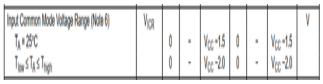
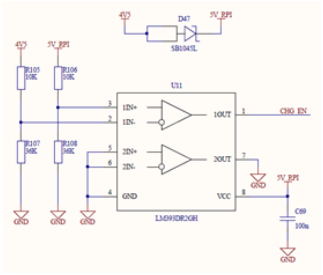
Modification discussion

#	Modification	Comment	Reply
1	Pull-up resistor on charger enable pin: We'd like to add a pull-up resistor on CHG_EN line between U11 and U5, e.g. 47k to the 5V_RPI signal.	Ok, We will add a PU resistor. I think, 5.1kOhm. Because the output current (Rpullup) of the comparator is 1uA to 4mA => 5.1kOhm is great for us. (5V / 5.1kOhm = 0.98mA). See DataSheet (http://www.ti.com/product/LM393?qgpn=lm393) p.12.	We don't have any 5.1kOhm in our BOM, I don't want to introduce a new value. I recommend an already existing 0402 resistor, 10K, 39K or 47K. The charger's pull up current is 1uA so we don' have to design it with a specif value.
		We will use 10kOhm.	
2	Voltage divider and filter on comparator inputs: According to the datasheet, we might be out of specification with the voltage divider what we use on the comparator's input:   We suggest dividing the voltage by 2 with e.g. 36k resistors instead of 10k. What do you think?	I think it isn't good idea. We tested the comparator with a lot of other variants the voltage divider. And there is a great variant the voltage divider on the schematic.	Ok, if you don't think that we are out of specification we can use these resistors.
		We tested the comparator with the voltage divider (10kOhm + 10kOhm). And it worked great. We will change the voltage divider.	
3	Moreover, we would like to have a low pass filter on the inputs with a low (<<50Hz) cutting frequency, maybe 1uF capacitors would be good in parallel with R107 and R108.	Do you have a problem with a noise (a low frequency) on the inputs of comparator? Rc (50Hz, 1uF) = 1 / (2*Pi*f*C) = 3.185 kOhm.	We can see that 50Hz noise can have effect on the inputs, and it's not a problem if we have a LPF on the inputs, we can have much lower cutting frequency than 50Hz.
		We will add 1uF + 100nF to the inputs of the comparator.	
4	Connect CHG_EN to MCU We'd like to be able to read the CHG_EN signal from the MCU. We don't have any free pin, because every remaining pin was connected to Raspberry Pi, which is good. I'd like to remove the DNP micro USB socket completely, and then you can connect the CHG_EN to the USB DM or DP. According to my discussion with Microchip we don't have a pre-installed SAM-BA bootloader in our MCU, so we can't use the USB interface anyway, it's not a big deal to sacrifice it.	No problem. Do we need to remove the microUSB connector (CON5) and D46?	You can completely remove the microUSB from the panel, D46 is already DNP, you can delete it.

5	Please apply thermal relieves We'd like to change all the ground and VCC pads of the Raspberry pi connector to hot pads in every layer. YuDaFu has some soldering issues which can be solved with thermal relieves. Please apply thermal relieves on the Battery pads also.	What exactly is "hot pads"? Now all RPi connector TH pads have thermal relieves GND planes, probably they want to reduce amount of connected copper to ease soldering, don't they?	Please add thermal relieves to Vcc and battery pads too. Yes, they want to make the soldering easier.
6	The footprint of U14 is not correct It's not your mistake, you just followed the datasheet, but you know it better than me, that Chinese ICs are sometimes slightly different than their datasheets. It seems that the thermal pad is bigger on the real ICs, so a very small angle difference can cause a short between the GND and any pin. Almost all the pre-production issues were due to this IC. I'll send you measurements of the real IC tomorrow.	OK, clear. According to real measurments real EP size is 2.7x2.7mm	I sent my measurements on Skype
7	Increase charging current Please increase the charging current to 800mA.	Ok. We will change Riset (R95) to 1.2kOhm (806.54mA).	OK
8	White silk-screen ring around LEDs Please add a big white silkscreen ring around the LEDs, so they'll be less noticeable through the semi-transparent ring.	OK, clear	
9	Jumper pads on the outputs of supplies Please add jumper pads (without solder mask) that can be cut when needed on the outputs of supplies, like this:  (https://docs.zephyrproject.org/latest/_images/nrf52_pca10040.jpg) If it's needed it can be cut and can be re-soldered later.	What signals will we use with a jumper pads?	the interesting points are: - 5V that goes to the LED panel (because I don't want to cut/solder the wire) - 3V3 that goes into the MCU - 3V3 that goes into the IMU - 5V_RPI that supplies the comparator (U11) and charger (U5, 800mA!) - 5V that supplies the speaker amp (U16) - VCCB lines that supply the level shifters (x4)

10	Additional test points (existing TPs must remain their current position!) a) Please add test points for the LED panel connector on the bottom of the mainboard. They will test the mainboard separately from the LED board and it would much easier to use bottom side test points instead of plugging in LED boards to the connector. A LED board must be attached because we show the self-test results on the LEDs. b) Please add test points to the bottom side of the LED board. TPs for the connector contacts and the speaker contacts. Please put the TPs outside of the battery area. c) Please add test points for the comparator inputs	Ok	
11	Please try to find out if we can have inter-board connector pins between the LED board and mainboard. I don't want to do big changes, but if it's feasible it would make the production much easier.	we will check	Chinese team suggested to keep the current cable for mass production
12	We found that we can't do any modifications in the Altium project because we don't have the libraries for the project. Could you share the final part library with us?	OK, we'll send library along with project files after all updates	
13	please change FB3 to a net tie. we have some issues with voltage spikes when the motors are running and shorting the ferrite bead greatly reduces this issue	Ok	
14	Please modify the LED board's connector pitch to 2mm		