

HST Project S5

CircuitVoyager Pre1



Submitted by:
Joel Schaller

Lecturer: Enrico Malacarne

22.09.2023

Abstract

Konzept (gestalterisch)

Methode

Wichtigste Ergebnisse

Contents

1	Introduction	1
1.1	"Lastenheft"	1
1.2	Mindmap	2
2	Main Body	3
2.1	"Pflichtenheft"	3
2.2	Extension PCB	4
3	Conclusion	8
3.1	HW development	8
4	Appendix	9
4.1	Journal	9
4.2	Project Agreement	10
4.3	GANTT Chart	11
4.4	Weekly plans	12
4.5	Extension PCB Schematics	14
5	Credits	15
	Bibliography	16
	List of Figures	17
	List of Tables	18
	Listings	19
	Acronyms	20

1 Introduction

The goal of this project is to develop a tiny extension Board for the STM32H747i-Disco Board, to allow it to act as a DMM. Additionally, a SW, that measures the DMM Values and displays them on the Touch Display. If there's more time I could extend the Project with Measurement Logging via a SD-Card or over USB to a Desktop application.

I want to learn how to implement high speed protocols such as Mipi DSI or QPSI. Later in the last year of my apprenticeship I'd like to develop a whole DMM on my own, but with a different approach as standard ones like these from Fluke. For example, I want to make the DMM rechargeable and modernize it a bit.

To realize this project I'm going to use the following tools: Altium Designer, STM32-CubeIDE, LaTeX, TouchGFX.

Also I won't make a diary, because it's easier for me to write my findings down sorted by theme rather than date. But to keep the chronological order of the stuff I've done, there's a Journal in Chapter: [4.1].

1.1 "Lastenheft"

This is a request from the imaginary customer, I'm making this project for:

I need a prototype for a DMM, that can measure voltage, current and continuity. The DMM should have a touchscreen that displays said values. The UI should be intuitive, so everyone who's ever used a DMM can use it to. Normal features as hold, minmax should be available and it would be great if you could fit in a power mode, where the DMM uses the voltage and current measurement to calculate the drawn power from the measured device. Because this project will only be used for the proof of concept, the DMM doesn't have to support mains voltage and we also won't need any safety circuits, AC measuring or negative voltages / currents. It's mainly about the SW. So you can also use DevBoards if there are any available.

1.2 Mindmap

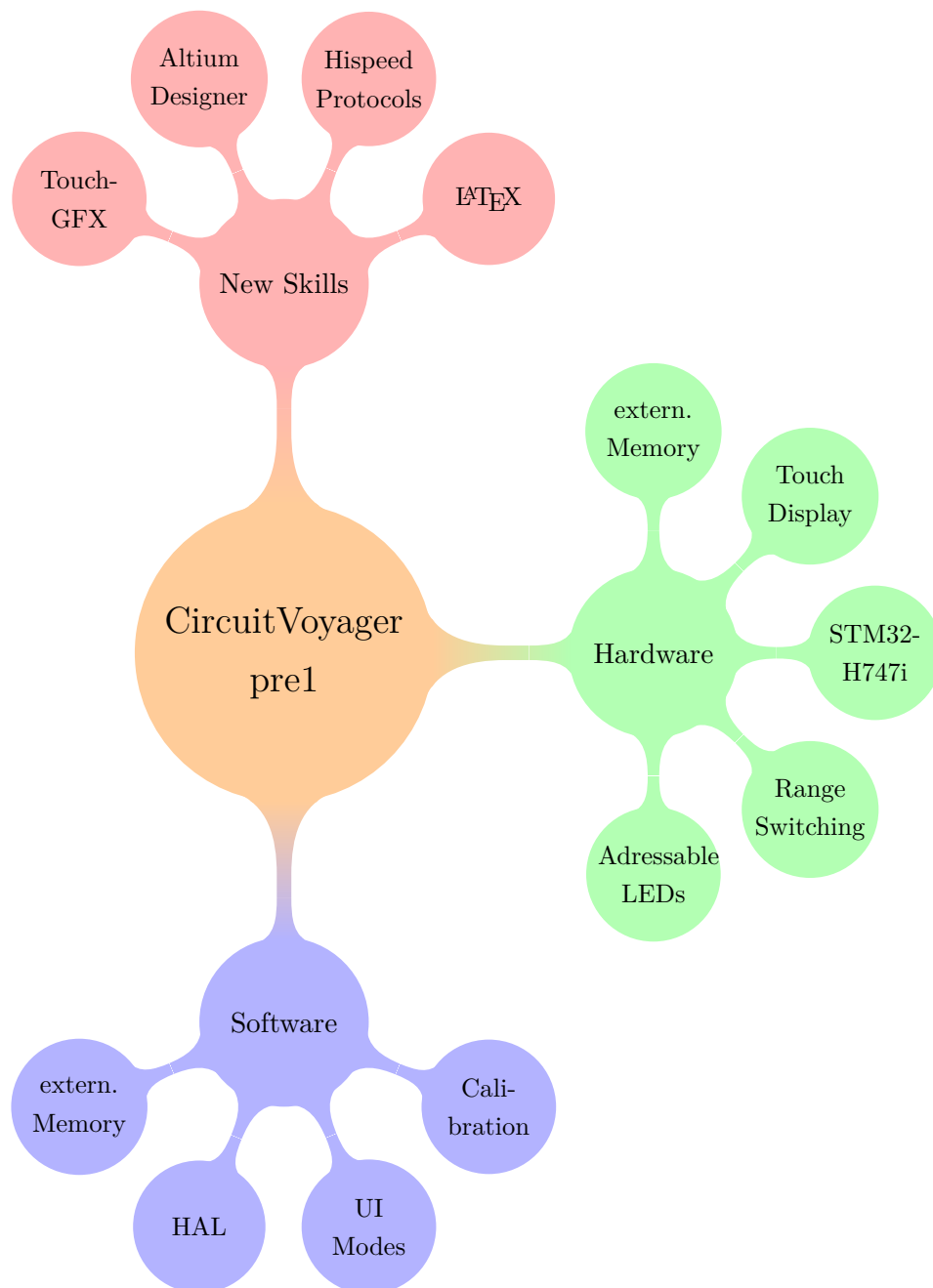


Figure 1.1: Project Mindmap

2 Main Body

2.1 "Pflichtenheft"

Cost

I've already bought two DevBoards one of them stays at TBZ and the other is at home. One of these boards was paid by Mr. Malacarne. Further expenses from the PCB will be paid by me and shouldn't exceed about 50 CHF, as the HW isn't that complicated.

Time

The most time of the project I will work at home because it's a rather big project to execute in one semester. I will also have much time in the fall holidays to work on it. The project will approximately take 100h to complete. Also the more detailed timeplan is in chapter: [4.3]

Tools

To realize this project I will mainly use, the SW STM32CubeIDE with HAL and Altium Designer. The documentation is written in LaTeX in VSCode. And I'm planning to order the PCB on JLCPCB and I will populate and reflow the PCB at ETHZ, where I'm also allowed to use the measurement equipment for the HW tests.

Technical Details

value	min.	typ.	max.	unit	description
supply voltage		5		V	over USB
curent to measure	0		1	A	
voltage to measure	0		10	V	

Table 2.1: Technical Details

2.2 Extension PCB

2.2.1 STMod+

Interface from DevBoard to Extension PCB.

- 5V Supply
- SPI
- I₂C
- ADC
- Interrupt
- PWM
- GPIOs

I will use the STMOD#14 connection that was intended to use as PWM, as a second ADC input. To measure current and voltage at the same time to later show the power consumption of the DUT.

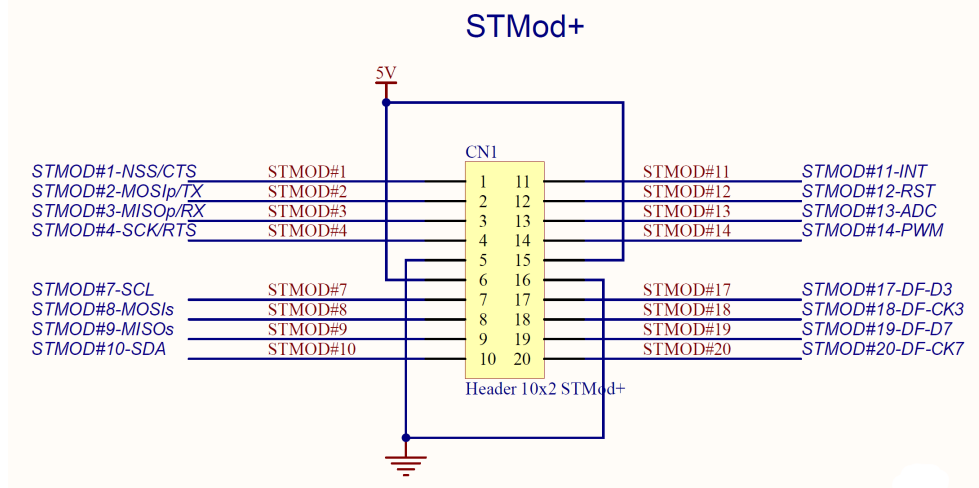


Figure 2.1: STMod+ Interface

2.2.2 Hardware concept

After some thoughts I came up with the following HW concept.

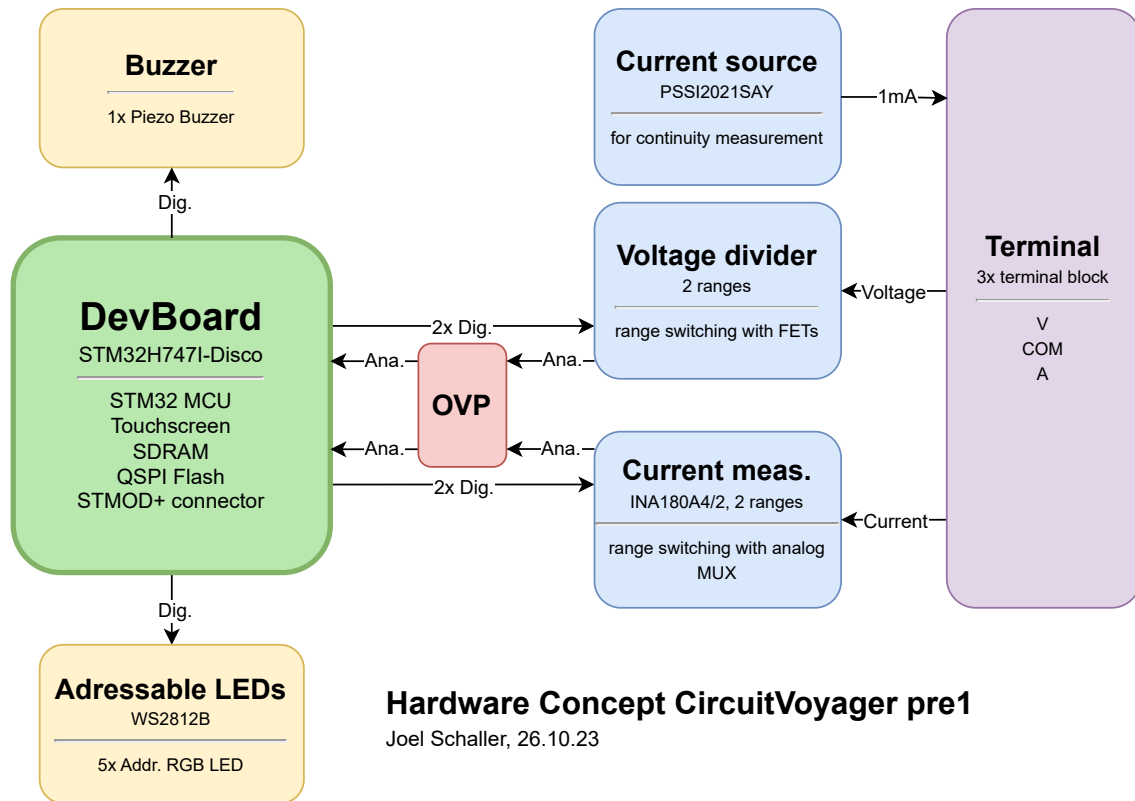


Figure 2.2: Extension PCB HW concept

Voltage measurement

To measure voltage, the DUT should be connected to the terminals V and COM. COM is connected internally to device GND. The V terminal is connected to the Voltage divider block. This block divides the input voltage down, so the ADC in the MCU doesn't overshoot. There are 2 ranges to measure voltage, which can be chosen by setting 2 digital output, that go from the MCU to the voltage divider. There's also an OVP, to protect the MCU from voltages higher than 3.3V. [1]

Current measurement

To measure current, the DUT should be connected to the terminals A and COM. COM is connected internally to device GND. The A terminal is connected to current measurement block. This block measures the current, by letting the current flow through one of two shunt resistors. The DMM can choose which resistor and

therefore range should be selected with the 2 digital Output that are connected from the MCU to the current measurement block. The voltage over the selected shunt is then amplified, by a current amplifier IC and then measured by the MCUs ADC. There's also an OVP, to protect the MCU from voltages higher than 3.3V. [1]

Continuity measurement

To measure continuity, both the voltage divider and the current source is used. The continuity between the V and COM pins is measured. For this a constant current produced by the current source is flowing out of the V terminal. Simultaneously the voltage across those terminals is measured and the resistance / continuity can be evaluated. If continuity is detected, either the buzzer beeps or the LEDs blink. [1]

2.2.3 Schematic

The schematic took me a bit longer than usual, because it's my first whole HW project in Altium before I used KiCAD and Altium is a lot more features and in my opinion is harder to learn. The schematic is in the Appendix [CHAPTER!!!! :)].

Buzzer Circuit

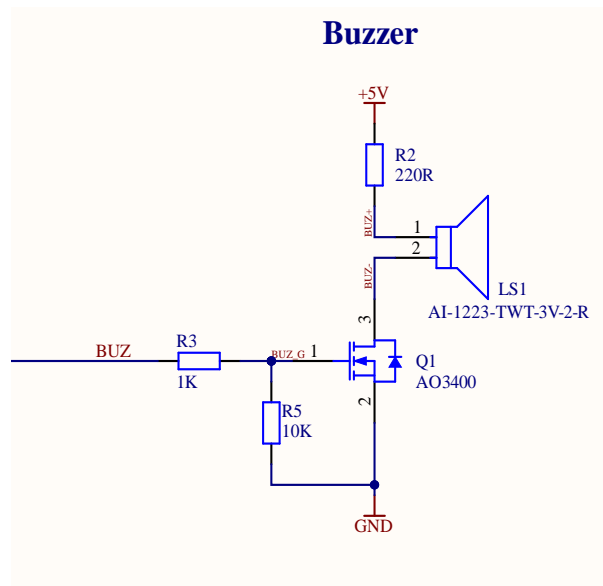


Figure 2.3: Buzzer Circuit

This is an active buzzer. If the BUZ line is pulled high by the MCU, the MOSFET starts to conduct and the buzzer starts beeping. This circuit will be used to give an acoustic feedback to the user, if for example a continuity has been detected.

The resistors R3 and R5 build a voltage divider with a ratio of 1/10. This has the advantage, that the gate capacitance of Q1 is charged with a limited current and if nothing's connected to the BUZ net the MOSFET turns the buzzer off and the whole machine isn't in an indeterminate state.

The resistor R2 limit the current flowing through LS1. As LS1 is rated for 30mA at 3V.

$$R_2 = \frac{U_{VCC} - U_{LS1}}{I_{LS1}} = \frac{5V - 3V}{30mA} = 66.\bar{6}\Omega$$

$$P_{R2} = I^2 \cdot R = (30mA)^2 \cdot 66.\bar{6}\Omega = 60mW$$

Finally, I've chosen a 220Ω resistor for R2. With this value the sound should be enough loud, that the user hears it. And the power loss of the resistor will be smaller. That means it should be perfectly fine to use a 0402 resistor that is rated for 62.5mW.

Adressable LEDs Circuit

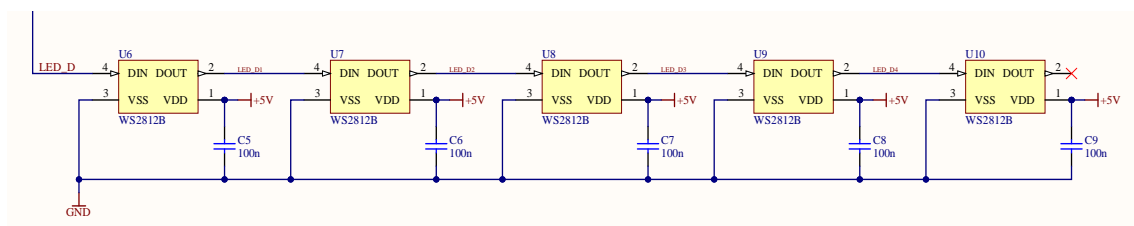


Figure 2.4: Adressable LEDs Circuit

An other option to show if continuity has been detected are these LEDs. The advantage of them is, that they only need one data connection and already include their logic and driving circuits. I've equipped the with one bulk C each, because they're integrated components and they're driven over the 5V rail, which they're specified for. But if they're driven by 5V they theoretically detect all voltages over 3.5V as a logical high. But the MCU only output 3.3V. I've used those LEDs much in past projects and this was never a problem, so I'm assuming that it should also work this time.

3 Conclusion

3.1 HW development

I've learned much in this project part. Mainly this was Altium Designer, as this was my first complete project I've realised in Altium Designer. This also led to some not nicely solved solution. All components for example have their own properties and this leads to an unreadable BOM.

But in the end I've also noticed, that the dataflow (that usually should go from left to right) goes in the wrong direction. This had already started in the HW-Chart and therefore also ended up in the schematic.

I've also used Draw.io one of the first times and by now it looks very promising. It's much easier and more straightforward than MS Visio. Everything just works as intended.

Gesamtschau, Arbeitsergebnis, Gesamturteil, evtl. Ausblick, was ich lernen konnte

4 Appendix

4.1 Journal

Date	Location	Duration	Activity
01.09.2023	TBZ	1.5h	Selected and bought DevBoard
08.09.2023	TBZ	2h	Tested DevBoard with demos
08.09.2023	TBZ	0.5h	Noted first ideas for DMM
15.09.2023	TBZ	1.5h	Written and signed Project Agreement [4.2]
21.09.2023	Home	3h	Created documentation template
22.09.2023	TBZ	2h	Started writing Journal [4.1]
24.09.2023	Home	1.5h	Made GANTT chart [4.3]
27.09.2023	Home	2h	Written detailed planning and introduction
29.09.2023	TBZ	1.5h	Added mindmap, Lasten-, Pflichtenheft
06.10.2023	TBZ	1h	Started with block diagram (extension PCB)
20.10.2023	ETH	0.5h	Started Altium Project, Schematic template
23.10.2023	ETH	3h	HW Concept / documentation
23.10.2023	ETH	3h	Start schematic / documentation
25.10.2023	Home	2h	Schematic: Current Src, Volt div
26.10.2023	ETH	1.5h	Schematic: Current meas, ERC
26.10.2023	Home	1h	Schematic: Cleanup, Comments, DS Saves
26.10.2023	Home	1h	Documentation & prepared Interview 1
27.10.2023	TBZ	1.5h	Documentation: Started with schematic part

Table 4.1: Project Journal

4.2 Project Agreement

Information <i>HST: BÜP</i>	TBZ/EE / 2337.00 jschaller Sem: 5	HST TBZ Technische Berufsschule Zürich Abteilung Elektro/Elektronik <small>1_Project_Agreement.docx</small>
--------------------------------	---	--

1 Projektvereinbarung

Verfasser/innen: Joel Schaller **Klasse:** BEN21
Titel: CircuitVoyager pre1

1.	Thema (Hintergrund, Überblick, gegenwärtiger Wissensstand) Develop a tiny extension Board for the STM32H747i-Disco Board, to allow it to act as a DMM. Additionally, a software, that measures the DMM Values and displays them on the Touch Display. If there's more time I could extend the Project with Measurement Logging via a SD-Card or over USB to a Desktop application.
2.	Eigene Fragestellung / Untersuchungsgegenstand 2.1 Eigene Fragestellung (Leitfrage) How to implement the following functions / protocols? (QSPI Flash, SDRAM, TouchGFX, Mipi DSI) and if the time is sufficient: (FAT with SDcards, Bootloaders) 2.2 Hypothese (Vermutung über das Ergebnis) I want to learn, working with High Speed MCUs and implement such protocols. 2.3 Methoden und Vorgehen (mindestens 2 Methoden müssen angewendet werden) HW-Dev (Altium), SW-Dev (STM32Cube with HAL), Documentation in LaTeX 2.4 Hilfsmittel Internet, literature 2.5 Kontaktpersonen, Informationsstellen, Institutionen Teachers, Instructor at ETH, Dad
3.	Persönlicher Bezug / Motivation In the next 2 Years I want to develop my own DMM, because I think there's much to improve with standard DMMs as Fluke. For Example: Touch Display, Rechargeable Battery...
4.	Bewertungsform This Project will only be done by me. Time: about 28 lesson and unknown time at home. Project delivery on: 12.01.2023
5.	Besprechungstermine mit Lehrperson (vorgeschrieben sind zwei Besprechungen) Termin 1: 27.10.2023 Termin 2: 01.12.2023
Datum: 15.09.23 Die Lernenden: Datum: 15.9.23 Die Lehrperson:	
<small>DMM = Digital Multimeter TouchGFX = Graphical Designer for Embedded Touch Displays</small>	

BEN21	Seite 1 (1)	1_Project_Agreement.docx
-------	-------------	--------------------------

Figure 4.1: Project Agreement

4.3 GANTT Chart

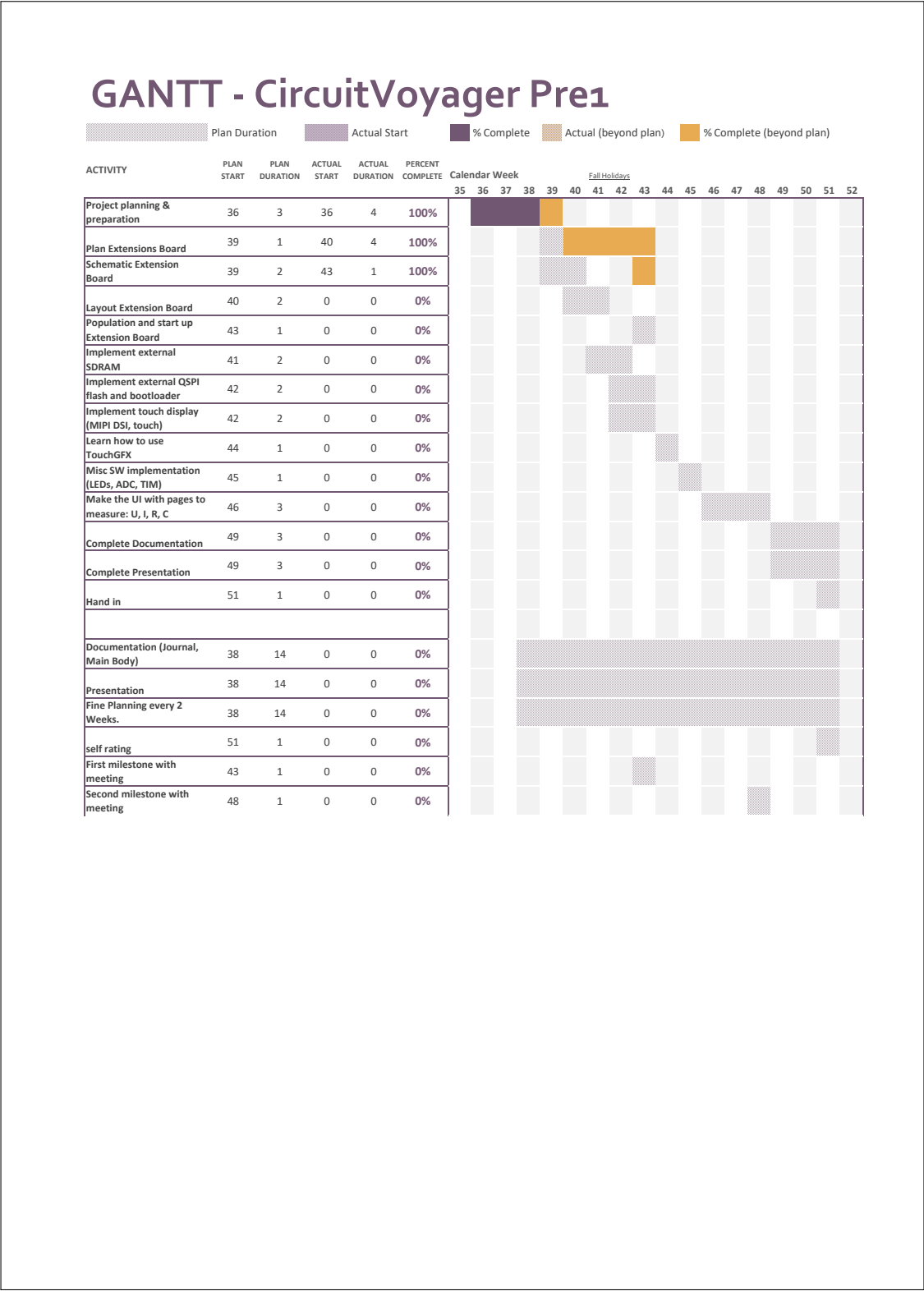


Figure 4.2: GANTT Chart

4.4 Weekly plans

4.4.1 KW39 & 40

- Write introduction
- Planning: Cost, Tools, When, Why
- Create project diagram (learning process)
- "Lastenheft"
- "Pflichtenheft"
- Make a HW-Digram for the Extension PCB.
- Make the schematic of the Extension PCB.
 - Part to measure voltage.
 - Part to measure current.
 - Part to measure continuity.
 - Addressable LEDs.
- Start with the Layout of the Extension PCB.
- Reflection of the start of the project.

4.4.2 KW41 & 42

Fall holidays. I planned to invest much time in the holidays, but it turned out that my plans changed, and I was busy.

4.4.3 KW43 & 44

Mainly catching up.

- Make a HW-Digram for the Extension PCB.
- Make the schematic of the Extension PCB.
 - Part to measure voltage.
 - Part to measure current.
 - Part to measure continuity.

- Addressable LEDs.
- Make the whole Layout of the Extension PCB.
- Order the Extension PCB and the components.
- Reflection of the start of the project.

and maybe if the time is sufficient I could start with implementing the SDRAM and QSPI Flash.

4.5 Extension PCB Schematics

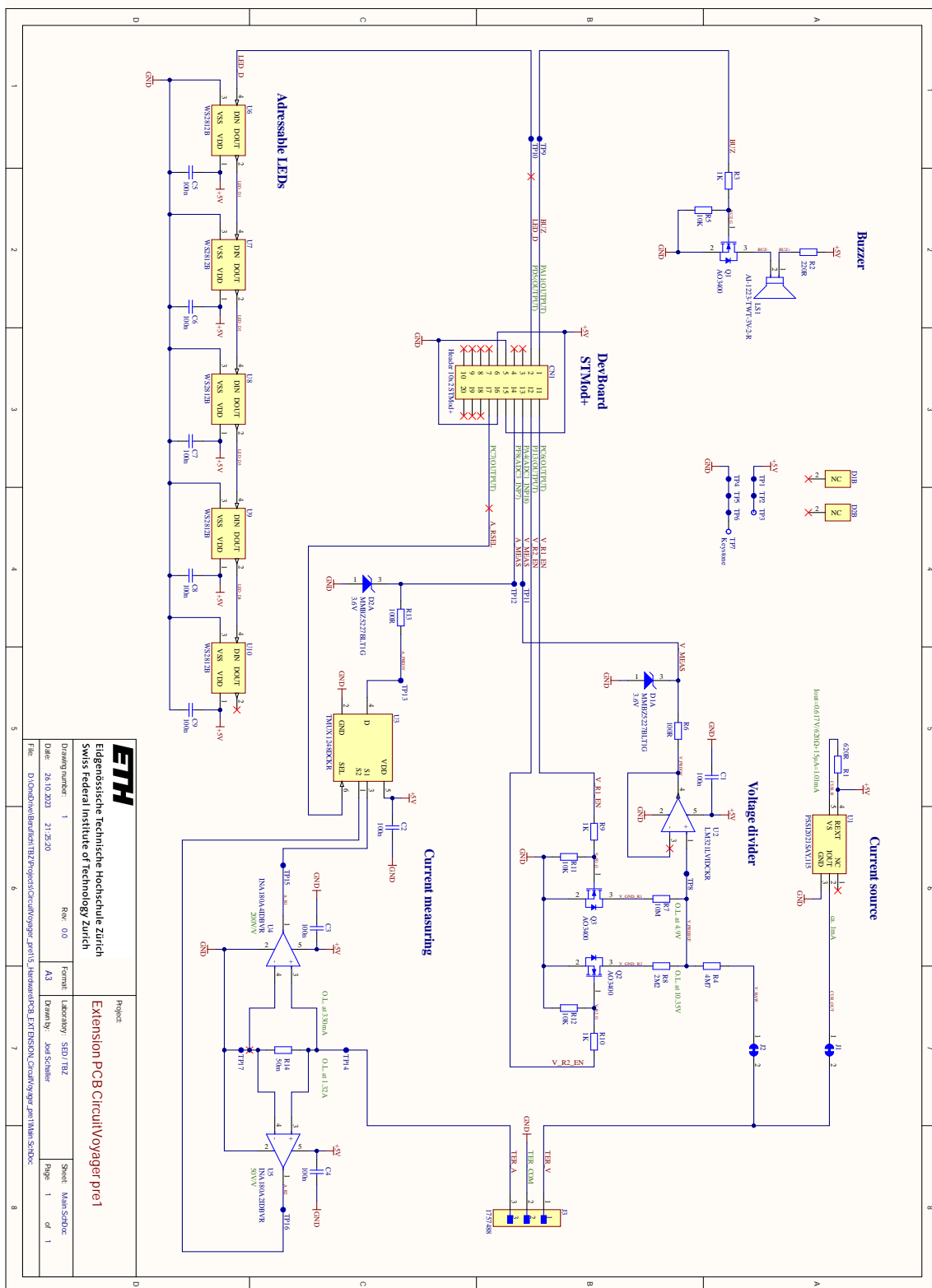


Figure 4.3: Extension PCB Schematics

5 Credits

Bibliography

- [1] ElectroNoobs, “Arduino 5 in 1 multimeter v2.0.” Available at http://electronoobs.com/eng_arduino_tut112.php (2019/11/29).

List of Figures

- 1.1 Project Mindmap 2
- 2.1 STMod+ Interface 4
- 2.2 Extension PCB HW concept 5
- 2.3 Buzzer Circuit 6
- 2.4 Adressable LEDs Circuit 7

- 4.1 Project Agreement 10
- 4.2 GANTT Chart 11
- 4.3 Extension PCB Schematics 14

List of Tables

2.1	Technical Details	3
4.1	Project Journal	9

Listings

Acronyms

CircuitVoyager The Name of the DMM I'm developing.

DevBoard main microcontroller development board. (STM32H747I-Disco)

DMM digital multimeter

HW Hardware

SW Software

QPSI Quad SPI

SPI Serial Peripheral Interface (low level protocol)

SDRAM Synchronous Dynamic Random Access Memory (external RAM)

TouchGFX Graphical UI designer for STM32 MCUs

UI User Interface

MCU Micro Controlling Unit

Mipi DSI Digital Serial Interface (Display Protocol)

FAT File Allocation System (Low Level Filesystem)

HAL Hardware Abstraction Layer (STM32 Abstraction Library)

ETHZ Eidgenössische Technische Hochschule

TBZ Technische Berufsschule Zürich

ADC Analog Digital Converter

TIM Timer (Hardware Block in STM32)

PCB Printed Circuit Board

DUT Device under test

OVP Over voltage protection