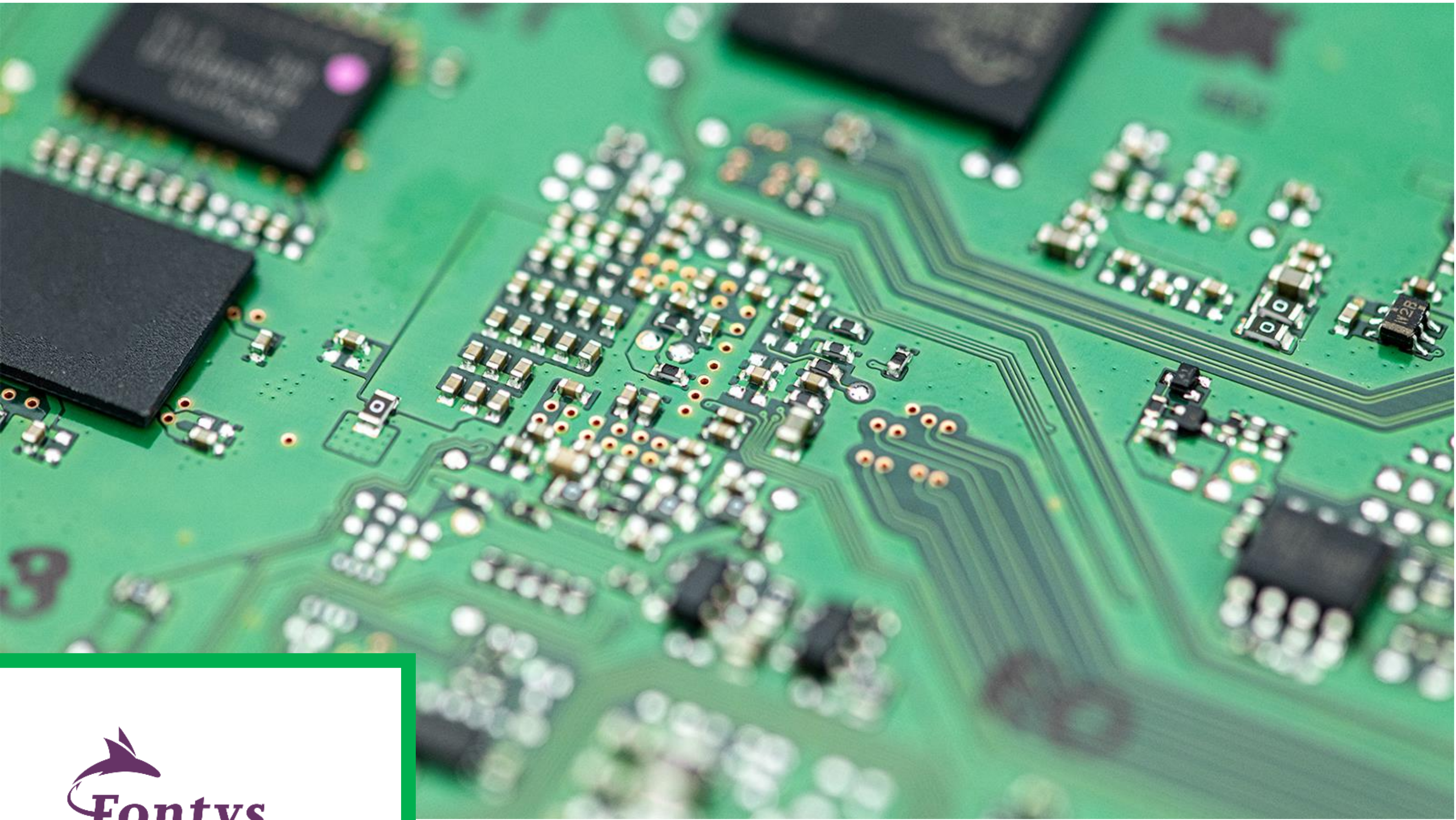

Fontys
University of Applied Sciences

PRINTED CIRCUIT BOARD DESIGN

Report for the Project DCRC at minor Adaptive robotics, Fontys engineering

Author	:	Max Jans
Student number	:	3497046
Minor	:	Adaptive Robotics
Project	:	Decentralized Cross Robot Control
Teacher	:	Esmaeil Najafi
Date	:	04/07/2021
Version	:	1.2

General information

Project:

Project name	:	Decentralized Robot Control PCB design
Part of	:	Minor Adaptive Robotics, DCRC, Fontys Engineering

Information student:

Name	:	Max Jans
Study	:	Fontys Engineering Mechatronics
Student number	:	3497046
Email	:	max.jans@student.fontys.nl

Index

List of figures

1. Introduction

This document is written for the design process of a Printed Circuit Board for the DCRC project of the Adaptive Robotics Minor. This project is about the designing and prototyping of a decentralized cross robot control system which should be able to be used on all kinds of different AGV's
To make this possible a control box needs to be placed on the AGV's, this box is called the CoLAB, which is an abbreviation of Communication and Localization Add-on Board. This box will include a Raspberry Pi to get tasks from a decentralized controlling robot which is running on all the CoLABs but only performed by one robot, even when that one stops running another robot takes over the task of sending out tasks.

Figure 1: CoLAB

To make this all work flawlessly the CoLAB should have a constant and regulated voltage source, in the case of a Raspberry Pi this is 5V. To make sure this 5V is always available, a Printed Circuit Board (PCB) should be made to regulate the varying voltages of the AGV's that the CoLAB could be placed on, but it should also have a backup battery in case the AGV runs out of charge. In the case the AGV runs out of charge the CoLAB can still send out that it will not be able to perform any more tasks so the system does not assign this AGV to tasks that it will not perform.

2. Assignment overview

To get the PCB to work with the Raspberry Pi and all available input voltages the PCB should be designed, simulated and eventually tested. Luckily, a prototype for such a PCB was already made by a prior project, the problem with that PCB is that it is around three times as large as it has to be as can be seen in figure 2 and not thought through on all parts.

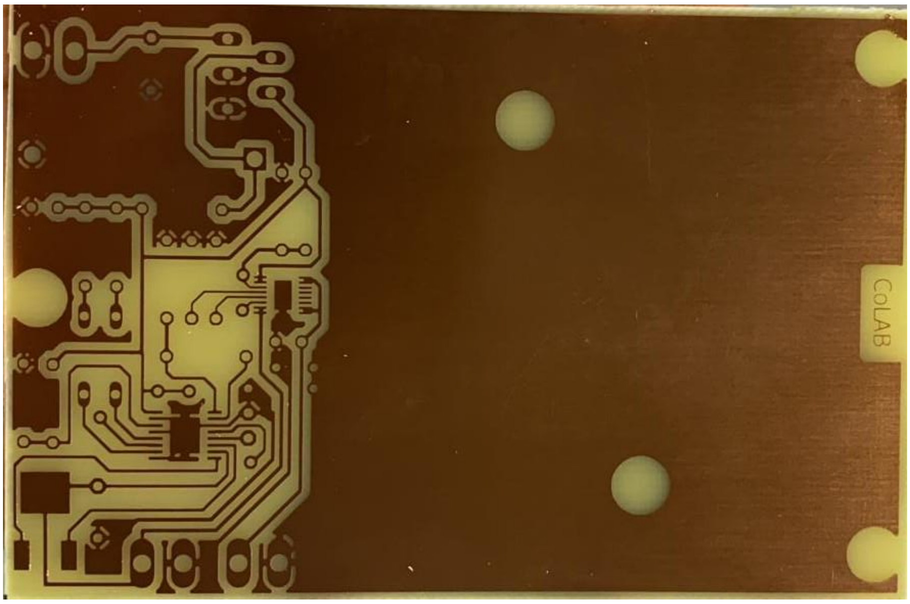

Figure 2: Prototype PCB

To make sure these problems do not reoccur in a new design, the components on this prototype PCB should be thoroughly researched to make sure they do fit in this design, the prototype PCB should be soldered to make sure the IC's work as they should in the final design and lastly, a simulation of a new PCB design should be made with the outcomes of the previous tasks. If everything goes according to plan, a new PCB can be designed in Autodesk EAGLE to be made and put in the CoLABs for further testing in real world scenarios.

3. Component research

To make sure the components that are chosen for the prototype PCB are compatible with the case it will be used for, research will be done to make sure this is the case. If the components are compatible no research for replacements will be performed since the components have already been ordered by the prior project group. The reason research will still be performed for these components even tough they are already there is because not a lot of reasoning was given by the prior group on why these components were chosen.

3.1 Voltage regulator

The main part of the PCB will be the voltage regulator, this voltage regulator will, as the name suggests, regulate the incoming voltage from the AGV. Since most AGV's have either 5, 12 or 24V output voltages for external devices, these three voltages should all be able to be accepted by the PCB to output a regulated 5V output. For this the prior group used the LM2596T 5V voltage regulator IC, this IC is ideal for this use since it can handle up to 40V (input) and 3A (output), both of these are around 1.5 times the amount of what is expected to be the maximum of this use. Also, the other components needed for this circuit to work are widely available for these high voltages. The schematic that is used for this IC is shown in figure 3 below.

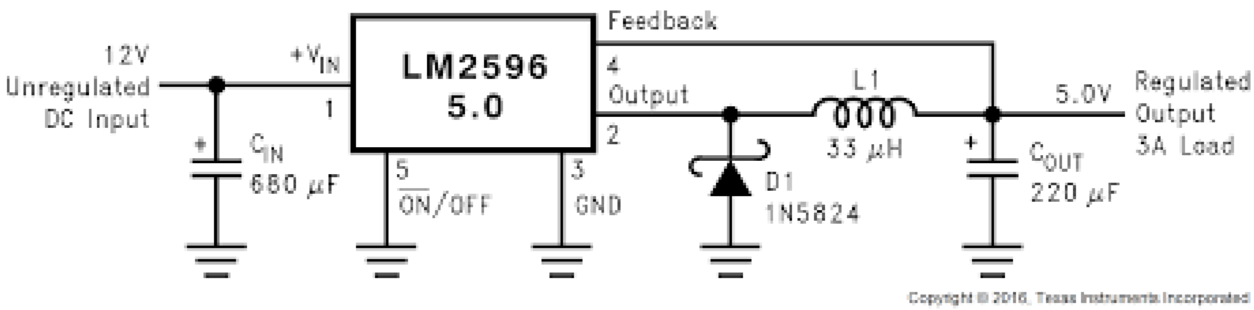

Figure 3: LM2596T 5V IC

This IC is already ordered and therefore also ready for use already.

3.2 Battery charger

To charge the internal battery of the CoLAB, a battery charger IC is needed. The batteries that are used are NiMH batteries, these batteries are widely available, easy and safe to use and have a high cycle count before they break down. For the charging of the batteries, another IC is placed on the PCB, the LTC4060. This IC is designed to charge up to four NiMH batteries in series, which outputs 4.8V fully charged, which is around the same as the regulated voltage from the DC-DC converter, and therefore also suitable for the Raspberry Pi to use. The LTC4060 is an independent IC, which means it does not need an external microcontroller to function, it prevents over- and undervoltage. The schematic with which this IC is used can be seen in figure 4 below.

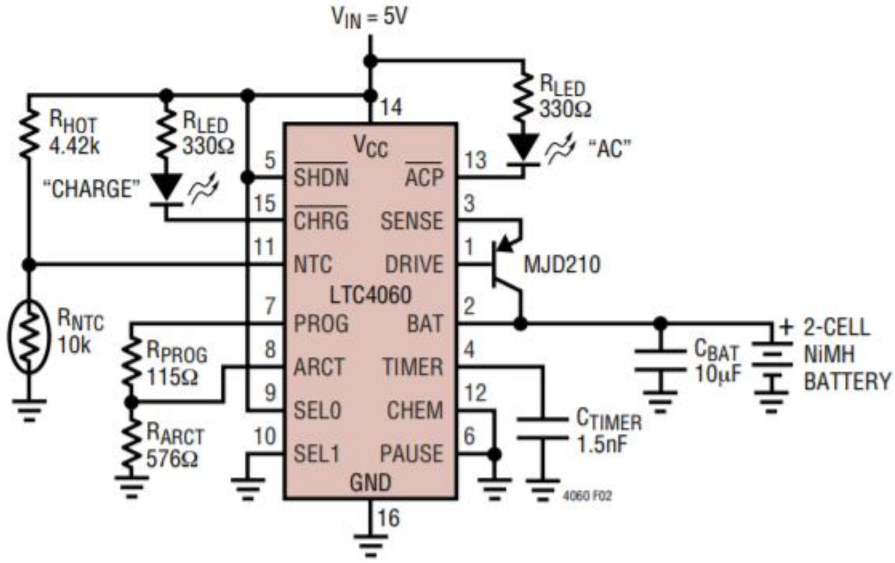

Figure 4: LTC4060

Not only will this IC charge the NIMH batteries, it will also show when it is charging via a LED and light up another LED when the voltage is low, which can be used for as a visual add-on for the user to make sure things are not going wrong.

3.3 Power selection

Now the voltage from the AGV can be output to 5V no matter what is put in, and a battery is added in case the AGV runs out of charge, but to choose which of these two power sources to use another IC is added to the PCB. This IC is the LTC4415, in normal circumstances, this IC will connect the AGV's power to its output, but when this voltage drops below ~4.5V it will switch its output to the Battery's power. This will make sure that even if the AGV's power falls away, the CoLAB will still be functional. The schematic of this IC is shown in figure 5 below.

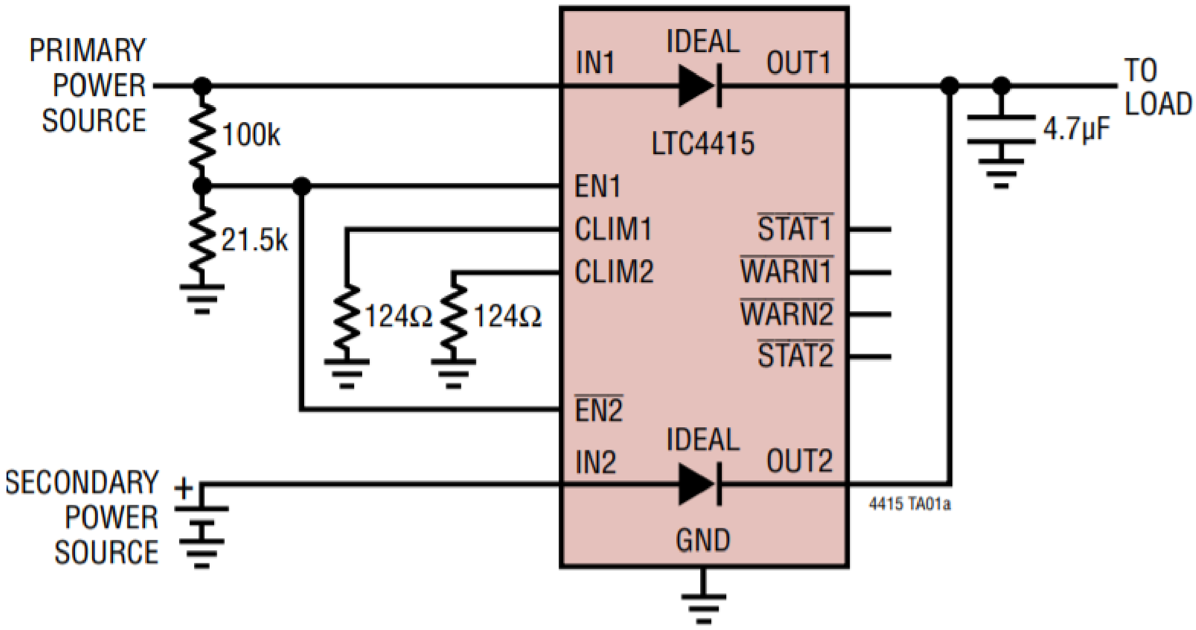

Figure 5: LTC4415

4. Testing prototype PCB

Because the components that are designed into the prototype PCB were indeed components that are suitable for this use, the prototype PCB will be soldered for the first time to see if it functions like it is supposed to be. Because there are three SMD components on this PCB some of the soldering should be done with special machinery or with an extremely steady hand, so the option to solder these components with the machinery at Fontys' location Nexus was chosen. Those components took a while to be added to the board because the machinery was not available at a moment's notice, which meant a delay in this process because nothing else could be soldered on before these SMD components were soldered on.

After some waiting time the SMD components were finally on, but then the next design flaw came up. All the components should be on the top of the PCB, but the copper faceplate was also only on the top side which meant the though hole components needed to go on the underside, which did make multiple components difficult to solder on, and therefore added more delay to this process. After this was resolved the PCB could be hooked up to a power supply and tested, but then the next problem arose.

The voltage regulator did not seem to work like it should it only had an output of 1.2V although 5V should be the output, which also meant the next ICs in line did not turn on and were not able to be tested on functionality. After a lot of testing with different circuits that could be found on the internet and the group's own thoughts, no solution was found, other priorities were set, and this testing was set to the background for some time.

After some time when other priorities were worked out the testing on this PCB came back. Then it turned out the mistake was not fully the fault of the current project group. The prior project group had said the LM2596T 5V version was bought, but it turned out to be the adjustable version, which needs a few external resistors to get a 5V output. The IC does say "-ADJ" on it but that was thought to be a number from the manufacturer, but it turned out to be an abbreviation of "Adjustable", this explained why the PCB did not work before, after some resistors were added in a less than optimal way to the PCB, it fully functioned. The LED's turned on when they needed to and the output voltage did not change, even when the input voltage was changed between 9 and 29V the output did not change by anything. This means the PCB now works perfectly as it should and a new and improved PCB can be designed.

5. Simulation

To make sure the PCB still works for the new design, and it was not a coincidence that some of the right resistors were used, the group used Multisim to simulate the circuit.

This did have some problems, only the voltage regulator is in the Multisim program, other free programmes for simulation that we found did either also not include the other ICs or were too difficult in use which meant we had to learn the program for longer than was still left in the project.

This let to the group to only simulate the IC that was changed from the previous design, from testing the prototype PCB it was concluded that everything other than that IC needed no changes in design. The result of the simulation can be seen in figure 6 below, and shows exactly what was expected, a regulated 5V output.

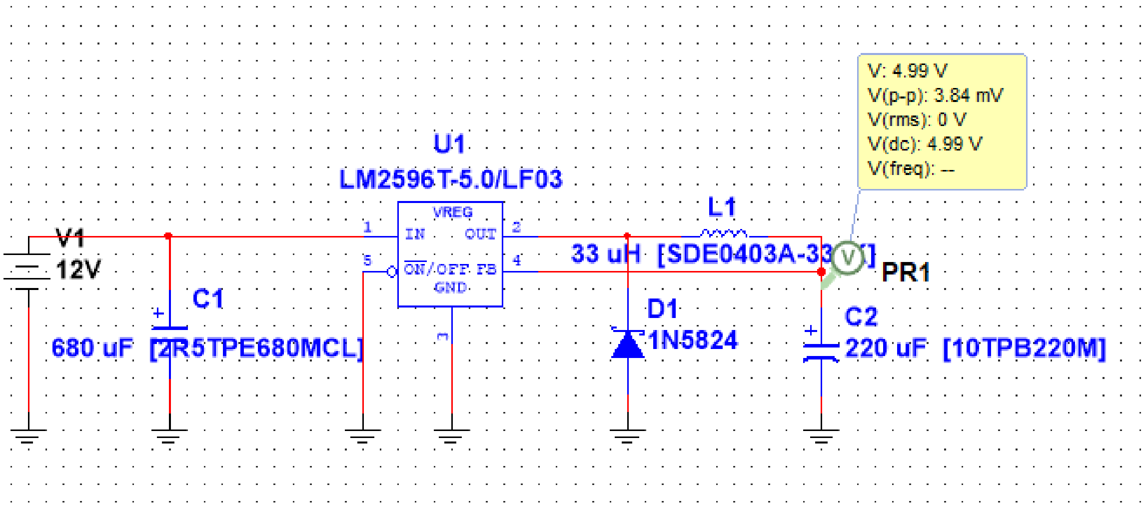

Figure 6: PCB simulation

From this the conclusion was that the PCB could be designed with the few changes that were made during testing.

6. New design

Because testing and the simulation concluded that a valid circuit is there a PCB was designed from this circuit, which is shown below in figure 8.

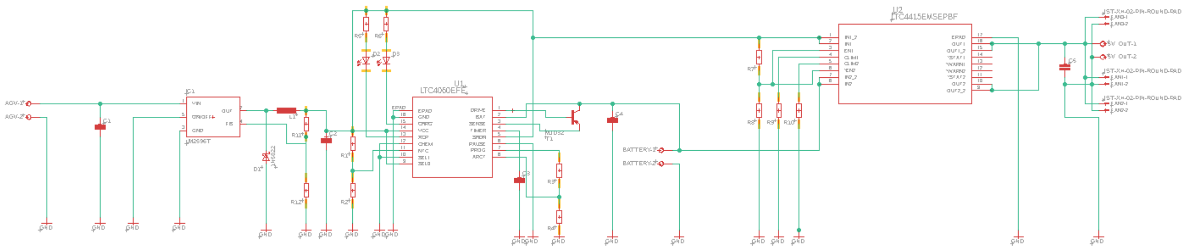

Figure 8: PCB schematic

From this circuit the PCB was designed in Autodesk Eagle, this design is shows below in figure 7.

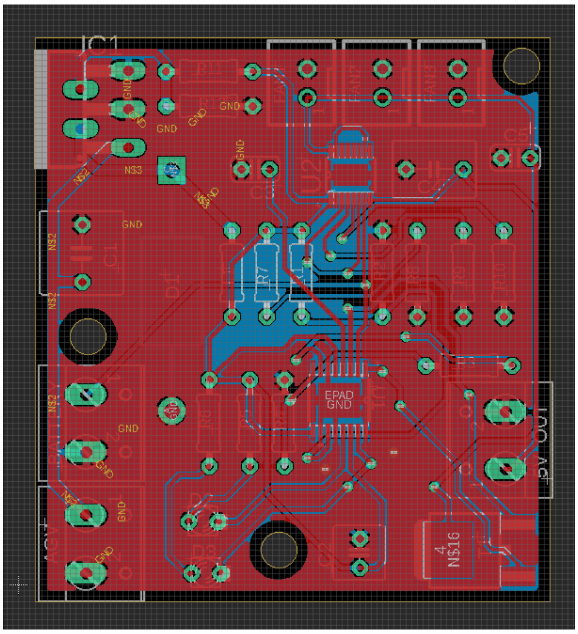

Figure 7: PCB design

To design this PCB some more unknown rules were used, such as rules for the width of the PCB traces (basically the cables on the PCB), this were made 32 thousands of an inch wide, which corresponds to the currents that we expect to be using. For the traces that needed to go into the SMD components, they are as wide as the components' legs allow it to be to not be touching each other.

The PCB gets the input from the AGV and outputs 5V for the Raspberry pi and charges the battery, all of these via screw terminals so any kind of connector can be used. Also three separate outputs for fans are added, the CoLAB currently only has two fans but with this design the CoLAB can be expanded in the future.

Now there is only a single problem left, making the actual new design. Sadly because of time restrictions the new design could not be ordered (yet), so no testing has been done on it. But looking at the result of the prototype PCB and the simulation, the group has no doubt that this design would work and fit very well in the designed CoLAB box since that was designed in combination with this PCB.

7. Conclusion & Recommendations

From this report it can be concluded that the new improved PCB design should work. The group has also made sure that the PCB would be easy to integrate into to CoLAB box for which it was designed. Even though several setbacks were overcome, they were not overcome quick enough to put out a working final product, only a working prototype and a final design.

Except for letting the PCB be designed by a professional with more experience and knowledge on how to make it work more efficient or to make the design smaller, not a lot of improvements can be performed on the PCB design. The only recommendations that this group wants to give to the next:

- Order the new design from a professional factory, it will make it look a lot better, will take less time, and the change of success will be much higher.
- Start with soldering as soon as possible, the process might be more difficult than expected.