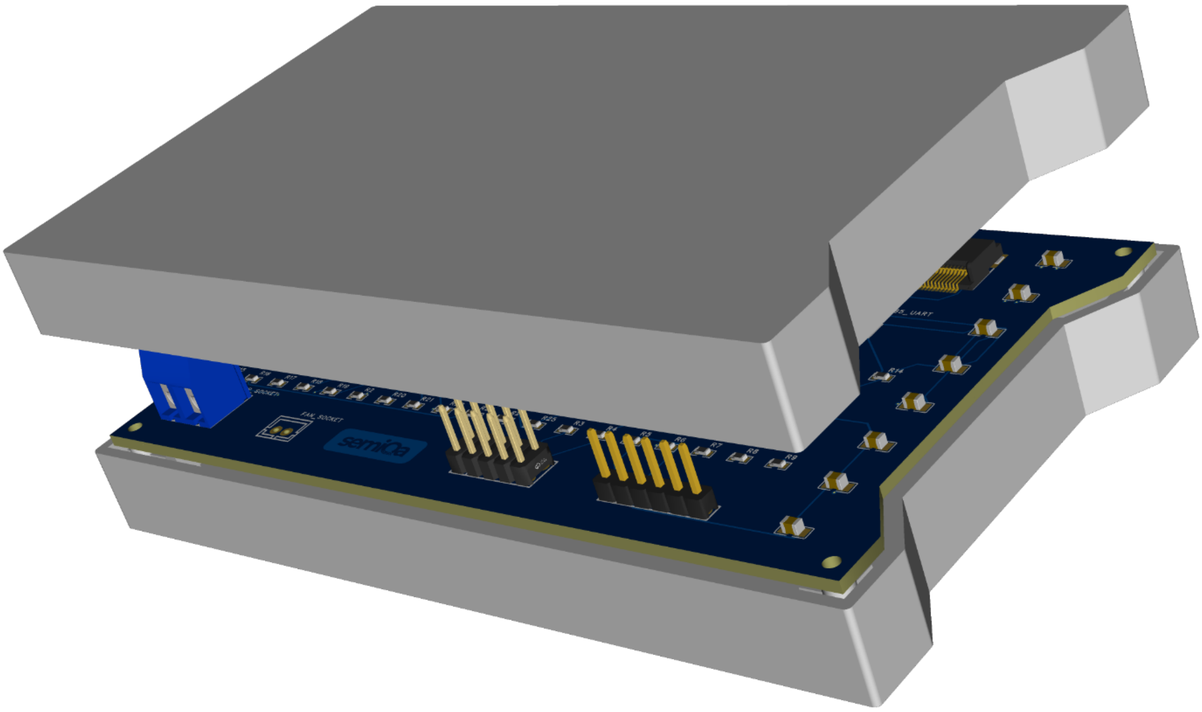


Figure 4 : Card-Case

Conclusion

Through this project, I was able to design and implement a complete electronic system that integrates several key functions, including HDMI input, M.2 connectivity, a high-speed multiplexer, RS485 communication, temperature monitoring, and fan control. I successfully created the schematic, selected appropriate components, and generated both the 2D and 3D PCB models. This work allowed me to apply my knowledge of circuit design, signal integrity, and PCB routing while gaining practical experience with design tools. I also learned the importance of protecting high-speed interfaces with ESD components and ensuring proper power distribution with decoupling capacitors. Overall, I am satisfied with the outcome of this project, as it strengthened my technical skills and gave me valuable insights into the design process of advanced embedded systems.