GAMEBOY CAMERA CLUB

A DESIGN GUIDE BY 2BITWIZARD

Design Guide

February 3, 2023
Version II.

Contents

1 Introduction

This document will cover all the design decisions made in the process of engineering this micro-camera flashcart. It will cover the reverse engineering process, the part selection process and the PCB design process. This project was started to create an ultra-compact GameBoy Camera flashcart to fulfil various purposes. Some of its use cases are as follows:

- A compact version of the camera usable in the GameBoy Advance or other low-profile applications in other GameBoy models.
- A full size camera design with enough free space to mount another piece of hardware on the cartridge using a flex PCB.

2 Reverse Engineering

Reverse engineering started off with the following tools:

- Fluke 179 True RMS Multimeter
- BM-51-2 Stereo Microscope
- Amaran F1 LED panel

The original GameBoy Camera PCB was placed under the microscope and every single leg was probed and then promptly plotted out in a KiCad schematic file. This set up a basic net of connections between the components. The symbol pins were then named based on datasheet information where possible for the following components present on the GameBoy Camera:

- U3 - M5M51008BKV (Correct datasheet) [1]
- U2 - MX23C8004E-12 (datasheet from same family) [2]
- U1 - MAC-GBD (no datasheet available)
- J1 - M64282FP (Correct datasheet) [3]
- Cartridge edge connector [4]

While this covers most of the pins on these chips there are still some pins to figure out at this stage, mainly related to the circuitry present on the top right of the GameBoy Camera cartridge. This circuitry is related to the coin cell battery present on this PCB. After probing around for voltages and diodes it was determined that this circuitry uses the $\overline{RST}$ signal from the edge connector to regulate the standby mode of the SRAM chip. In detail this is done using an NPN mosfet and a dual BJT chip utilising a PNP and NPN BJT transistor, most likely to achieve the longest battery life possible. Above this piece of circuitry is one final piece of circuitry which mainly consists of a 3V0 voltage regulator and an electrolytic capacitor. Finally there are some stray components on the PCB which were easily figured out, by reading the silkscreen present on the PCB and occasionally using the multimeter for quick measurements. It has to be noted that the capacitors present on the original board have not been measured, instead the values of capacitors on the custom PCB are determined based on previous experience.

3 Selecting components

3.1 FRAM

To get rid of the battery present on the original GameBoy Camera a different type of RAM had to be used compared to the SRAM present on the original cart. This leaves three unique options to

consider: FRAM, MRAM or NVRAM. The datasheet of the original SRAM dictated that I needed a RAM memory chip with 1MBit capacity arranged in a 128kx8 configuration. It also needs to run at a 3V supply rail. Finally it should match or beat the 120ns access time of the original chip. With these things in mind I filtered through all memory categories and came across three chips that could potentially work.

- FM28V100-TG (FRAM)
- CY14B101LA-SP25XIT (NVRAM)
- MR0A08BCMA35 (MRAM)

Unfortunately the MRAM option was out by default, because it was only offered in a BGA package. The NVRAM option looked compelling enough, it even being offered in an SSOP package. Unfortunately it lists its minimum voltage at 3V, which is pushing the boundaries a little too much. The FM28V100-TG FRAM was ultimately chosen, because it matched both the footprint and pinout of the original SRAM memory chip. This is a massive advantage over the other chips, because the MAC-GBD pinout appears to be designed with that exact SRAM pinout in mind. Finally while the FM28V100-TG does have a typical voltage rating of 3.3V, its datasheet mentions a minimum supply voltage of only 2.0V, which is firmly below the 3.0V present in my design.

3.2 Flash

To be able to make a flashcart, you do in fact need flash memory. Flash memory is a bit of a tricky one, because it has some other requirements beyond its technical compatibility with the MAC-GBD chip. The original ROM present on the cart is an 8Mbit chip arranged in a 1Mx8 configuration, running on a 5V0 input. Unfortunately there are no chips currently available for sale on Mouser that match these criteria. This meant I had to turn to the internet to find out which Flash chips were commonly used on flash cartridges for all kinds of systems, looking for chips that fit the bill. I ended up with a selection between two chips:

- AM29F080B
- AM29F016B

The AM29F080B has the exact amount of memory required to match the original ROM and also presents itself in the smaller footprint of the two. The first option would have allowed me to store two different ROMs on the flash cart, switching between them using a physical switch. This would have taken up too much space on my compact cart. As mentioned before, there are requirements beyond the technical compatibility with the MAC-GBD chip. It has to be easily flashable with readily available software when mounted on the cart. This is also the reason why the AM29F080B was chosen. It has seen previous use on flashcarts and therefore seems to be natively supported.

3.3 Other Components

Voltage Regulator The voltage Regulator has been replaced with a more modern LDO, which produces the same 3V0 output. The LDO with the best output has been chosen, though this is not strictly necessary. The electrolytic capacitor near the regulator has been replaced with a tantalum capacitor to minimize vertical obstruction within the game cart.

EMI filter The DC resistance of the ferrite beads present on the original cart has been measured. This has been used to select equivalent ferrite beads to solder onto the micro camera cart.

Diode The forward voltage of the diode D2 on the original cart was determined to hover around 215mV, so a similar schottky diode with a similar footprint has been selected to take its place.

This diode features a slightly lower voltage drop around 190mV, which should only benefit the circuitry.

Battery circuit The original battery circuit along with the battery has been removed and replaced with a single signal inverter. This takes the $\overline{RST}$ signal and outputs an inverted signal with an amplitude equal to the 3V0 supply used for this component. This should replicate the functionality of the original battery circuit putting the SRAM, or in this case FRAM, into a standby mode.

Capacitors All decoupling capacitors have somewhat arbitrarily been chosen to have a value of 10nF, except for one lone capacitor near the 5V0 pin of the edge connector. A single 100nF capacitor has been placed here as would be standard when decoupling circuitry.

Passives All passives have been changed to use an 0603 footprint. This makes the board more friendly to solder by hand.

4 PCB Design

To make this micro camera as compact as possible the routing of this PCB had to be as compact as possible. First off the MAC-GBD chip was placed as close to the bottom of the PCB as possible, restricted by two factors. The original cover for the GameBoy Camera has a plastic ridge which covers the connector pins. The decoupling capacitor still had to fit between this plastic ridge and the MAC-GBD chip. The address and data traces also still had to fit below the MAC-GBD chip with sufficient clearance. After this the MAC-GBD was shifted as far right as possible while still allowing the FRAM address lines to reach the MAC-GBD chip and wrap around a decoupling capacitor present inbetween the two chips.
Then the FRAM chip was placed, which was placed as close to the notch in the PCB as possible and as far right as possible. With these two chips in place the other address and data lines for the FRAM could be drawn. This would eventually determine the final height of the PCB as the sensor connector would still have to fit above these address and data lines.
The flash was placed evenly spaced between the notch on the left and the top of the PCB as far left as possible. This was done to allow as much space as possible between the decoupling capacitors and the flash chip.
With these major components in place all other components were placed inbetween with a similar layout as seen on the original camera PCB. After placing the main three components and routing a few traces to determine the height of the PCB all other traces were done. For concistency the address and datalines going towards the MAC-GBD chip were all placed on the front-side copper layer, with the data and address lanes splitting off to the flash memory being partially placed on the back-side copper layer. This only happened where necessary. This was done to ensure a large groundplane on the back of the PCB. The advantage of this large groundplane is that it minimizes hops through vias to ground each part of the PCB properly. Just like the original PCB a 5V0 loop was constructed on the back of the PCB, supplying power to all components that require it.
While there is much more that went into the actual design of this PCB when it comes to the thousands of choices made when routing, the countless adjustments, zoning problems and general clearance headache, this should detail the main important decisions made when it comes to the unique design of this PCB. The last noteworthy thing is the footprint for the sensor's JST connector. This footprint was created to support both a horizontal as well as vertical connector. The cart might not stick out of some GameBoys enough to mount a vertical camera connector, in which case a horizontal connector might have to be used.

5 Conclusion

This has been quite the adventure, to design an extremely compact GameBoy Camera flashcart. Reverse engineering was suprisingly simple. Finding components to make a flashcart was also taken care of rather quickly. I did however underestimate the amount of effort that would go into designing the PCB for this project. The PCB went through two major revisions. The first revision was scrapped after receiving some fair criticism. Instead of trying to resolve this criticism I decided to start over from scratch, believing I could do better than I did in this first revision. This lead to the second and current revision of this board. A prototype of the PCB is currently being produced as this is written and should be underway soon. The PCB successfully passes DRC and JLCPCB cleared the gerbers for production.

References

[1] "M5M51008BKV Ram datasheet," M5M51008BKV Datasheet | Mitsubishi - Datasheetspdf.com. [Online]. Available: https://datasheetspdf.com/datasheet/M5M51008BKV.html. [Accessed: 03-Feb-2023].

[2] Alldatasheet.com, "MX23C8000 datasheet," ALLDATASHEET.COM - Electronic Parts Datasheet Search. [Online]. Available: https://pdf1.alldatasheet.com/datasheet-pdf/view/74426/MCNIX/MX23C8000.html. [Accessed: 03-Feb-2023].

[3] Alldatasheet.com, "M64282FP datasheet," ALLDATASHEET.COM - Electronic Parts Datasheet Search. [Online]. Available: https://www.alldatasheet.com/datasheet-pdf/pdf/146598/MITSUBISHI/M64282FP.html. [Accessed: 03-Feb-2023].

[4] M. Fayzullin, "Hardware," GameBoy Hardware. [Online]. Available: https://fms.komkon.org/GameBoy/Tech/Hardware.html. [Accessed: 03-Feb-2023].