Hardware development Plan
Mälardalen University Solar Team (MUST)

Henrik Särnblad*, Mathias Strand†
School of Innovation, Design and Engineering, M.Sc.Eng Robotics
Mälardalens University, Västerås, Sweden
Email: (*hsd16003, †msd16007)@student.mdh.se

CONTENTS

I Introduction 2

II PCB design standards 2
II-A Block diagrams . . . 2
II-B Design . . . 2
II-B1 Schematics . . . 2
II-B2 Layout . . . 2
II-C Version Control . . . 3
II-C1 Create a new PCB . . . 3
II-C2 Update existing . . . 3
II-D Testing standard . . . 3

III 3D printing design standard 3
III-A Design . . . 3
III-B Testing standard . . . 3

IV Component selection 3
IV-A Hardware to avoid . . . 4
IV-B Encouraged hardware . . . 4
IV-C External power considerations . . . 4

V Encapsulation 4

VI Documentation 4

VII OneDrive 4

I. Introduction

This document contains design guidelines for hardware development in Mälardalen University Solar Team (MUST). These guidelines aims to make the development of hardware easier but also more reliable. The document will cover the design rules for circuit boards, encapsulation and hardware documentation.

II. PCB design standards

The following sections will cover the design standards for PCBs. These guidelines are put in place to ensure quality and reduce the risk of damage or injury. The following sections will cover the design standards for PCBs. These guidelines are put in place to ensure quality and reduce the risk of damage or injury.

A. Block diagrams

A block diagram should be created before designing a PCB. The block diagram should be featuring the different abstractions layer of the car. The block diagrams will be designed in draw.io and stored on *Team-Embedded - Solar Team*'s github.

- Make use of arc line jumps
- Keep text below 20 pts(except headlines) and boxes reasonably small.
- Indicate high voltage with yellow, low voltage with red and other signals with blue. Unspecified connections can be black.
- don't work on an item at the same time as someone else.

B. Design

PCB design will be preformed primarily in KiCad and use version control (see section II-C). The design layouts are the following:

1) Schematics:

- The schematic should be structured in a logical way and have a multisheet structure for easy re-usability.
- All special designed components and footprints in KiCad should be project specific.
- The files should be saved using version control described later in the document.

2) Layout:

- Board stackup for two layer PCBs: Power, GND.
- Board stackup for four layer PCBs: Signal, GND, Power, Signal.
- Traces should be angeled at 45° or less.
- Traces should not enter a pad at an angle.
- Traces going in to SMD pads should be pairwise co-linear when possible to counteract rotation when reflow soldering. See fig. 1.
- High speed digital trace lengths(>500kbit/s) should not match $12.5 \pm 0.5cm$.
- High speed digital signal traces should not be routed under any components.
- CAN and USB are examples where differential pairs must be implemented.
- Digital and analog components should be placed as far apart as possible.
- SMD components should be placed on one side of the board for easier reflow soldering.
- Connectors should be kept on one side of the card (Some exceptions can be made for low current).
- Power planes and ground planes should be used when possible.
- PCBs should be made as small as possible without jeopardising electrical safety.
- All PCB´s must have M3 mounting holes.
- For larger boards there should be at least 5 screw holes to counter the bending force from the connectors.
- Soldermask should include name of components, name of switches and connectors, name of board and revision number starting with 0.0.

The following should be double checked before the PCB is sent for manufacturing:

- Does the PCB have text on it with name of board, revision number, connection names, hardware config header names(if present), open hardware logo?
- Does all the footprints match their components?
- Are the size of the solder pads big enough to be soldered?
- Are the hole sizes correct?
- Are all the components one the top side? (View in 3D)
- Are any of the components flipped?
- Is the isolation distance on the traces correct?
- Did the PCB pass the Design Rules Check(DRC)?
- Does the board settings match the manufacturing capabilities of wegstr or specified board house?

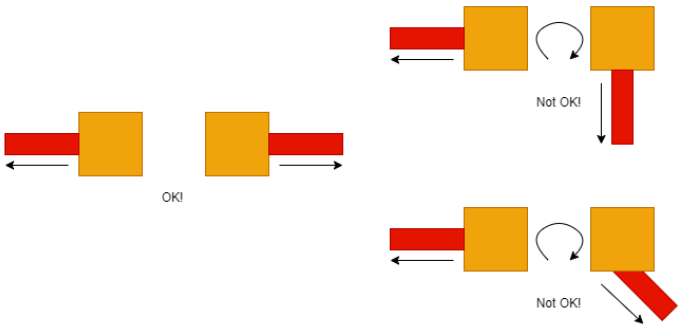

Figure 1. Tracing guide

C. Version Control

All PCBs created should have version control which is accomplished using Sharepoint/Teams or Github if working alone on a project. When working on a file it should always be in the checked out state to avoid collisions. To avoid files being checked out and forgotten all changes should be committed at the end of the day.

1) Create a new PCB:

- Create a new folder in Deliverable/<Deliverable>/PCB in Sharepoint/Teams. The folder and project should have a descriptive name.
- Create a KiCad project locally on your computer
- Upload all the files to Sharepoint/Teams and continue to section II-C2.

2) Update existing:

- Open the correct folder in Sharepoint/Teams and select all files.
- To check out the files, press the three dots on the action bar and select check out.
- If the checkout is not successful make sure that someone else does not have the files checked out already.
- If the checkout is successful then proceed to either download all of the files to a local folder OR create a shortcut on your OneDrive (instructions in section VII).
- To commit the changes upload the files to the folder and then select check in and enter a suitable comment.

D. Testing standard

The following section describes what and how tests should be performed before integrating the PCB in to the solar vehicle. The test should be performed independently by at least two people to reduce risk of human errors.

1) Before applying voltage to the PCB it should be visually inspected for short circuits.
2) The direction of directional components such as diodes and ICs should be visually inspected.
3) Resistance between traces should be measured and compared to theoretical values.
4) The testing should be performed using lab equipment with current limiter and NOT the battery.
5) The voltage and current consumption should be measured and saved to the documentation for each module

III. 3D printing design standard

The following sections will cover the design standards for 3D printing parts for the solar vehicle. 3D printed parts should not be used for encapsulation as it does not comply with the Encapsulation design standards.

A. Design

- The files should be saved using version control described later in the document.
- The plastic must not soften when exposed to temperatures around $80°$C.
- The sliced file should be stored with the CAD files to allow reproduction of the part.

B. Testing standard

The following section describes what tests should be performed before integrating the 3D-printed part in to the solar vehicle.

- The item should be exposed to $80°$C and tested to confirm the structural integrity.

IV. Component selection

In order to easily implement hardware in to the different designs of the solar car, some careful considerations about hardware needs to be addressed. This is intended for final application, for development this is not as important.

A. Hardware to avoid

- SPI modules
- Mitsubas motor controller terminal
- TBC...

B. Encouraged hardware

- CAN modules
- I2C modules
- BLE modules
- STM32 nucleo development boards.
- Specially designed PCBs

C. External power considerations

The Bridgestone World Solar Challenge (BWSC) regulations tightly regulates the amount of energy storage that the solar vehicle can utilise. This must be considered when implementing safety features and wireless modules that re not connected to the main power supply. The regulations states that 2 Wh is the limit on external energy storage sources, i.e outside of the battery container. This is equivalent to about 3 cr2032 coin batteries and thus is not a lot of energy. One battery needs to be present in order to have a working rtc module that retains the clock between power offs. In the 2021 iteration this battery is situated in the battery bms and thus is not counted towards the 2 Wh. This energy storage could be used for safe emergency shut down and letting **mcu!** (**mcu!**)'s saving all of the current data before power runs out. This would mean that the low voltage systems of the MCUs would have a capacitive energy backup that lasts a short amount of time.

V. Encapsulation

The electronics need to be protected against dust and moisture. Any electronics that are not placed inside one of the dedicated electronic boxes must have an ingress protection of at least IP64. The encapsulation should be modular and have waterproof connectors on the exterior. The cables inside the encapsulation shall not be soldered directly on to the PCB but must be connected using one of the connectors defined in the standards document. Cables within the box should be kept as short as possible prevent overheating. Holes bored in the encapsulation must be sealed using appropriate sealant or make use of a cable entry system.

VI. Documentation

The following items should be included in the documentation if applicable.

- The name of the creator and date.
- A brief description of the hardware purpose.
- A guide on how the item is intended to be used. (User guide)
- A list of components used - Bill of Material (BoM).
- Links to data sheets and components.
- Required libraries to interface with the device.
- Code examples.
- If components has an address (IP, i2c, CAN, etc.) they should be provided.
- Component specification (Supply voltage and current consumption).
- Specifications for the chosen enclosure.
- Functional block diagrams.
- Electronic schematics.
- future work

VII. OneDrive

If you want to setup mirroring and easier access of the files from teams you can setup OneDrive. It can be done by the following step. NOTE: Using OneDrive require extra responsibility. It is not possible to check out a file from OneDrive, one must do this through Sharepoint/Teams before working on it.

1) If not done already, download and install the OneDrive client from Microsoft here OneDrive.
2) Link OneDrive to your MDH email. This can be done by right clicking on the taskbar icon and selecting settings -> Account -> Add an account.
3) Open the directory containing the folder with the PCB files in Sharepoint.
4) Rightclick the folder you wish to work on and choose *Add shortcut to OneDrive*.
5) There should be a new shortcut in your MDH onedrive with the same name as the PCB folder now. If the status has a padlock it means that someone else has checked out the file.