Please read the whole “guide” and BOM first before you start to work with the files and the wheel!

Electronics

PCB variants

There are 3 variants of PCB design available:

- **Normal PCB v2**
  - This is the latest recommended version of the PCB
  - The whole PCB was redesigned to make the manufacturing easier at JLCPCB: all the components are on one side.
- Normal PCB (This is the original version of the PCB for the GTE PRO wheel)
  - Has all the electronics (USB HUB, FT232 for screen, FT232 + atmega328 for LEDs)
- Simplified PCB version
  - This is basically a daughter board with the buttons, rotary encoders, LEDs

I created the simplified version from the original PCB, but removed all the components except the buttons, rotary encoders and LEDs. I connected them to 2.54 pin header, so you can use this board as a simple daughter board and can connect your well-proven solution you already know (eg. arduino, leo bodnar, etc.) .
Of course in this case you need to add also USB HUB, USB-UART converter, to run the screen and the buttons as well.

If you never soldered SMD components (0805, LQFP48, SSOP20, …) or don’t really understand the electronics/schematic of the standard PCB, then I recommend to use the simplified one or the V2 PCB version ordered 100% assembled from JLCPCB or PCBway. Because if there is a solder error or something, could be hard to debug the PCB.

I recommend the full PCB only for those, who have some electronics experience, have electrical background and who understand the schematic of the PCB.

PCB versions

- v1.0 – I’m using this version of the PCB (note recommended - removed from latest package)
  - Known issues:
    - All the connectors needs to be soldered no the bottom (don’t have space on top)
    - The crystal (Y3) for atmega328 needs to soldered on the bottom (don’t have space on top)
  - Analog inputs are not used, so don’t need to be populated (U2, J14, J15, …)
    - You can use it for clutch with firmware modification (source files attached)
  - Recommended to solder 10kOhm resistors on top of C35 and C38 capacitors. (For USB communication stability between USB HUB and STM32)
- v1.1 – Fixed issues form v1.0 (note recommended - removed from latest package)
  - Connectors and crystal (Y3) moved to bottom
  - Analog input components removed from BOM
  - Recommended to solder 10kOhm resistors on top of C35 and C38 capacitors. (For USB communication stability between USB HUB and STM32)

- v1.2
    - 10k pull-down resistor added parallel with C35, C38, so don't need to solder on top of these resistors
    - Added programmer header for atmega32 to upload bootloader
    - BOM components updated
- **V2.1**
    - Full redesigned PCB for easier, cheaper manufacturing.
    - **For further guidance see the GTEPRO manual**

PCB manufacturing

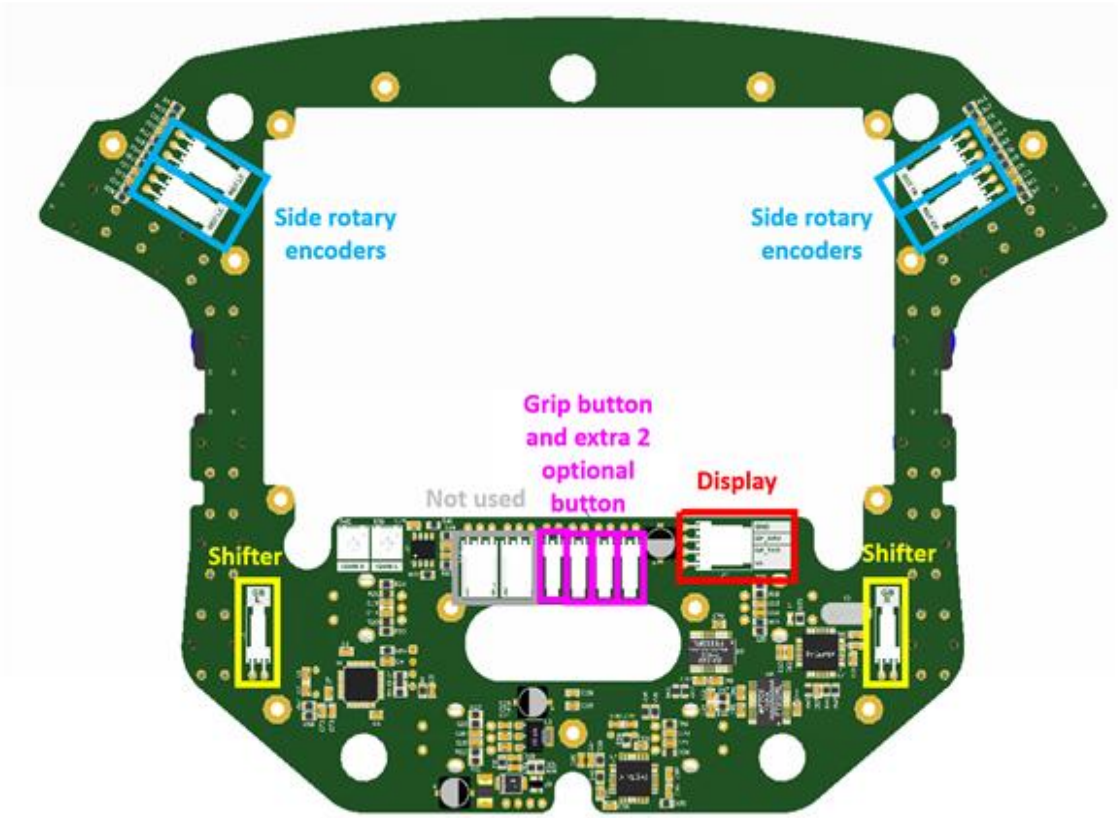
My recommendation is to use **JLCPCB.com** or **PCBWAY.com**.

JLCPCB is cheaper, but don't has all the components, so you will need to solder some for yourself.

Electrical specification

- Always use the PCB from 5V USB (abs max. 5.5V, min 4.5V (measured on PCB)
- PCB needs min. 1A current (recommended to use from powered USB hub with <1m cable)
- Don't recommended to use the LEDs over 30% brightness (in SimHub) (at 10% is already enough bright I think)
- Handle the PCB with care and protect from ESD
- **Don't touch WS2812B RGB LEDs, they are very sensitive!**

PCB Connections

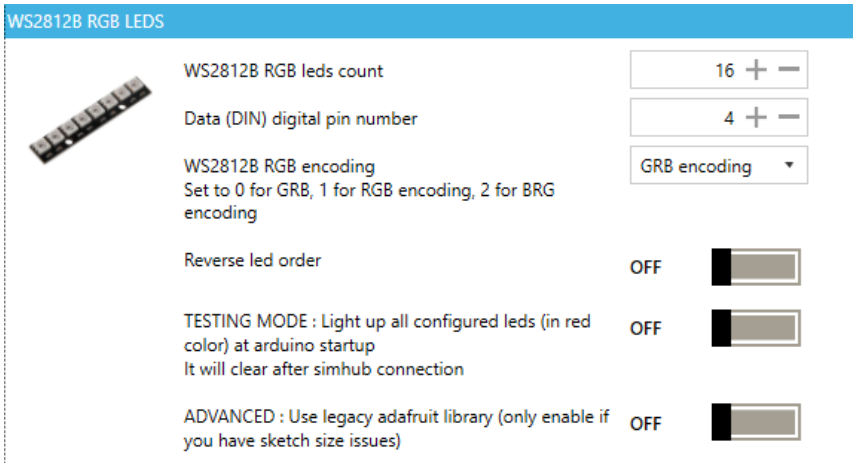


For the rotary encoders connect the middle pin (GND) to the connector middle pin (GND) the side pins can be swapped.

Assembly tips for PCB

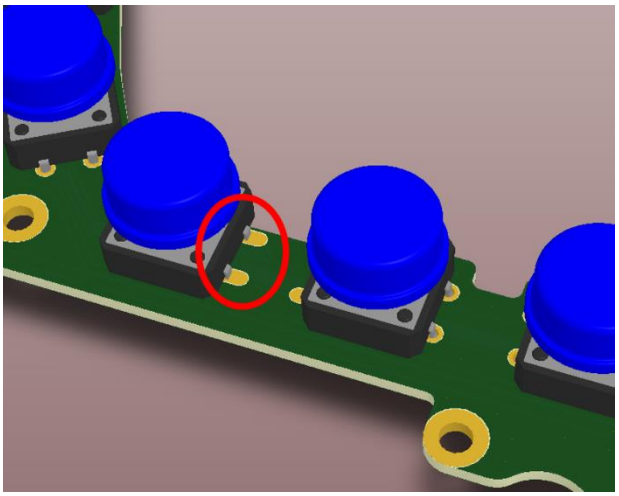
1. General tips
    - I recommend to check the PCB for short circuit after all the steps
    - Always check the polarity before you connect power to the PCB (and if you connect the screen to the PCB)
    - If you find an issue after one step don't go further, debug the problem
    - Never solder while the board is powered up
2. Recommended to solder first all the passive 0805 components
3. Solder all the parts of the power input (USB IN) and the voltage regulator
4. Measure the current consumption of the board and the 3.3V power supply. (If there is no 3.3V or the power consumption is high, then search for solder errors).
5. Solder the STM32 and the USB HUB IC and the parts which belong to them
    - USB HUB won't work without the STM32 because it needs reset signals which is provided by the STM32
6. Programming
    - Connect your programmer (STLINK V2, STM32 discovery/nucleo board) (SWDIO, SWCLK, NRST, GND signals)
    - Power up the board (5V to "USB IN")
    - Upload the provided .hex firmware with "**ST-Link Utility**" or "**STM32CubeProgrammer**"
    - If everything is OK the PC should recognize the board as USB HID Game Controller
7. Solder the FT232 (and the parts belongs to) and test it: connect the PCB again to the PC and test the FT232 with terminal (short the RX to TX pin). If everything is fine you can connect your display and test it. (Double check the polarity.)
8. Solder the FT232 and Atmega328P for the RGB LEDs (Atmega328p should have already Arduino Nano bootloader, so the best is to remove one from an Arduino and solder it on the GTE PRO PCB)
    - GTE PRO PCB v2 and v1.2 already has programmer pins for atmega328p. So if you atmega doesn't have bootloader, then you can upload with this connector. You can use another Arduino as AVR ISP programmer. More details: https://docs.arduino.cc/built-in-examples/arduino-isp/ArduinoISP
9. Connect your PCB again to the PC. Now you should se the game controller device and 2 new COM port (FT232).

10. With SimHub you can create the firmware for the Atmega328P
    - Arduinio page -> My Hardware -> Open Arduino setup tool -> Select Arduino Nano
    - You can name the device as you want
    - Set the WS2812 LEDS to D4:

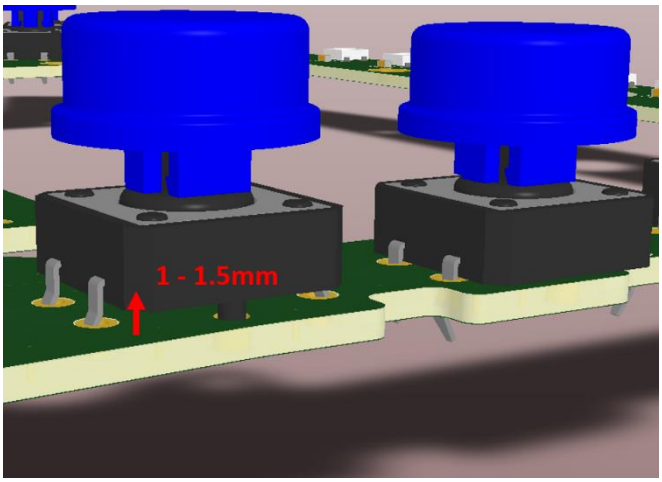


    - Select the COM port for the Atmega328P
    - Hit Upload
11. Solder the WS2812B LEDs and connect the PCB to the PC. Now you can test the RPM LED bar with SimHub
12. Solder the rest components (buttons, encoders, connectors)

Additional information for PCB



Some of the buttons have pads without hole. It's no error/bug, it's because on the other side is the side rotary encoder and would short circuit the button. On those pads you need to cut the pin of the button and solder like SMD component.



The buttons cap stands of in normal case ~1mm from front plastic cover part. If you want to stand off more, than you need to lift 1-1.5mm the buttons and solder in this state. I recommend to put something (eg. print a spacer plate) under the button, solder the button and then remove the plate.

Plexiglass (acrylic glass)

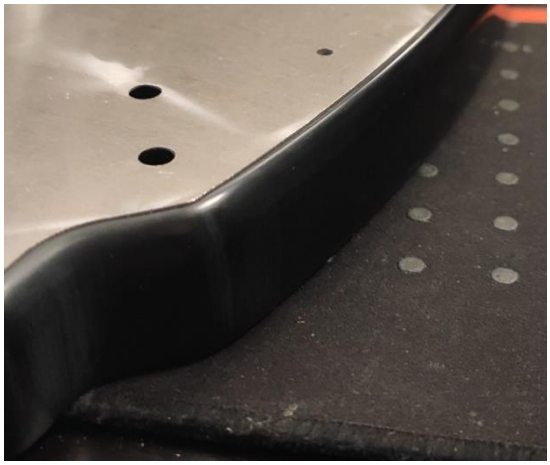
–The front acrylic for the display should be laser cut from 2 or 3mm (as you prefer) thick water clear material.

To have Porsche text and black background plexiglass can be direct printed. For direct printing all the files are attached in many format (DXF, SVG, PNG). Also help for the manufacturer (how should look like and which parts should be transparent/white/black).

I mounted the plexiglass with double sided tape cut to the acrylic shape.

3D printed parts

Side walls (Mainhousing-top, Mainhousing-bottom)



- 0.1mm layer height
- 100% infill (3mm wall)
- PLA
- Post-production: sanding with following grits: 120, 200, 280, 400, 600, 800, 1200, 2000 and then polished it. There is no paint on it.
- I recommend to sand under/with water.

Girps

- 0.2mm layer height
- 60% infill + 100% infill under the screw heads
- PLA
- Covered with seude leather (glued with Pattex universal glue and superglue on the edges)
- Check the button for right orientation before you connect glue in. (with front plate cutouts)

Shifter 3D printed parts

- 0.2mm layer height (0.1mm for nicer result without sanding)
- 100% infill
- PLA
- No post-production

Button plate

- 0.05mm layer height
- SLA black
- Post production: coated with acryl lack spray

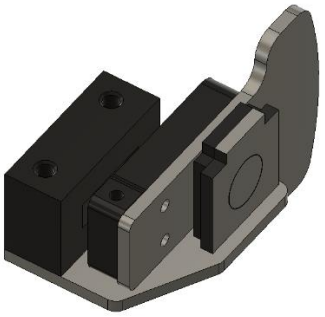
Rotary knobs

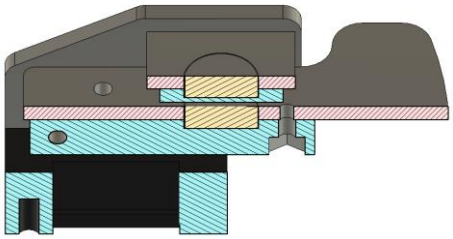
- 0.05mm layer height
- SLA black
- Post production: sanded with grit 1200

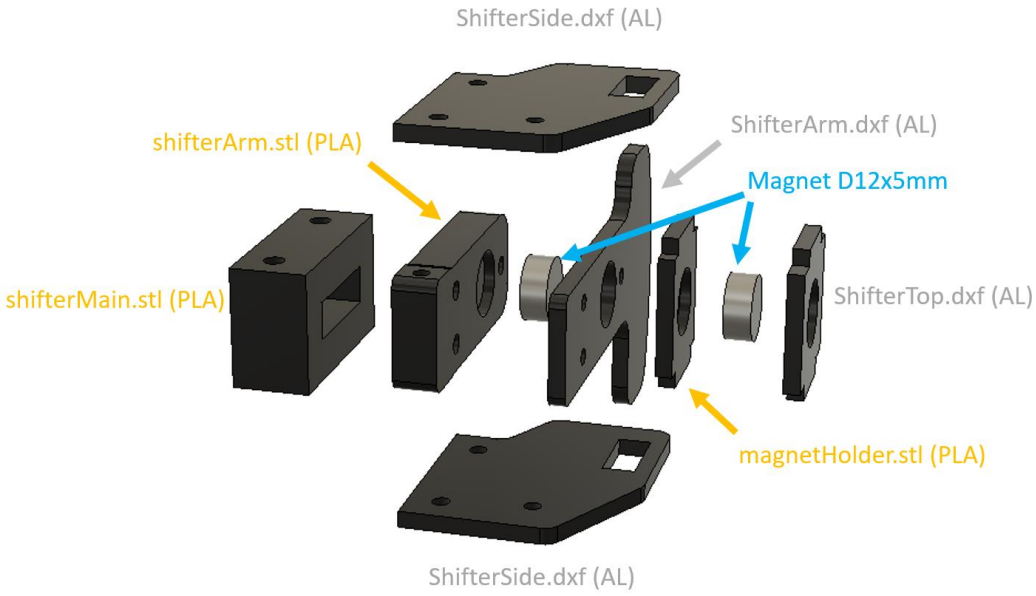
Shifter design

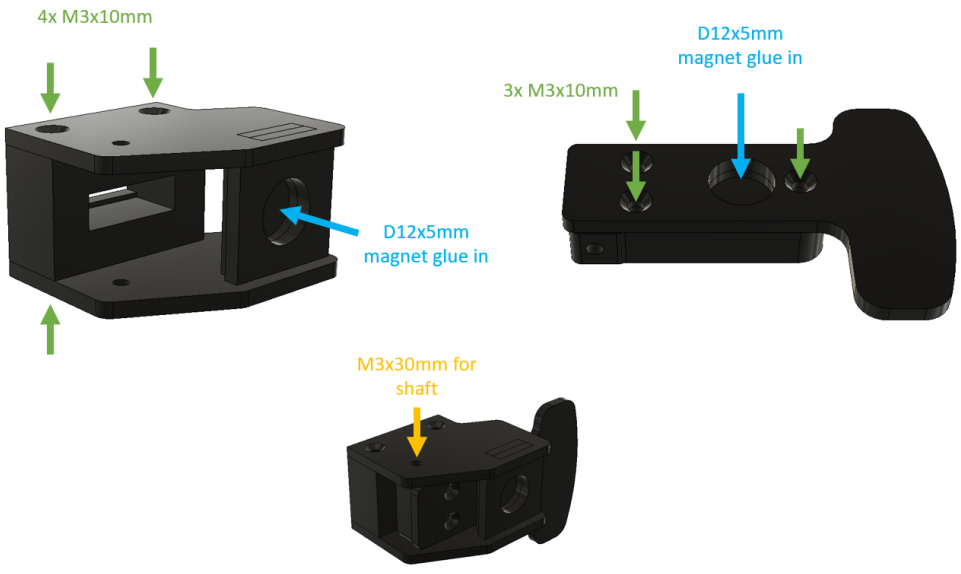
The main structural parts of the shifter made from 3mm aluminum.

The shifter main (3D printed from SLA) needs 7x threaded insert. I recommend to insert it with solder iron.





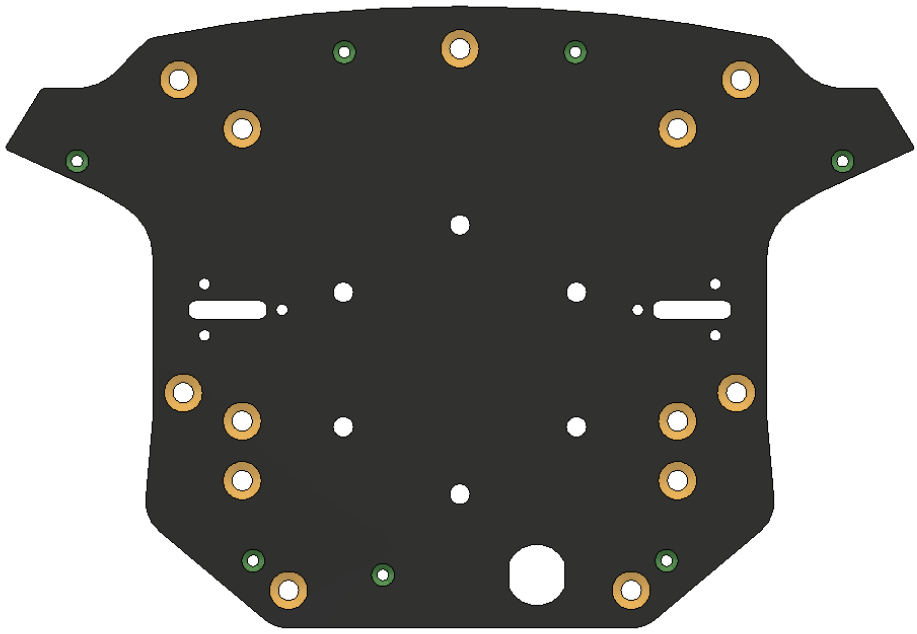




Holes to countersunk

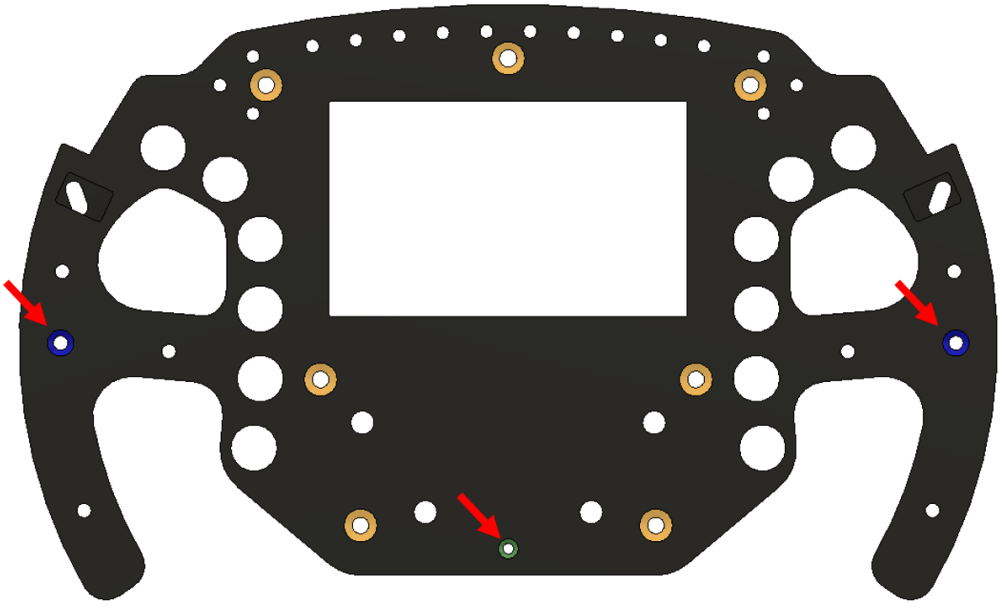
Yellow mark – Countersunk for M5 screw
Blue mark – Countersunk for M4 screw
Green mark – Countersunk for M3 screw

Backplate:



(You can also countersink the 2x3 holes for the shifter from inside if you wan't to use countersunk head screw for it.)

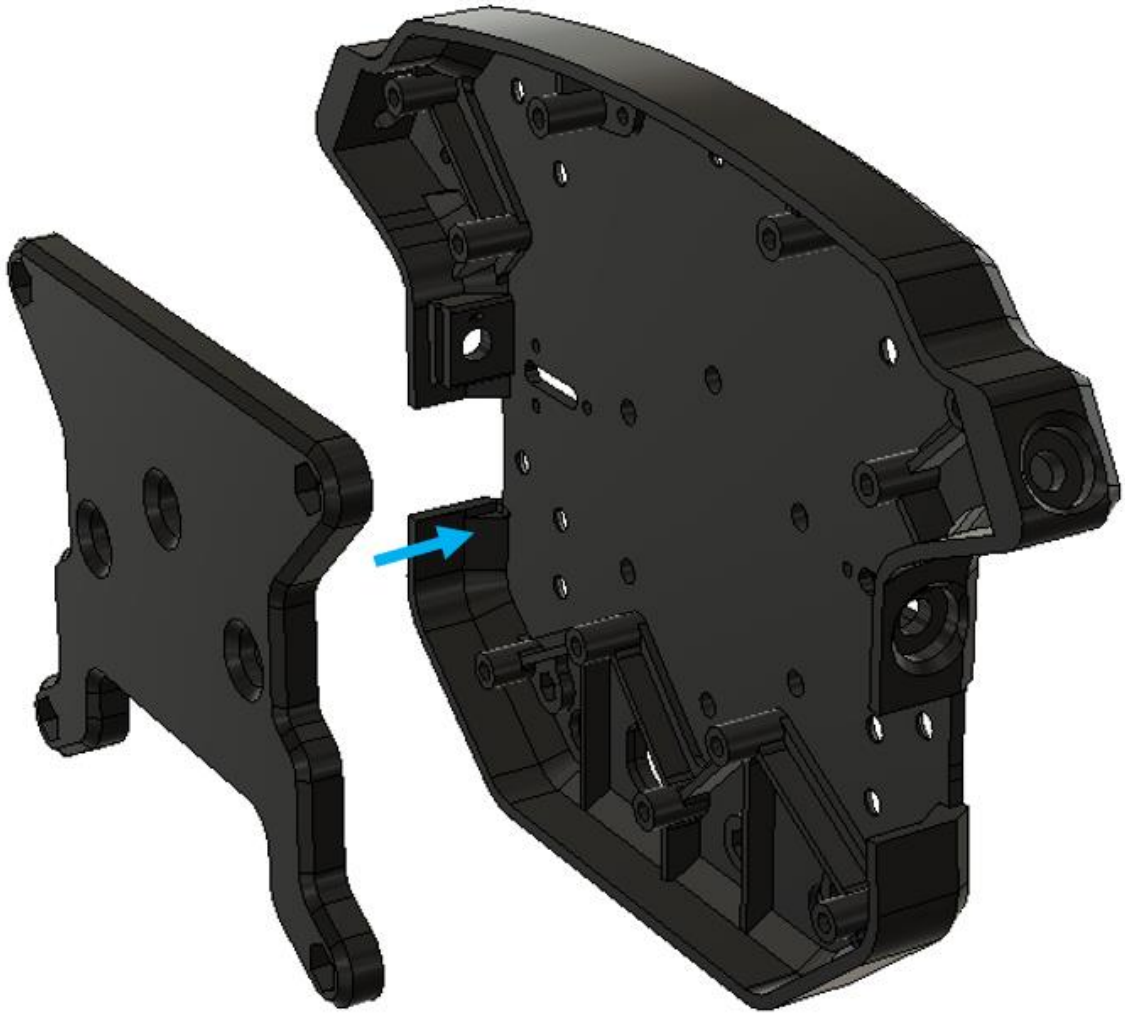
Front plate:



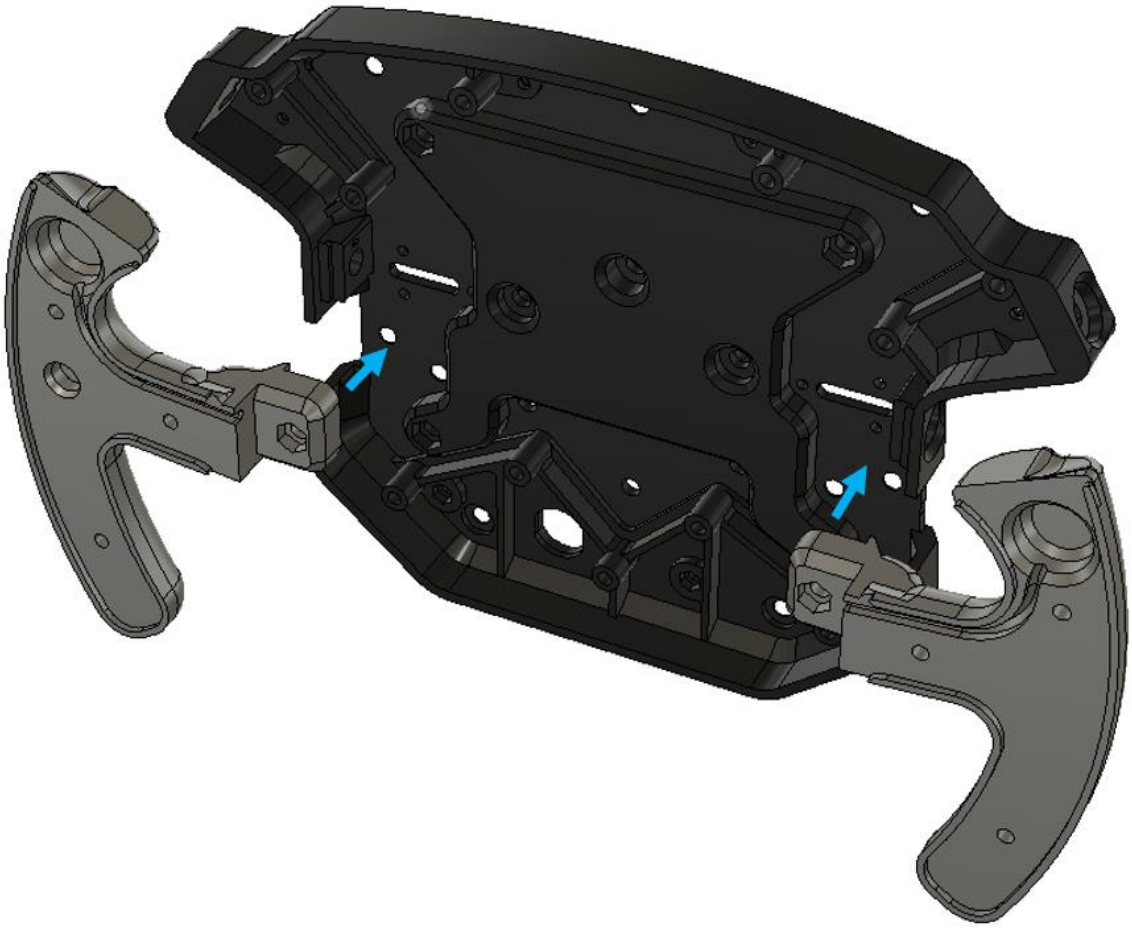
Assembling steps

Fasten the side walls to the backplate with M3x10mm countersunk screws and M3 nuts.

Fasten the backplate strength part to the backplate with M5x10mm countersunk screws and M5 nuts to backplate. (Optional, only if you want to use this brace part.)

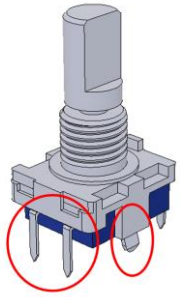


Fasten the rear part of grips with M5x15mm countersunk screws and M5 nuts to the backplate.

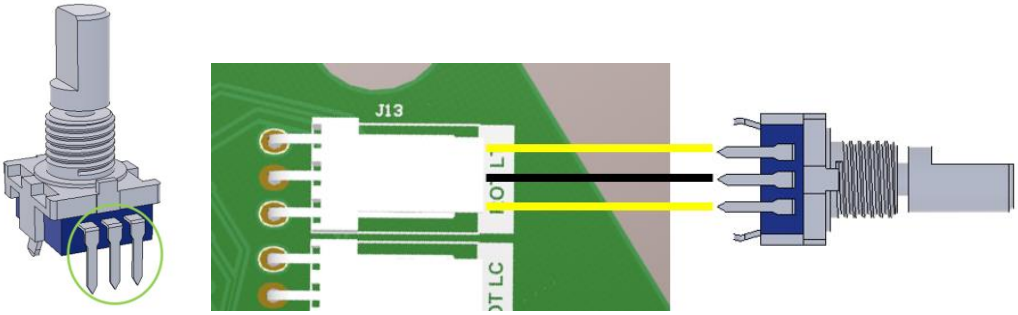


Solder wires on the encoders. 3 pin on one side need to be used. All other pin should be cut, because the space is tight.

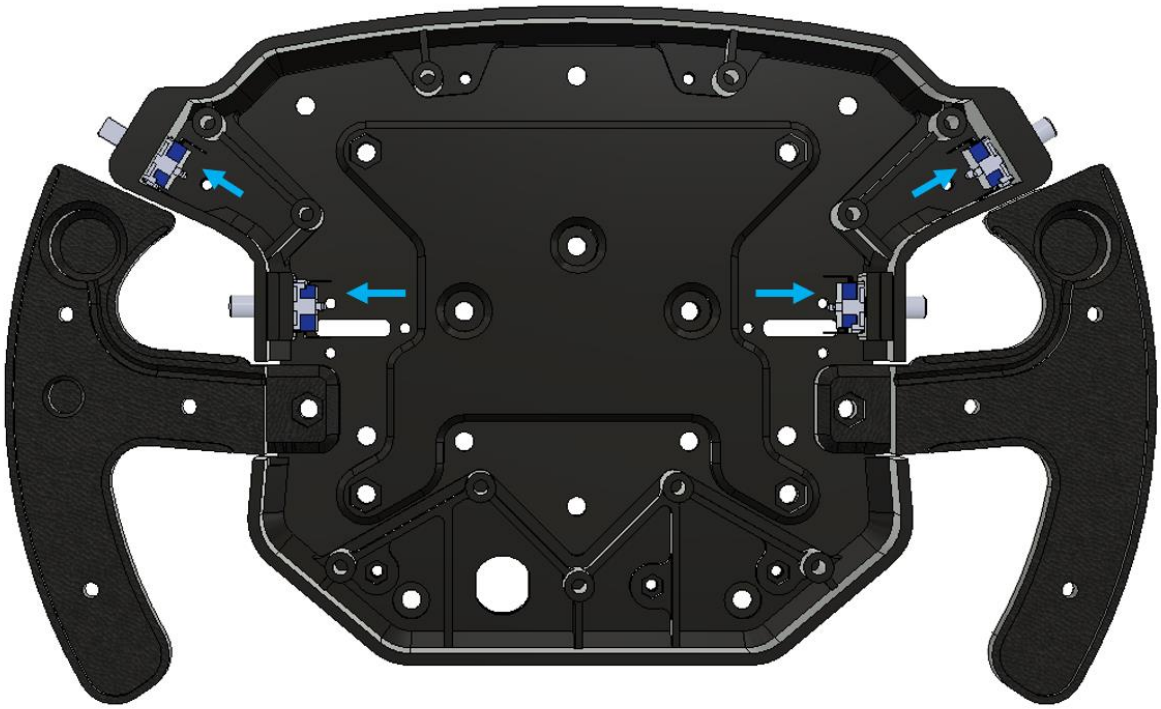
Red marked pins should be cut: side mounting pins and 2 push switch pin:



Solder wires to green marked pins, DON'T cut these!



Put in the side encoders. If the encoder has 7mm long thread, use the nut for it, if the thread length is smaller, the encoder can be glued in.



Solder wires to pre-assembled shifters and mount on the backplate with M3x8mm screws.

Mount the GX16 4pin circular connector to the backplate.

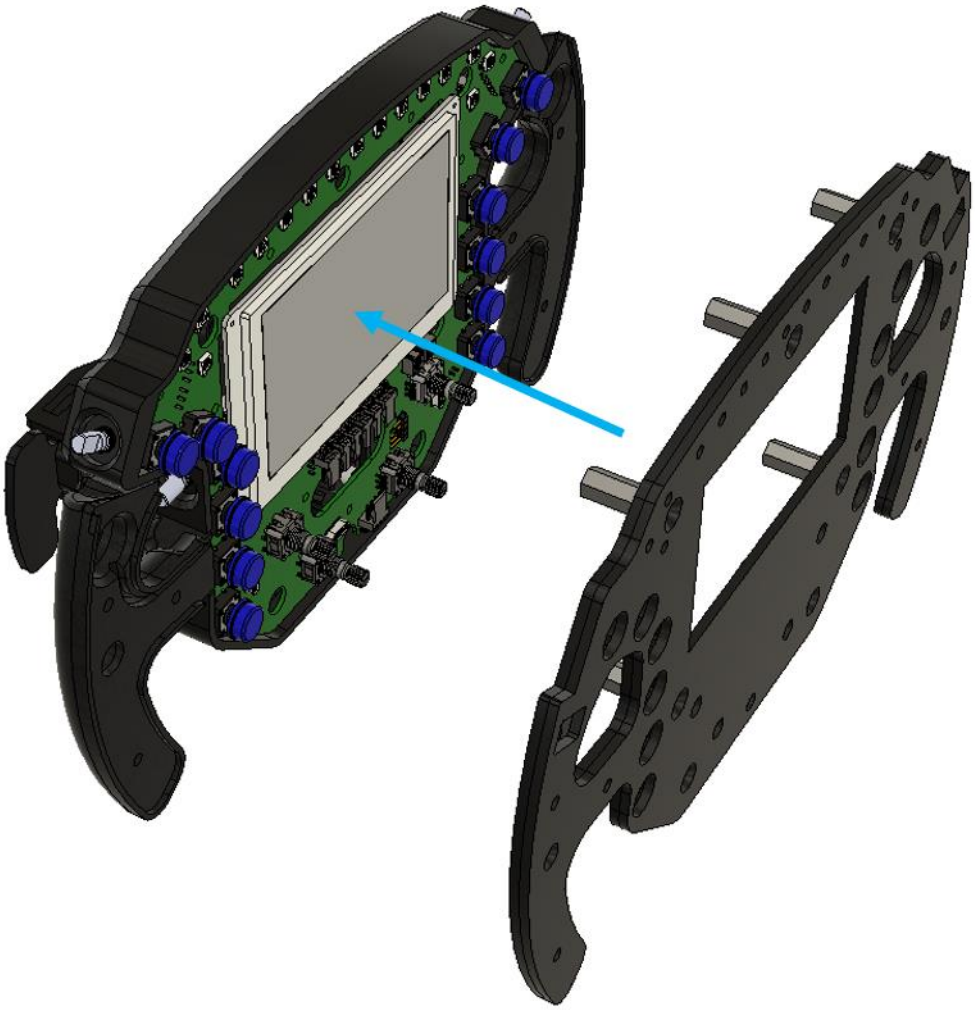
Connect the side rotary encoders, the shifters, grip button cables and the USB canle (circular 4 pin connector) to the PCB.

Put the PLC in place and fasten it with M3x5mm screws.

(PCB may look different according the version you have.)

If you don’t want to disassemble anymore recommended to mount the quick release before you put the PCB in!

Fasten the front plate to the already assembled housing.



Button plate/plastic cover parts and plexi can be glued (2 sided tape is recommended) already.

Don’t forget to fasten the 7 spacers to the front plate with M5x15mm countersunk screws, before you put on the plexi and such.

I also recommend to fasten the 2 front plate (A and B) together with the 7pcs spacers and M5x15mm countersunk screws before powder coating this part.

Place the grip button into the front part of the grips.

Check the direction according the front plate cutout! (red marked) The pins of the buttons should stay according the cutout of the frontplate!

The plastic position pins can be cut from the button.

Solder the buttons to the wires (yellow) and move the front grip in place. Push the left wires into the hole in rear part of the grip. Don’t leave more than 2-3cm wire, because won’t fit in.

If you want to disassemble the wheel, always unsolder the grip buttons first.

This step is unfortunately needed, because the rear part of the grip is also connected to the backplate to make it stronger. So the rear part of the grip is between the PCB and backplate.