Wk6 Peer Review of PCB

Review of Group 32

JNSRYA006 TRFDEV001 CRGALE002

Peer Review of PCB Discovery HAT

For each of the following, rank each aspect on a scale of 0-5 where 0 is unacceptable and 5 is excellent. If you rank something as less than 5, indicate in the supplied comment section why you ranked this lower and what you would suggest changing.

1. **Adherence to Mr Pead's <u>basic</u> requirements.** Have the requirements of the project been met according to your understanding based not only on the initial project description but also based on all further lab presentations and design discussions. Consider **ALL** the implications of each of these in making your assessment.
   <u>Reminder of basic requirements:</u>

- (at least) 1x digital sensor (some communication port)✔
- (at least) 1x analog sensor (ADC input) ✔
- An 18650 connector (Do Not Populate - Link provided on Padlet) ✔
- A Li-Ion battery charger ✔
- Input voltage polarity protection ✔
- Battery polarity protection✔
- Battery Under-voltage cutout protection ✔
- (Soft requirement) Modularity between perceived elements to allow for bypassing to ensure you meet partial mark requirements in the event of module failure. ✔
- USB micro connector, detect and input and have an onboard FTDI - USB connector LCSC number provided on Padlet ✔
- Working in teams of 3 each team will have a budget of R1500 for all components, manufacture, assembly and shipping. Currently, this is approximately $100 (useful to think in USD as the part prices are listed in such) ✔
- Each team member must take the lead in 1 submodule:
  - Power management
  - Microcontroller interfacing
  - Sensing

Unacceptable 1	Ok attempt 2	Reasonable 3	Good 4	Excellent 5
				✔

Comment:

We like the 3V regulation as it looks like it includes the UVLO, rather than needing an additional circuit. Furthermore, the choice of a P-MOSFET to protect battery polarity is clever as it is included in the supply switching circuit, again, rather than needing an additional circuit.

---

2. **Adherence to Mr Pead's debugger requirements.** Have the requirements of the debugger been met according to your understanding based not only on the initial project description but also based on all further lab presentations and design discussions. Consider **ALL** the implications of each of these in making your assessment.

   Reminder of high level debugger requirements:
   1. Input 5V - This is to provide high level power to your board.
   2. Analog Data - This is to both see what your sensor outputs and to push a signal into your board. If we are to push an analog voltage you need to ensure we don't destroy your sensor (So maybe a jumper to disconnect) ✔
   3. Digital Data - Same story - we want to see it working / control it from off the board - where possible have the means to facilitate your systems not dying. ✔
   4. Depending on what digital communication port you chose, I allowed for 4 pins.
   5. Plug Detect is your USB detect line - which we can control and monitor to change your devices operation **X**
   6. The data the device is sending out. (will need to disconnect the FTDI - Think Jumpers) ✔
   7. GND ✔

Unacceptable 1	Ok attempt 2	Reasonable 3	Good 4	Excellent 5
			✔	

Comment:

There doesn't seem to be an output that shows a USB has been detected. Some of the connections of the Microcontroller Interface module aren't connected, so the traces on the PCB will not be connected because of this.

---

3. **Schematic** Is the schematic neatly laid out, well labelled, are components labelled, is it clear what function each subsection serves.

Unacceptable 1	Ok attempt 2	Reasonable 3	Good 4	Excellent 5
		✔		

Comment:

The light sensor needs an additional resistor below V_{out} because the datasheet says there is a constant V_{ce} at ambient temperature. This means V_{out} will read 5V (as per the V_{ce} on the datasheet) regardless of the amount of light incident on the sensor.

The Power Module includes a good idea of using the idea of a supply switch and, also using this switch to protect the circuit for polarity protection. The Power Module had a good choice of ICs as they seem to perform multiple functions and, according to the BOM, are also affordable choices.

---

4. **PCB Layout.** Is the PCB neatly laid out, does it appear care has been taken regarding trace location, routing, and thickness?

OBJ OBJ

Unacceptable 1	Ok attempt 2	Reasonable 3	Good 4	Excellent 5
	✔			

Comment:

There could be less vias, as some of them don't seem necessary. Furthermore, some of the traces seem a bit long, and components could be placed a little closer together to reduce the length of traces and to make the layout a bit simpler. The trace width seems to all be the same size and not a bit larger to accommodate the 5V lines.

---

5. **Silk screen.** Is there a silkscreen, have key components been labelled that will make a user and engineer's life easier when they need to either service or use the board? Is there a version number and design name? Has space been well used?

Unacceptable 1	Ok attempt 2	Reasonable 3	Good 4	Excellent 5
		✔		

Comment:

The Silkscreen doesn't contain a version number and Design Name, The only values on the silkscreen are the component numbers and their outlines. Some of these are covered by the components, and considering the space around the components, these values could be more visible.

OBJ OBJ

6. **Low voltage protection circuit (review on schematic).** Based on the schematic, do you expect the circuit to function as expected?

Unacceptable 1	Ok attempt 2	Reasonable 3	Good 4	Excellent 5
				✔

Comment:

The chosen IC for the 3V regulation appears to solve the problem of Low Voltage Protection, rather than needing an additional circuit. It could be cool to include a LED to show a status, similar to the circuit shown in the datasheet.

OBJ OBJ

7. **Physical board design.** Given the use case, is the size, shape, and layout of the PCB well thought out?

Unacceptable 1	Ok attempt 2	Reasonable 3	Good 4	Excellent 5
	✔			

Comment:

The placement of the pins to the Discovery board, will mean that all the components on the PCB will be inaccessible whilst connected to the STMDiscovery. The choice to have the battery holder on the other side of the board is clever, as this component won't fit in between the board and the STMDiscovery. The EdgeCuts layer on the board is missing/deleted, as when rendering the PCB, there is no defined size, so the whole schematic is filled. JLCPCB only accepts up to 100X100mm boards for cheap production, based on the project

specifications. This means that there needs to be a defined EdgeCuts layer defined. The contents of the PCB also seem a bit spread out, and the placement of components could be closer together.

Furthermore, there seems to still be an additional pair of pins on the PCB which are unnecessary.

---

8. **Test points and recovery approaches.** Are test points for each submodule available, do you perceive them to be accessible and sufficient, is there any portion of the circuit that you think additional (or less) test points would be valuable? Are the mechanisms in place to handle failures of key components? (Eg jumpers and redundant routing). Consult the submitting team's git repo where necessary to understand the design.

Unacceptable 1	Ok attempt 2	Reasonable 3	Good 4	Excellent 5
	✔			

Comment:

There are no test points for the data lines. There are not any jumpers which might be used to disconnect faulty modules. THis would be very useful in the power module. Jumpers would be useful on the sensing lines for the sensing submodule in order to test sensors before plugging them into the STM. Jumpers on the digital data lines would also be useful in order to test code in the event of one of the microcontroller interfacing submodules failing. They are quite easy to add and can be sourced cheaply.
