

# GAMEBOY CAMERA CLUB

A BUILD GUIDE BY 2BITWIZARD

---

## Build Guide

---

August 25, 2023  
Version I.

# Contents

<b>1</b>	<b>Introduction</b>	<b>2</b>
<b>2</b>	<b>Ordering the PCB</b>	<b>2</b>
<b>3</b>	<b>Ordering Parts</b>	<b>2</b>
<b>4</b>	<b>Assembling the PCB</b>	<b>3</b>
<b>5</b>	<b>Troubleshooting</b>	<b>4</b>

## 1 Introduction

This document will cover the build process of the 2BitWizard flashcart. It will go over part selection, the ordering process and the actual build process. This document should only be used as a guideline, common sense will still be necessary to successfully complete this project. Following this guide will also not guarantee that your flashcart will work, user error, defect parts and other potential problems may still arise.

## 2 Ordering the PCB

There are a plethora of sites available to order PCBs from, the 2BitWizard flashcart however is designed with JLCPCB in mind. It complies with the tolerances of JLCPCB's manufacturing process and also contains a line of placeholder text below the flash chip, which JLC uses to mark the PCB with a production number. The PCB design should comply with the tolerances of many more common (Chinese) PCB manufacturers, but this guide will only cover the JLCPCB ordering process.

The process starts by downloading the correct (usually latest) gerber zip package off the 2BitWizard GitHub. At the time of writing there are two options available, a normal model and a "longboard" model. The longboard model contains a long neck on top of the PCB with a second connector on it. This PCB can be used to mount this project in a traditional GameBoy camera shell. This longboard model is designed to have its neck sawed off if or when you do decide to opt for a smaller cartridge.

After downloading the files they should not be unzipped. The zip file itself should instead be uploaded to the JLCPCB website. This will redirect you to the ordering screen. At this point you will be presented with a lot of options. Below is a large list of all the options and how they should be set. JLCPCBs ordering website may change over time, this list is only guaranteed to be complete at the time of writing, see table 1.

## 3 Ordering Parts

Now that the PCB has been ordered, it is time to select parts for your PCB. There are many suppliers available for all the different parts required. The viability of parts suppliers is highly dependent on your location. This part of the process might be tricky if your parts supplier does not provide you with a comprehensive filtering system. Common parts suppliers would be Mouser, Digikey and Farnell and AliExpress.

Start off by downloading the BOM.xlsx file from the GitHub. This file contains all the parts required for this project. Some of the parts on this BOM have specific parts names/numbers, other parts are more generic. It is recommended to buy matching part names/numbers as listed in the BOM. If these are not available you may try to find alternatives at your own risk by comparing datasheets of various other parts to the datasheet of the original part. Alternatively you could simply ask someone knowledgeable to do this for you.

To find these parts you can use the websites mentioned above to look for these parts, I personally used Mouser to source most of my parts. There are a few parts which are most easily acquired through AliExpress however. The flash memory chip is almost exclusively available on Aliexpress, the JST connector is easiest to find on AliExpress and the FRAM chip is cheapest on AliExpress. Generic components, like capacitors and resistors, may also be acquired through AliExpress

Table 1: An overview of JLCPCB options

Base Material	FR-4
Layers	2
Dimensions	Auto-filled (leave be)
PCB Qty	As desired
Product type	Industrial/Consumer electronics
Different Design	1
Delivery format	Single PCB
PCB thickness	0.8
PCB Color	As desired
Silkscreen	As desired
Surface Finish	ENIG
Outer Copper weight	1 oz
Via Covering	Tented
Board Outline Tolerance	±0.2mm(Regular)
Confirm Production file	No
Remove Order Number	Specify a location
Flying Probe Test	Fully Test
Gold Fingers	Yes
30°finger chamfered	No
Castellated Holes	No

at your own risk. If you're not using AliExpress, but instead a website like Mouser, it's easiest to make use of a filtering system. There are millions of capacitors and resistors out there. Most websites with a filtering system allow you to fill in certain parameters to narrow your search. It may help you out to filter by value, footprint and even an attribute like rating (X7R for the capacitors for example). Once you have found your parts, please double or even triple check that all values match the values listed in the BOM. Some parts may be available in multiple package options, in which case it is necessary to check that your part matches the footprint in the BOM. Finally it is recommended to buy enough parts to make mistakes during assembly. Buy more than you need, in case you break something.

It is important to note that the JST connector in the BOM is available in two options, a horizontal and a vertical version. It is recommended to buy a horizontal version. At the time of writing there are no custom shells available which make use of a vertically oriented connector.

## 4 Assembling the PCB

Assembling the PCB is where most people mess things up. This part requires the right tools and a considerable amount of skill. If you are not prepared to do your own soldering, contact someone within the community to do this for you. It is extremely easy to damage your valuable parts or the PCB. Assembling the PCB actually starts by disassembling the original GameBoy Camera PCB to harvest the MAC-GBD chip. This is most easily done with a hotplate set to roughly 230 celsius, it can however also be achieved with a heatgun or even chipquik, but these methods are inferior and not recommended.

Now to start the actual assembly process. It is recommended to use a high quality soldering iron with a fine tip of your choice and a high quality flux. While the parts may be placed on the

PCB in any order you like, there is an optimal part order for assembly when it comes to ease of soldering. Start off by soldering the MAC-GBD chip in the center of the board. This is followed by the flash chip and the SDRAM. Then follows the parts cluster on the top right of the PCB, values for each reference can be found in the BOM. Start off by soldering C15, C17, and U2. After this you may install U4 and C12. C16 is an optional capacitor, some alternate voltage regulators require this capacitor in their pinout, like the original GameBoy Camera regulator. The regulator specified in the BOM does not require a C16 capacitor to be mounted, in which case the pads may be left exposed. After finishing up the cluster of components in the top right you can solder all other components, saving J2 for last. J2, the connector which houses the sensor cable has a fairly complex footprint. This is because there are two available options for your connector. When using a vertical connector, solder its pins to the top row of pads (the pads closest to the PCB edge), when using a horizontal connector, solder its pins to the bottom row of pads. (the pads closest to the J2 silkscreen text), when using the longboard version of the PCB, leave the footprint on the main portion of the PCB alone and exclusively install a vertical JST connector on the extended part of the PCB.

After wrapping up the soldering process, plug in your PCB once more to verify that the RAM is at the very least detected. Detected RAM chips are still not guaranteed to work, so after this it is time to either flash the stock ROM or a custom ROM to the GameBoy Camera and boot it up with the sensor board attached. Observe the screen and check for any artifacting in camera mode. If the camera runs fine you've just made it. If it displays some sort of artifacting in the image you're most likely dealing with a poor connection between the MAC-GBD and the FRAM chip/

## 5 Troubleshooting

If the camera is not working as intended there are a few easy things to check before seeking help from third parties online, like a Discord server. If your cart is not recognized at all when trying to flash a ROM, you will have to check each and every connection to verify that there are no shorts, floating pins or cold joints. If the cart detects the flash memory but has trouble writing to the actual flash memory, take a close look at the pins on the flash chip (left side of the board) and the left/bottom side pins of the MAC-GBD chip. If your camera does flash software and boots into the game and then suffers from artifacting in the camera image you're most likely dealing with FRAM issues. Please closely inspect the pins on the FRAM chip (right side of the board) and the top/right side pins of the MAC-GBD.