

Introduction

Battery Management System (BMS) is any electronic system that manages a rechargeable battery. In case of this particular BMS, there is a self-made battery pack with 21700 cells in 6S4P configuration (max 25,2V) that will be managed.

Few words about why this PCB was created in the first place. It is a part of much bigger project made by PUT Motorsport, which is a racing team of the Poznan University of Technology. We have been designing, manufacturing and testing racing cars continuously since 2014, and then checking them during the international competition where teams from all around the world compete. This year, we focused on developing and improving an electric car. It should be noted that many design decisions are dictated by the regulations that we have to follow when building a racing car.

So presented circuit is one of the most important ones, as it is the low voltage battery that powers all the systems in the car.

Some important functions this PCB will include:

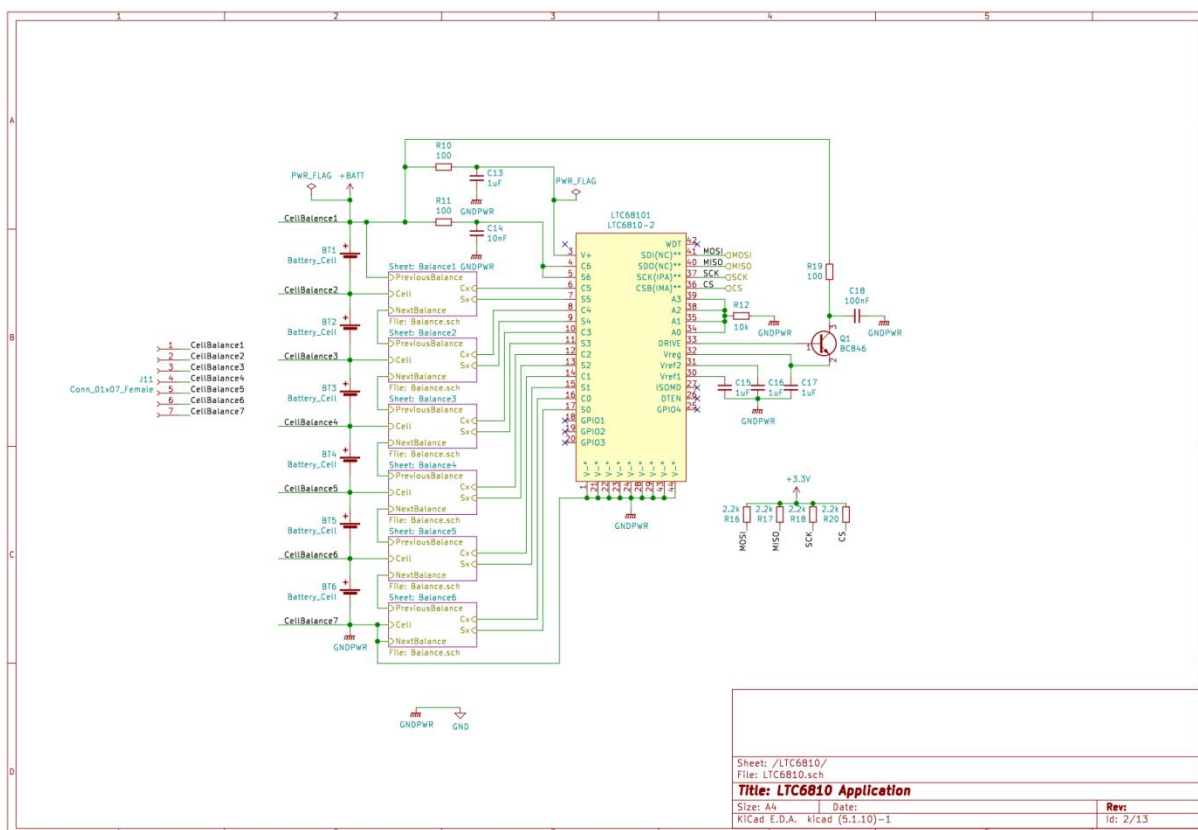
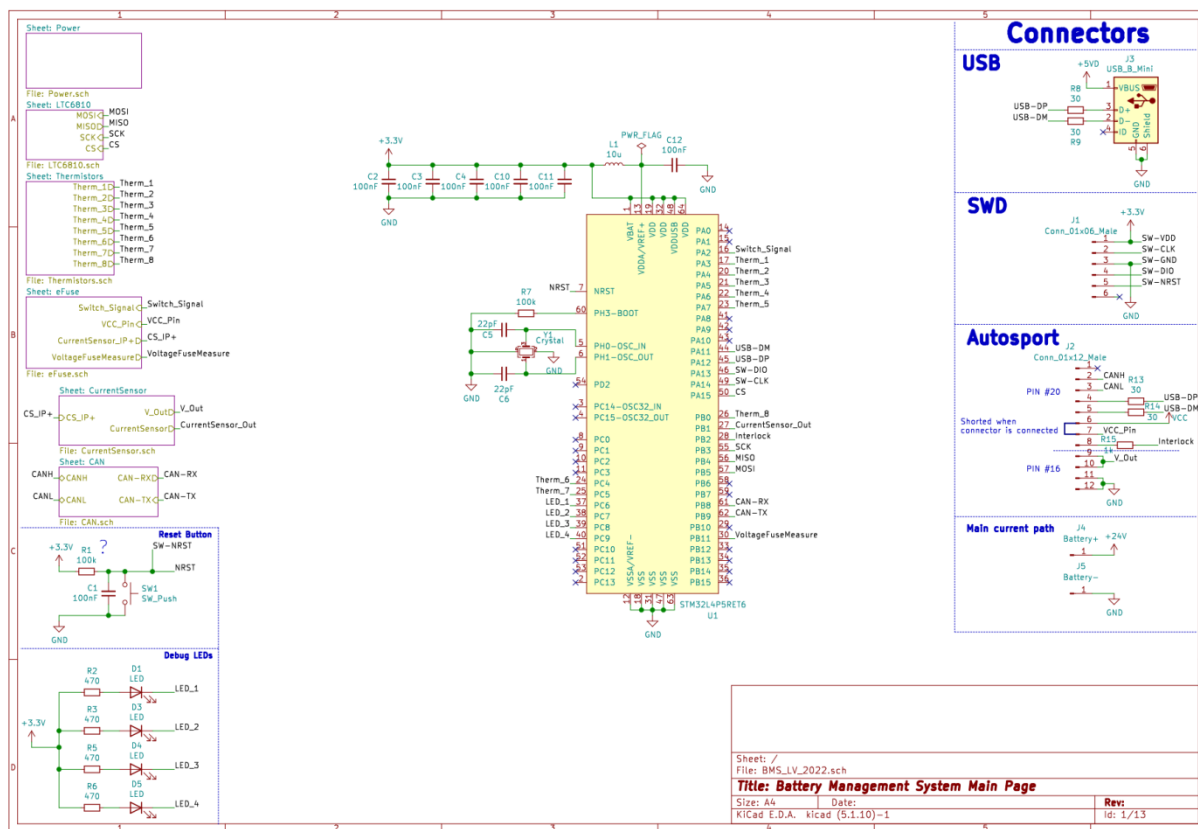
- Voltage monitoring on individual cells (6S – 6 measurement points),
- Temperature monitoring on 33% of all cells,
- Battery current consumption monitoring,
- Battery protection against critical errors specified in the regulations of Formula Student Germany. In the event of an error, the battery power is disconnected from the rest of the car's systems thanks to my eFuse,
- Battery cell balancing system while charging,
- Information processing and sending via serial port (for user) and CAN (for other car systems),
- Voltage measurement point behind fuse placed on the main current path (so there is information about blown fuse).

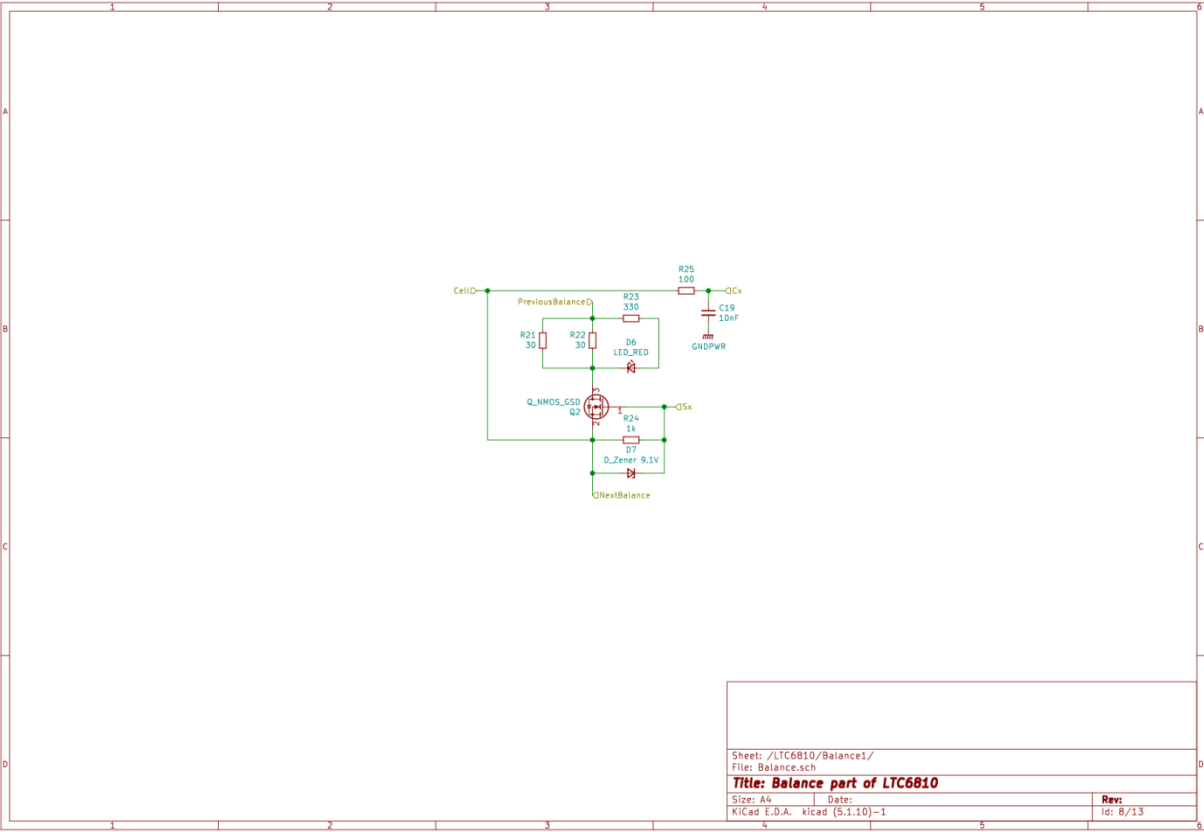
Voltage monitoring and cell balancing is done by LTC6810-2 via SPI.

If some want to see more details about whole project, it can be reached on GitHub:

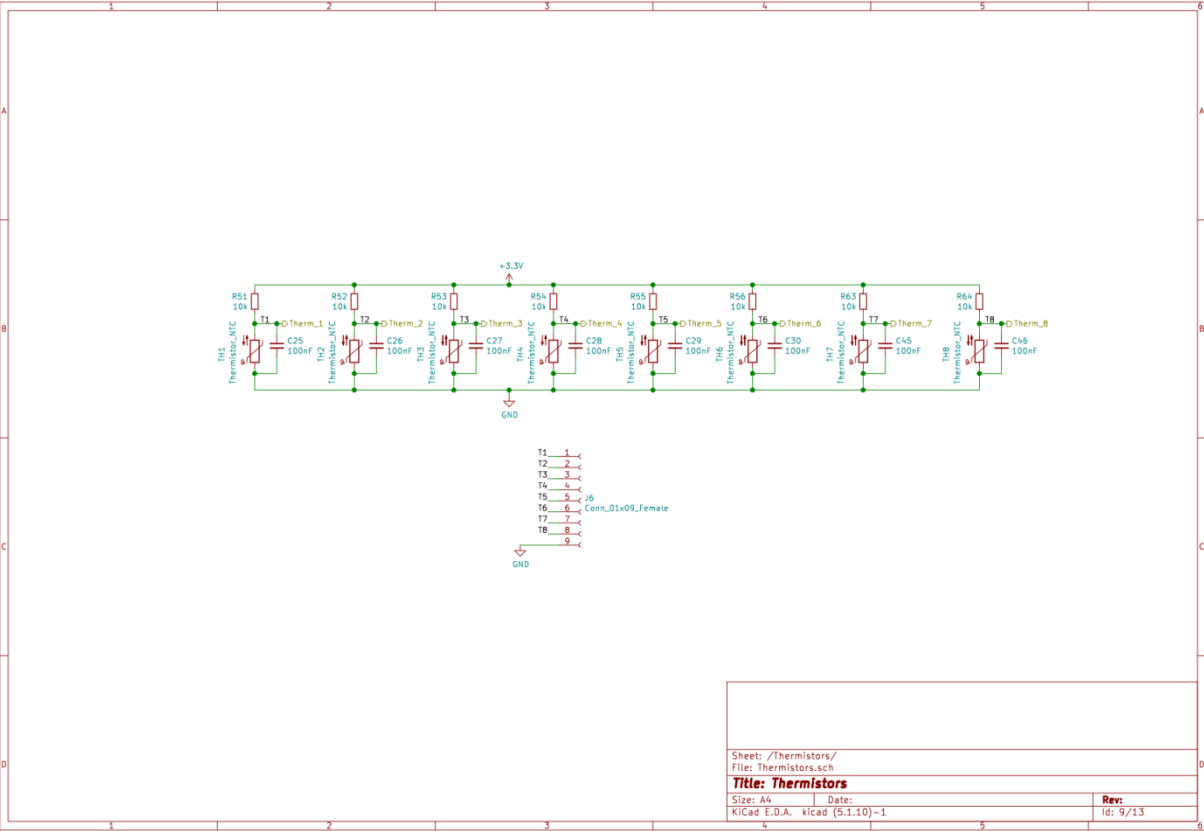
https://github.com/PUT-Motorsport/PUTM_EV_BMS_LV_2022.

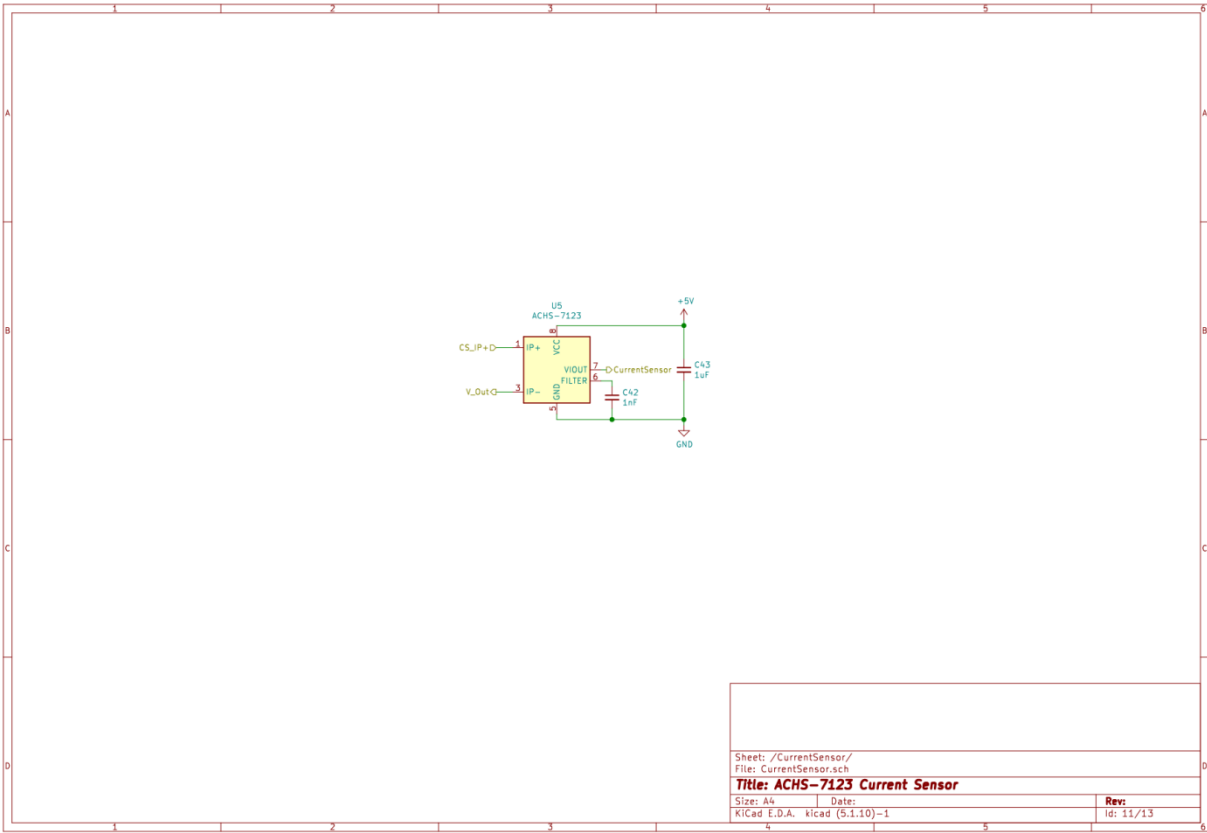
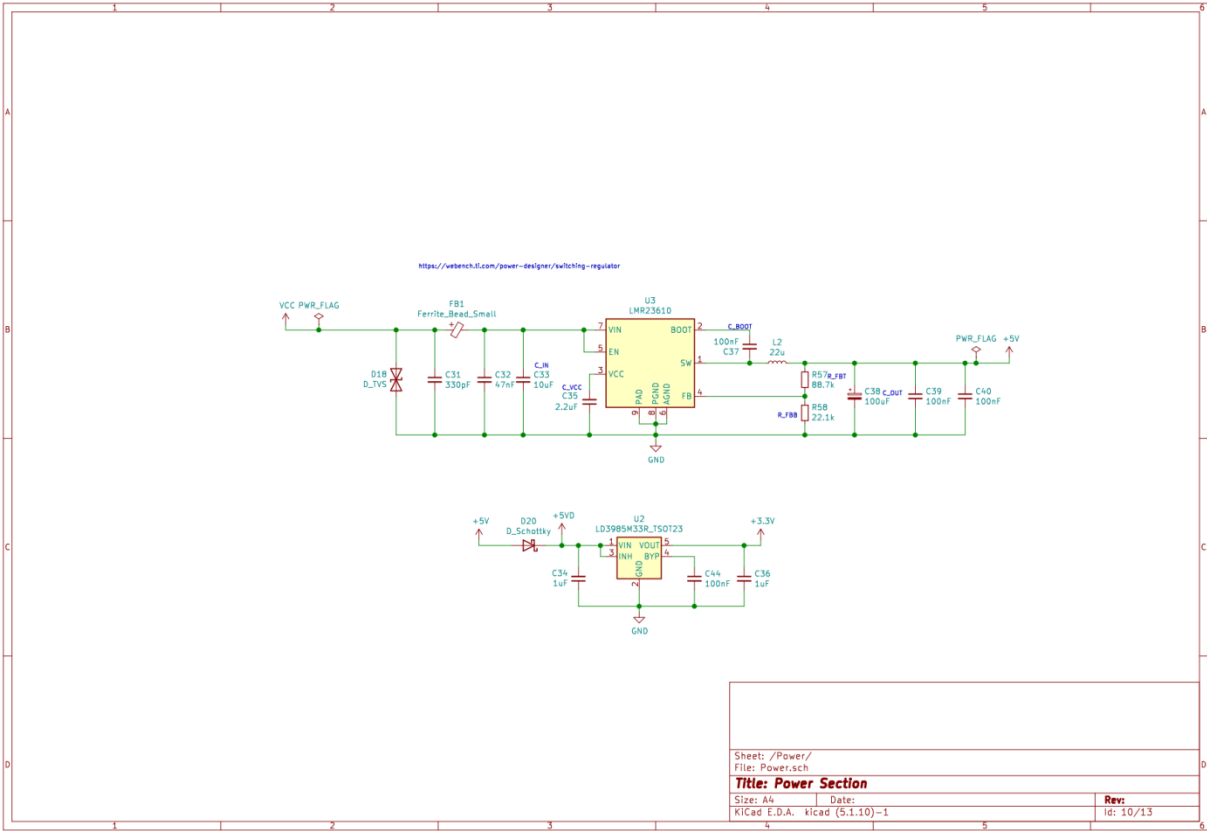
Schematics





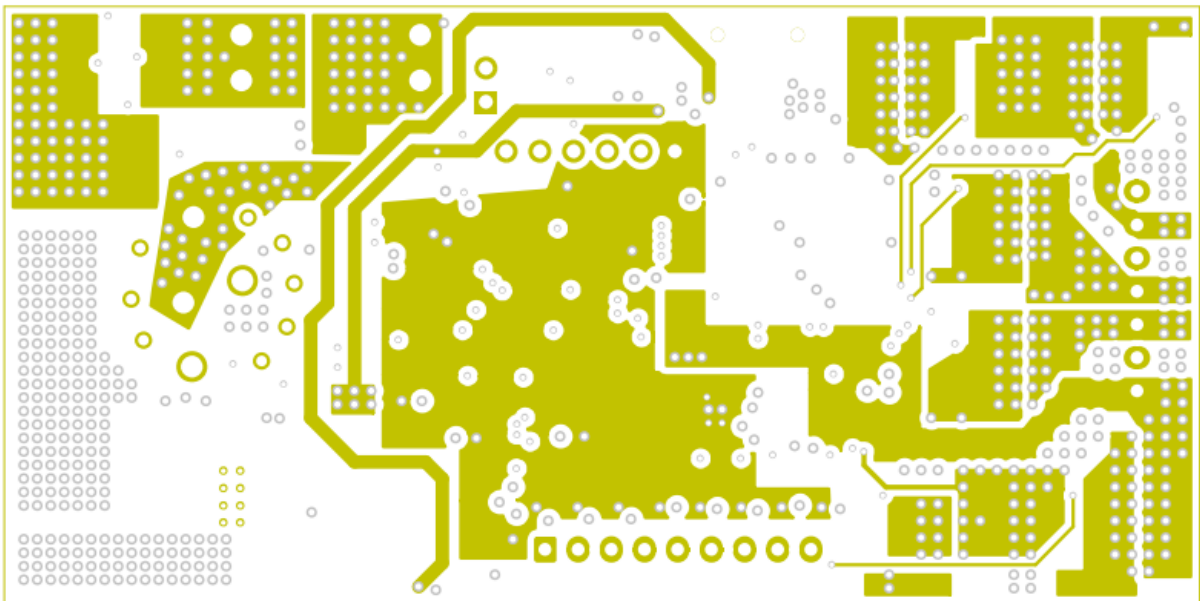
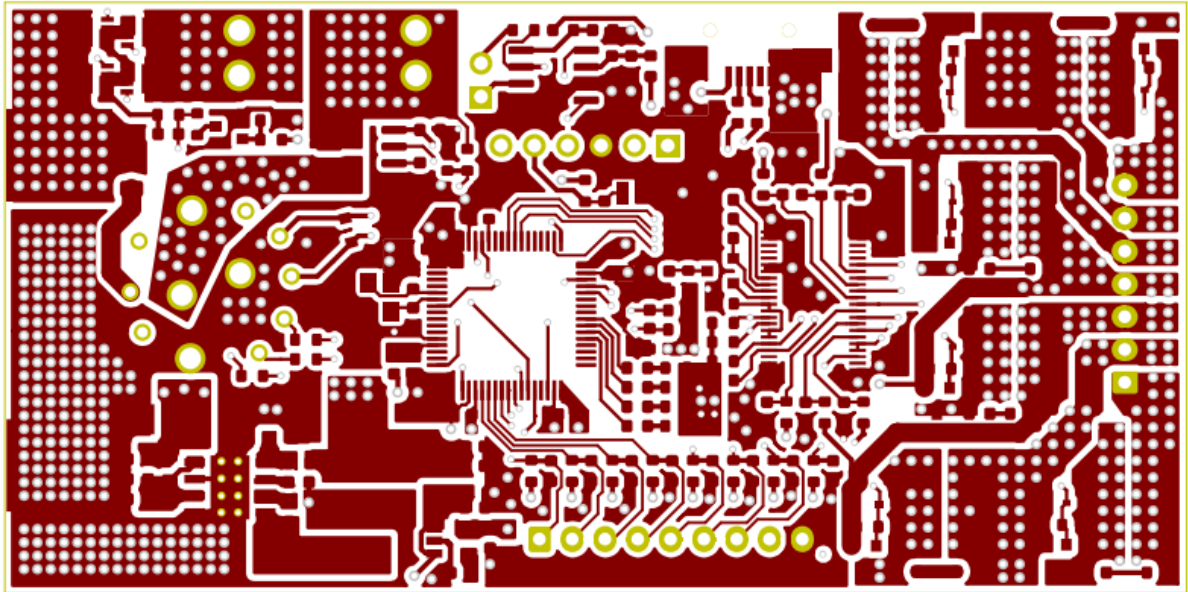
Balance1- Balance6 are the same thing logically.

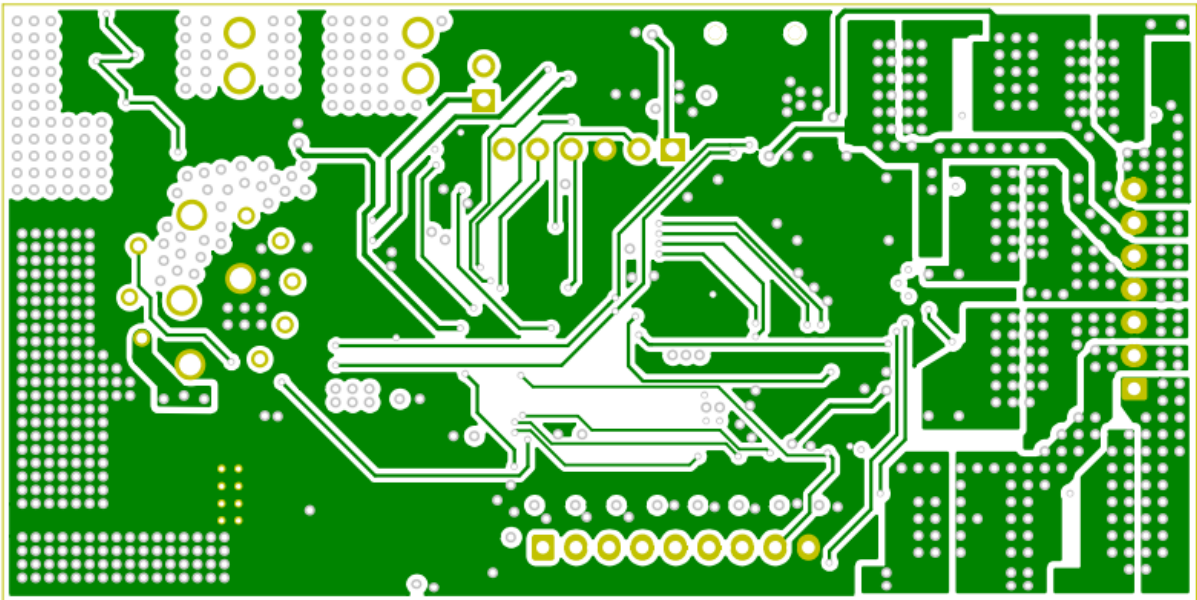
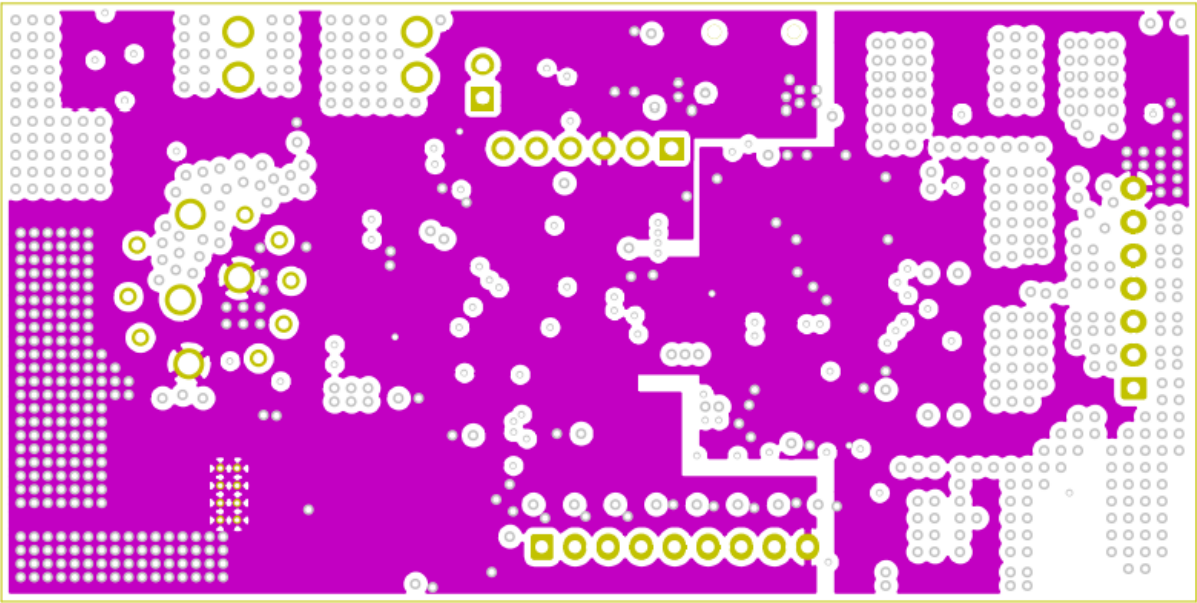




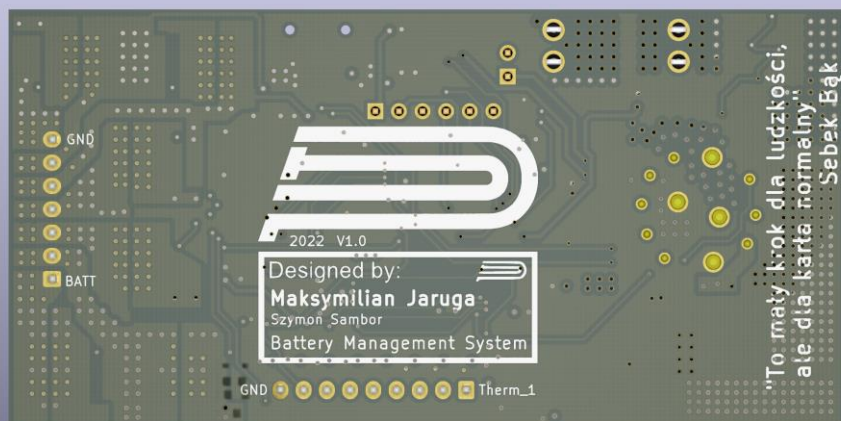
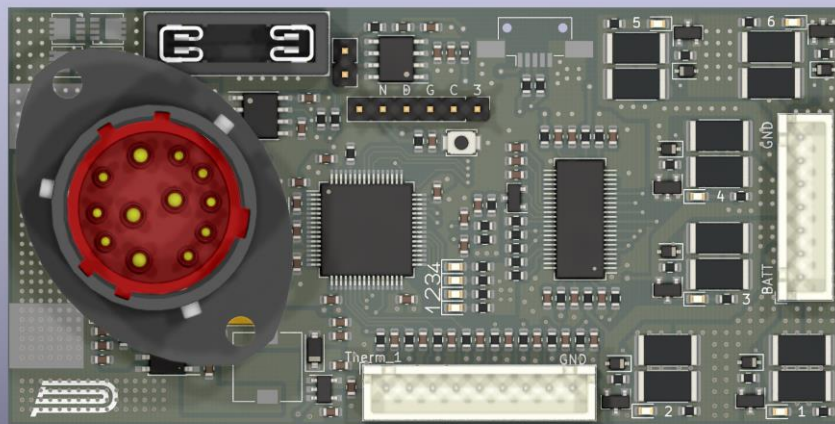
PCB Layers

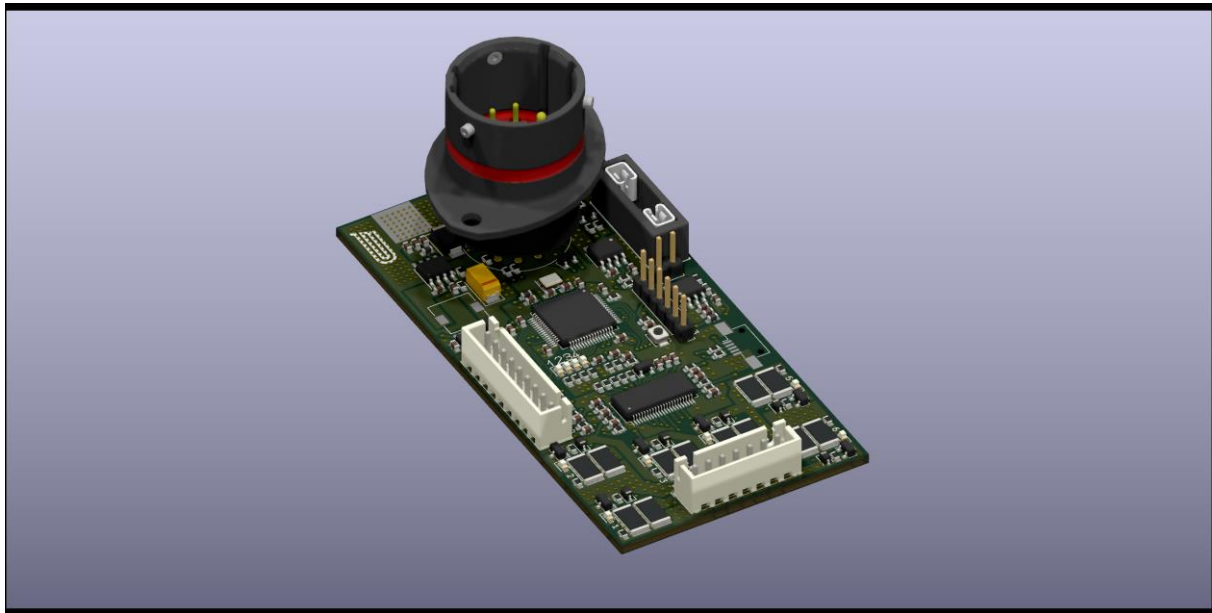
Created PCB has 4 layers. The following layers are presented from the first to the last.





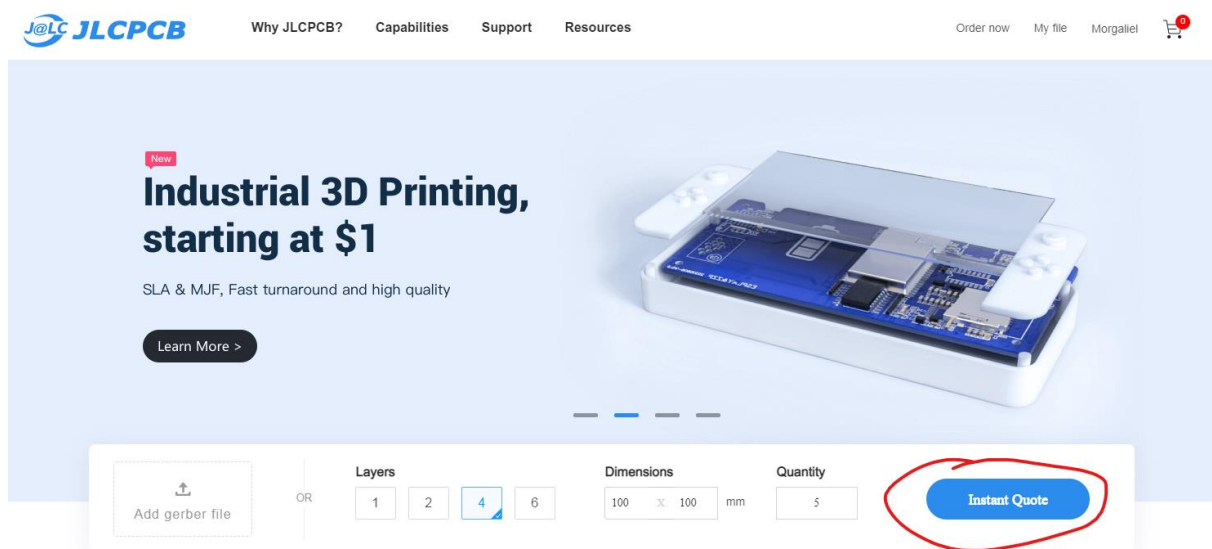
3D Model





Ordering PCB on JLCPCB

Now that we've got the PCB design, it's time to order. For that, you just have to go to jlcpcb.com, and click on the "Instant Quote" button.

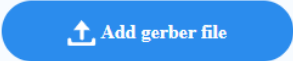


You don't have to worry about the settings here, because you can adjust everything in the new appeared tab.


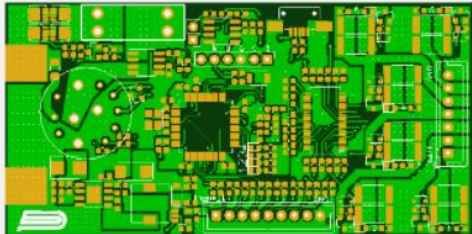
JLCPCB is also a sponsor of this whole project (not only this PCB, but all the PCBs in the car!). JLCPCB (Shenzhen JLC Electronics Co., Ltd.), is the largest PCB prototype enterprise in China and a high-tech manufacturer specializing in quick PCB prototype and small-batch PCB production. You can order a minimum of 5 PCBs for just \$2 (2 layers), just enough to have a few test/spare ones.

To get the PCB manufactured, upload zipped Gerber files as you can see below. How to get them in KiCad 5? JLCPCB made the whole tutorial right here:

<https://support.ilcpcb.com/article/149-how-to-generate-gerber-and-drill-files-in-kicad>.

PCB	SMT-Stencil	3D-Printing
<div><div>Add gerber file</div><div>Only accept zip or rar, Max 10 M, View example ></div><div>Instructions for ordering Upload History >></div></div>		

After uploading a zip file, you'll see the top and bottom of your PCB as example shows.

PCB	SMT-Stencil	3D-Printing
<p>Detected 4 layer board of 45x90mm(1.77x3.54 inches) .</p> <p>Your upload has finished processing. Enter the project details below and we'll move on to checking all the individual layers to make sure that they're correct.</p> <div></div>		

Then, you have multiple options to adjust from number of layers to your PCB colour. Every option here is very clearly marked and described (when you hover over the question mark).

Base Material	<input type="radio"/> FR-4	<input type="radio"/> Aluminum					
Layers	<input type="radio"/> 1	<input type="radio"/> 2	<input checked="" type="radio"/> 4	<input type="radio"/> 6			
Dimensions	<input type="text" value="45"/> * <input type="text" value="90"/> <input type="text" value="mm"/>						
PCB Qty	<input type="text" value="5"/>						
Different Design	<input checked="" type="radio"/> 1	<input type="radio"/> 2	<input type="radio"/> 3	<input type="radio"/> 4	<input type="text"/>		
Delivery Format	<input checked="" type="radio"/> Single PCB	<input type="radio"/> Panel by Customer	<input type="radio"/> Panel by JLCPCB				
PCB Thickness	<input type="radio"/> 0.4	<input type="radio"/> 0.6	<input type="radio"/> 0.8	<input type="radio"/> 1.0	<input type="radio"/> 1.2	<input checked="" type="radio"/> 1.6	<input type="radio"/> 2.0
Impedance	<input checked="" type="radio"/> No	<input type="radio"/> Yes	Impedance calculator >				
Layer stackup	<input checked="" type="radio"/> Fill in your layer sequence <input type="checkbox"/>						
	L1(Top layer)	<input type="text" value="BMS_LV_2022-F_Cu.gbr"/>					
	L2(Inner layer1)	<input type="text" value="BMS_LV_2022-In1_Cu.gbr"/>					
	L3(Inner layer2)	<input type="text" value="BMS_LV_2022-In2_Cu.gbr"/>					
	L4(Bottom layer)	<input type="text" value="BMS_LV_2022-B_Cu.gbr"/>					
PCB Color	<input checked="" type="radio"/> Green	<input type="radio"/> Purple	<input type="radio"/> Red	<input type="radio"/> Yellow	<input type="radio"/> Blue	<input type="radio"/> White	<input type="radio"/> Black
Silkscreen	<input checked="" type="radio"/> White						
Surface Finish	<input checked="" type="radio"/> HASL(with lead)	<input type="radio"/> LeadFree HASL-RoHS	<input type="radio"/> ENIG-RoHS				
Outer Copper Weight	<input checked="" type="radio"/> 1 oz	<input type="radio"/> 2 oz					
Inner Copper Weight	<input checked="" type="radio"/> 0.5 oz	<input type="radio"/> 2 oz					
Gold Fingers	<input checked="" type="radio"/> No	<input type="radio"/> Yes					
Material Type	<input checked="" type="radio"/> FR4-Standard Tg 130-140C	<input type="radio"/> FR-4 TG155					
Confirm Production file	<input checked="" type="radio"/> No	<input type="radio"/> Yes					
Flying Probe Test	<input checked="" type="radio"/> Fully Test						
Castellated Holes	<input checked="" type="radio"/> No	<input type="radio"/> Yes					
Remove Order Number	<input checked="" type="radio"/> No	<input type="radio"/> Yes	<input type="text" value="Specify a location"/>				

Advanced Options
☐

After making sure your PCB looks good – by clicking *Gerber viewer*, you can now place the order at a reasonable price. In case of this project, there is 8\$ for 5, 4-layers PCB and it's only 6-7 days build

time(the time is extended due to the chosen colour). To place the order, click on the “SAVE TO CART” button. Fast and cheap, right?

Charge Details ^	
Special Offer	\$8.00
Build Time ?	
PCB: ● 6 - 7 days	\$0.00
Calculated Price \$8.00	
Additional charges may apply for special cases	
Weight ?	0.07kg
SAVE TO CART	

The quality is always really satisfying so I sincerely recommend JLCPCB.