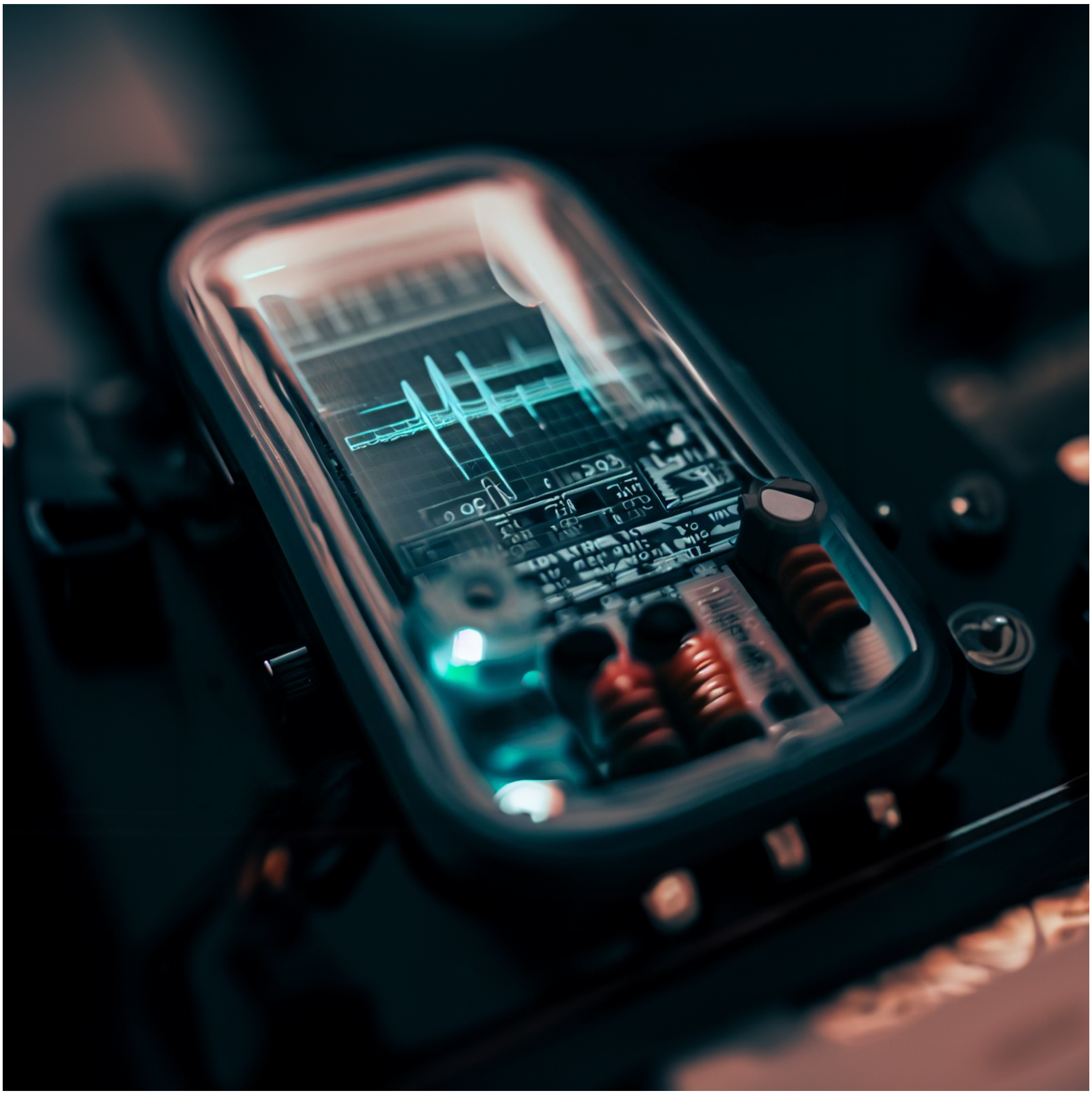
HST Project S5

CircuitVoyager Pre1



Submitted by:

Joel Schaller

Lecturer: Enrico Malacarne

22.09.2023

Abstract

Konzept (gestalterisch)
Methode
Wichtigste Ergebnisse

Contents

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

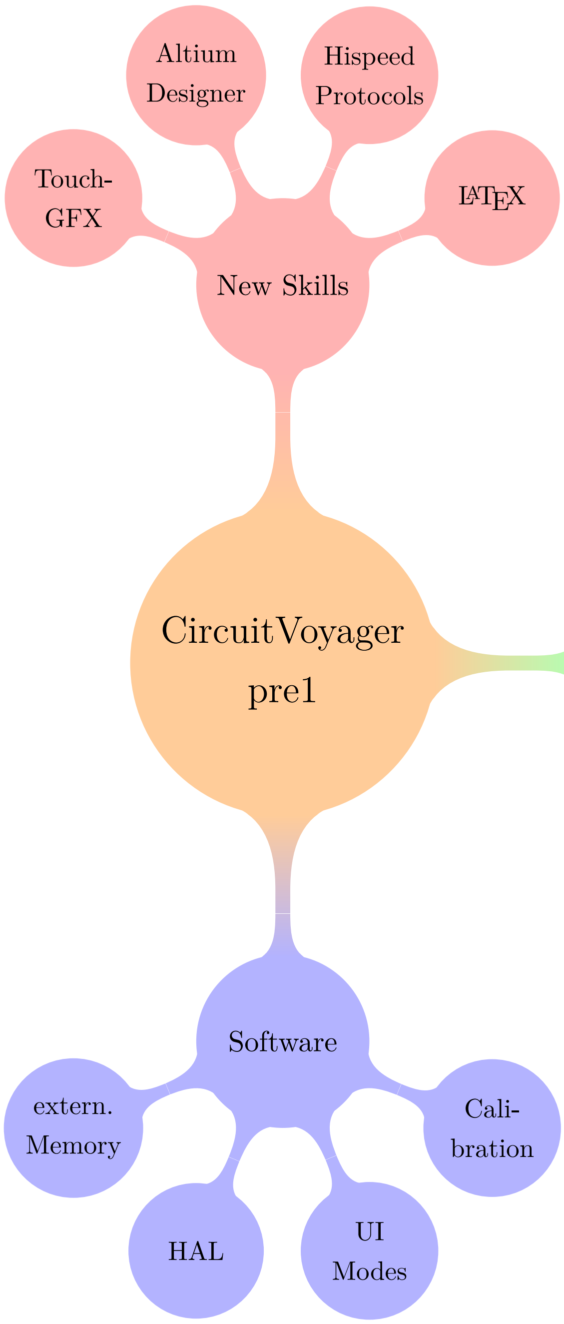


Figure 1.1: Project Mindmap

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
- I_2C
- 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 cosumption of the DUT.

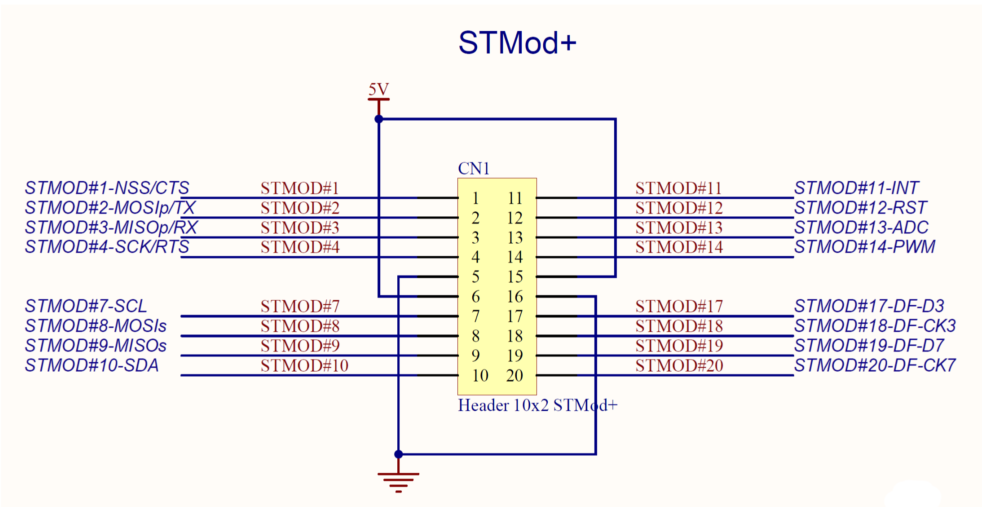


Figure 2.1: STMod+ Interface

2.2.2 Hardware concept

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

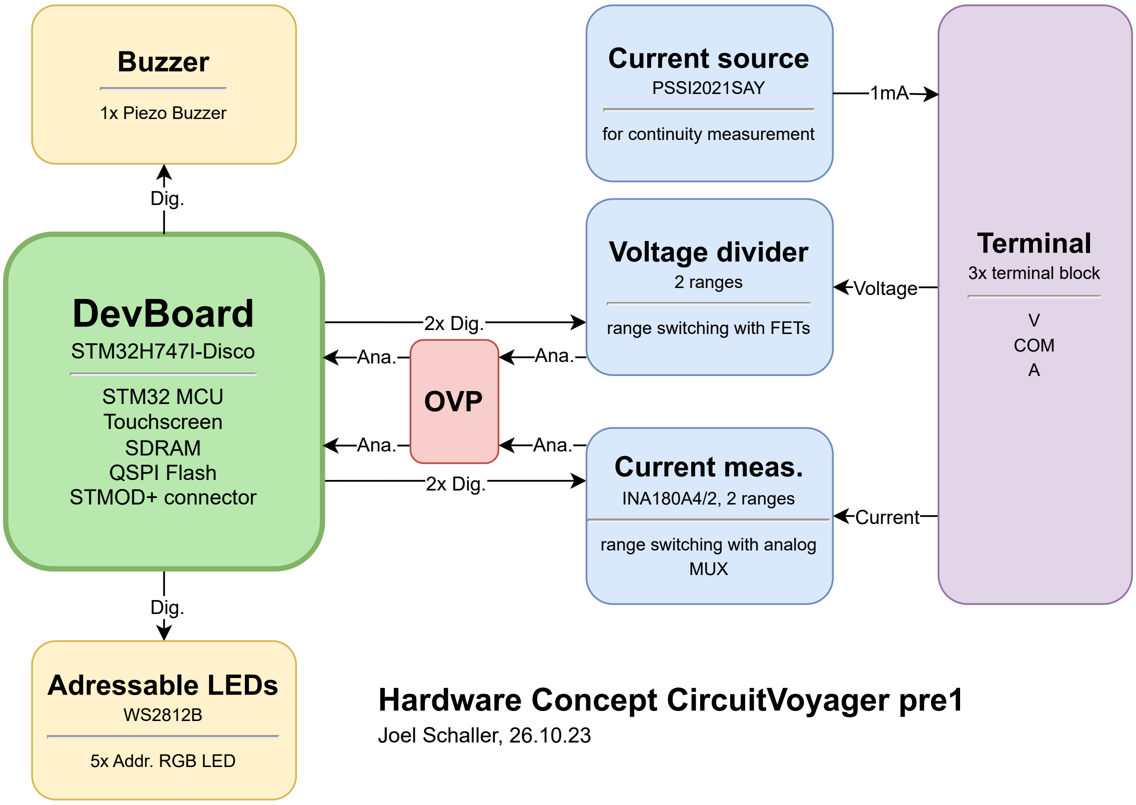


Figure 2.2: Extension PCB HW concept

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 4.5.

Buzzer Circuit

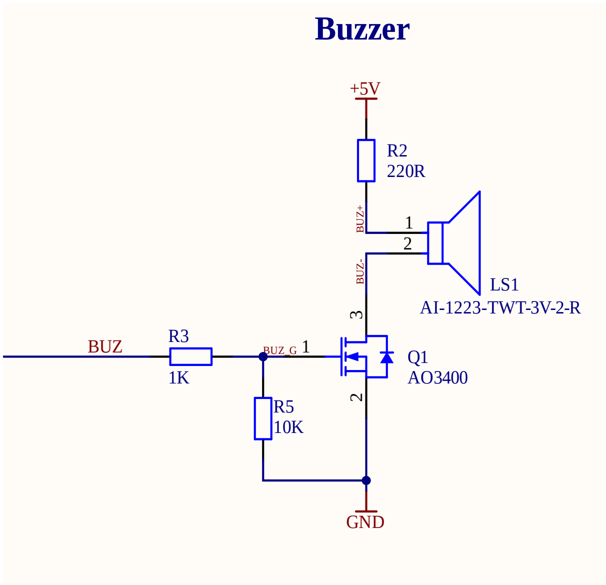


Figure 2.3: Buzzer Circuit

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.

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

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

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

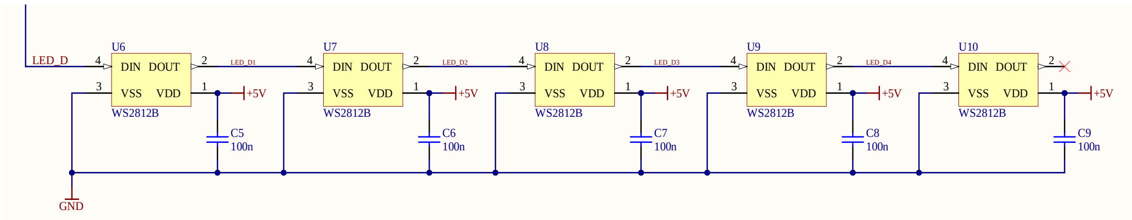


Figure 2.4: Adressable LEDs Circuit

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 equiped 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.

Overvoltage Protection Circuit

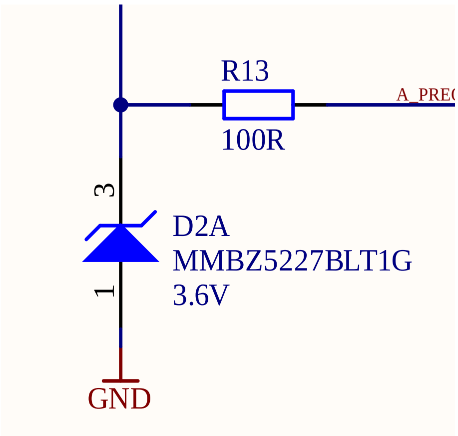


Figure 2.5: Overvoltage Protection Circuit

My Problem is, that the voltages coming from the Extension PCB, can't exceed 4V according to the MCUs datasheet. But because this PCB is powered by the 5V rail, theoretically these voltages could damage the MCU. So the goal was to develop a circuit which protects the ADC inputs. The first idea that came to my mind was to use varistors. I've already used the once in a project, but after reading some application notes on this topic, I decided to use a zenerdiode for the OVP. [2]
The zenerdiode I've used has a nominal zenervoltage of 3.6V (max. 3.78V) at 20mA. Together with R13 this should result with a maximum voltage of 3.78V at the ADC inputs. As long as the PREOV voltage doesn't exceed 5.6V.

$$U_{PREOV(MAX.)} = R_{13} \cdot I_Z + U_Z = 100\Omega \cdot 20mA + 3.6V = 5.6V$$

$$P_{R13} = R \cdot I^2 = 100\Omega \cdot (20mA)^2 = 40mW$$

Voltage Divider Circuit

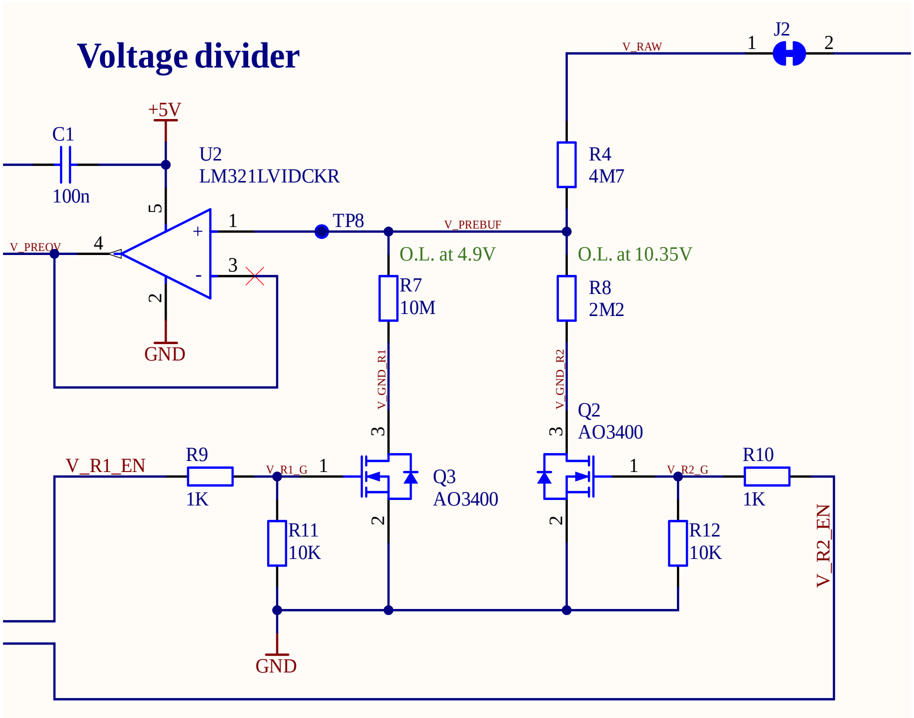


Figure 2.6: Voltage Divider Circuit

The voltage divider is used to measure voltages over the V terminal. This circuit is one approach to range switching, which is described more detailed in the following chapter [Write more detailed text about how to execute range switching and what are the pros and cons. Minimum 3 options]. This voltage divider features 2 ranges and I came up with this idea by myself. The ranges are calculated as following:

$$U_{OL(R1)} = \frac{(R_4 + R_7) \cdot U_{OL(MCU)}}{R_7} = \frac{(4.7M\Omega + 10M\Omega) \cdot 3.3V}{10M\Omega} = 4.851V$$

$$U_{OL(R2)} = \frac{(R_4 + R_8) \cdot U_{OL(MCU)}}{R_8} = \frac{(4.7M\Omega + 2.2M\Omega) \cdot 3.3V}{2.2M\Omega} = 10.35V$$

This divided voltage is then fed into an impedance converter which guarantees that the voltage divider isn't manipulated by the ADC internal resistance and also helps on the DevBoards analog paths, because they're very long and therefore vulnerable to electromagnetic fields.

Text to put into ranges part: This range switching part has the advantage of using many ranges at once, as for example with 2 range switches can use 3 different ranges and with 3 range switches can use 7 ranges. On the other hand it has the disadvantage of only being able to divide

the input voltage and not being able to amplify. Additionally, the internal resistance is always changing and therefore impacting the DUT. I wouldn't recommend this circuit because of said factors and only switch on the measured side. For example use different Amps / Divs to create the right voltage ranges and only switch the high Z path, that doesn't have an impact on the DUT.

Current Source Circuit

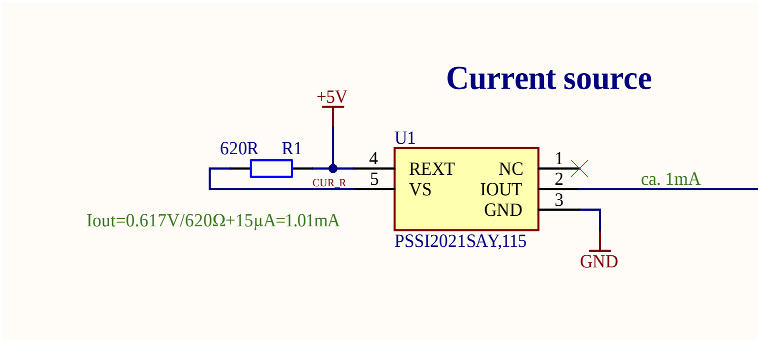


Figure 2.7: Current Source Circuit

The current source circuit will be used to measure continuity and resistance together with the voltage measuring function. A constant current will be induced to the DUT and because the voltage over the DUT can be measured, the resistance can be calculated. The constant current can be calculated as following:

$$I_{OUT} = \frac{U_{VS}}{R_1} + 15\mu A = \frac{0.617V}{620\Omega} + 15\mu A = 1.01mA$$

But this doesn't matter that much, because the DMM will be calibrated using the solderbridge J1.

Current Measuring Circuit

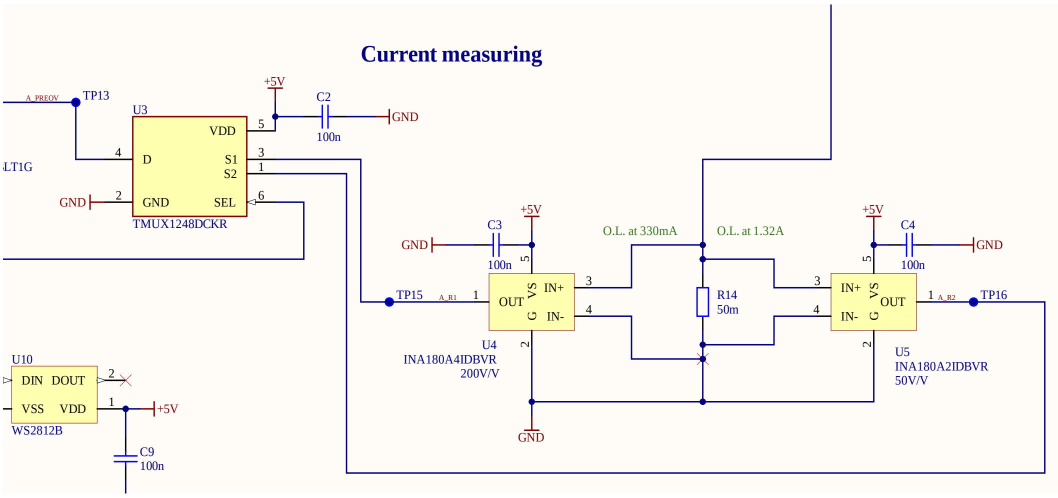


Figure 2.8: Current Measuring Circuit

The current measuring circuit is used to measure currents with the A terminal. This circuit is one approach to range switching, which is described more detailed in the following chapter [Write more detailed text about how to execute range switching and what are the pros and cons. Minimum 3 options]. This circuit features 2 ranges and I came up with this idea, while reading some articles on DMMs [3] [4]. The ranges are calculated as following:

$$I_{OL(R1)} = \frac{U_{OL(MCU)}}{R_{14} \cdot G_{U4}} = \frac{3.3V}{50m\Omega \cdot 200\frac{V}{V}} = 330mA$$

$$I_{OL(R2)} = \frac{U_{OL(MCU)}}{R_{14} \cdot G_{U5}} = \frac{3.3V}{50m\Omega \cdot 50\frac{V}{V}} = 1.32A$$

These voltages are then fed into an analog MUX, which switches the processed voltages to the ADC. This has the advantage, that the DMM doesn't manipulate the actual path, where the current is flowing through and therefore not influencing the DUT.

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 leaded to some not nicely solved solution. All components for example haver their own properities and this leads to a 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 straight forward than MS Visio. Everthing 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
28.10.2023	Home	2h	Documentation: schematic parts
29.10.2023	Home	1h	Docu: schematic done, resources cleanup
29.10.2023	Home	0.5h	Layout: Outline, Placement start
31.10.2023	Home	1h	Layout: Placement Done
01.11.2023	Home	1.5h	Layout: Routing Done
02.11.2023	Home	1h	Layout: Cleanup & Export

Table 4.1: Project Journal

4.2 Project Agreement

Information	TBZ/EE / **2337.00** jschaller	HST
HST: BüP	*Sem:* **5**	1_Project_Agreement.docx

TBZ Technische Berufsschule Zürich Abteilung Elektro/Elektronik

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: ....

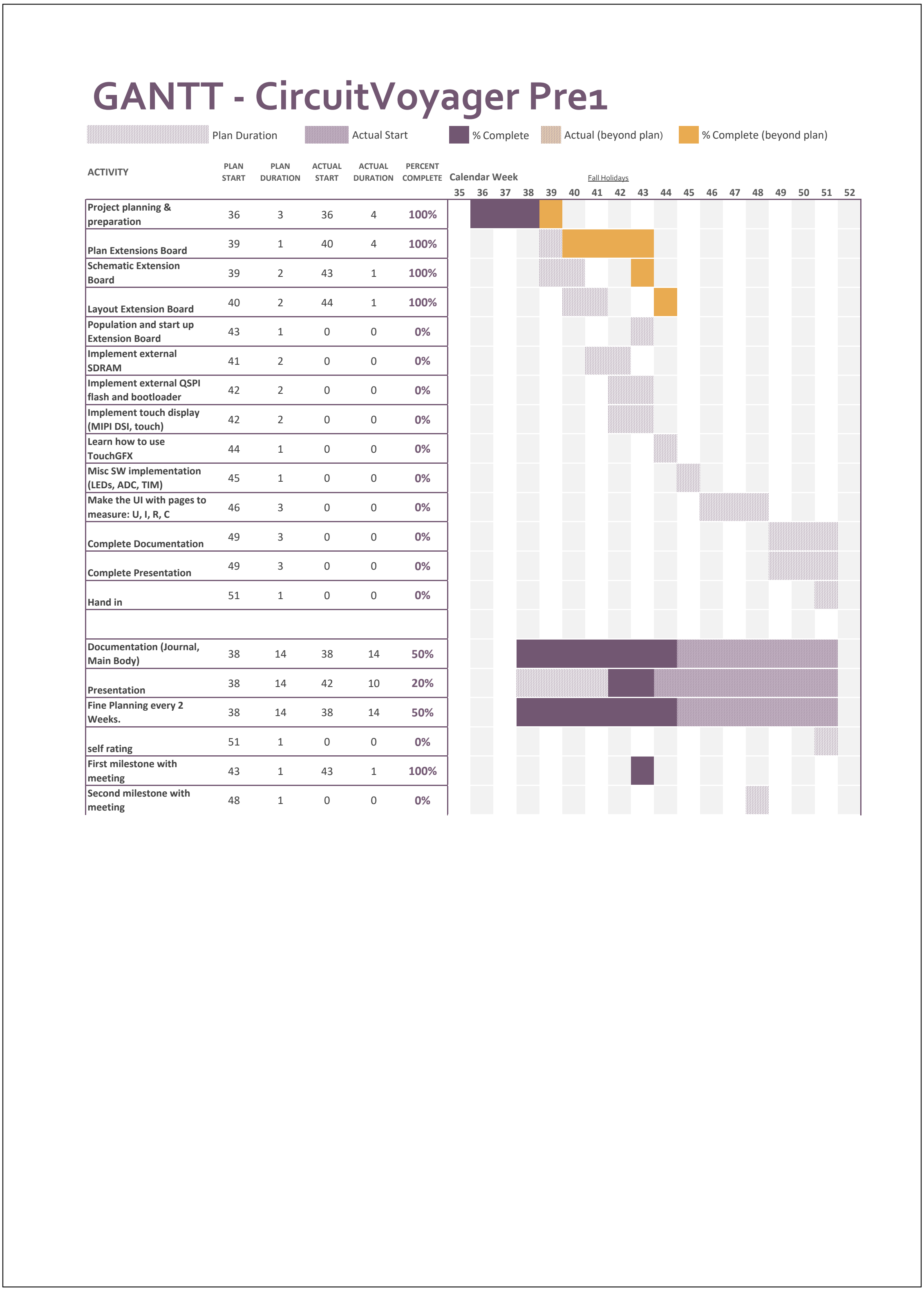
Datum: 15.9.23 Die Lehrperson: ....

DMM = Digital Multimeter
TouchGFX = Graphical Designer for Embedded Touch Displays

BEN21 Seite 1 (1) 1_Project_Agreement.docx

Figure 4.1: Project Agreement

4.3 GANTT Chart



ACTIVITY	PLAN START	PLAN DURATION	ACTUAL START	ACTUAL DURATION	PERCENT COMPLETE
Project planning & preparation	36	3	36	4	100%
Plan Extensions Board	39	1	40	4	100%
Schematic Extension Board	39	2	43	1	100%
Layout Extension Board	40	2	44	1	100%
Population and start up Extension Board	43	1	0	0	0%
Implement external SDRAM	41	2	0	0	0%
Implement external QSPI flash and bootloader	42	2	0	0	0%
Implement touch display (MIPI DSI, touch)	42	2	0	0	0%
Learn how to use TouchGFX	44	1	0	0	0%
Misc SW implementation (LEDs, ADC, TIM)	45	1	0	0	0%
Make the UI with pages to measure: U, I, R, C	46	3	0	0	0%
Complete Documentation	49	3	0	0	0%
Complete Presentation	49	3	0	0	0%
Hand in	51	1	0	0	0%
Documentation (Journal, Main Body)	38	14	38	14	50%
Presentation	38	14	42	10	20%
Fine Planning every 2 Weeks.	38	14	38	14	50%
self rating	51	1	0	0	0%
First milestone with meeting	43	1	43	1	100%
Second milestone with meeting	48	1	0	0	0%

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

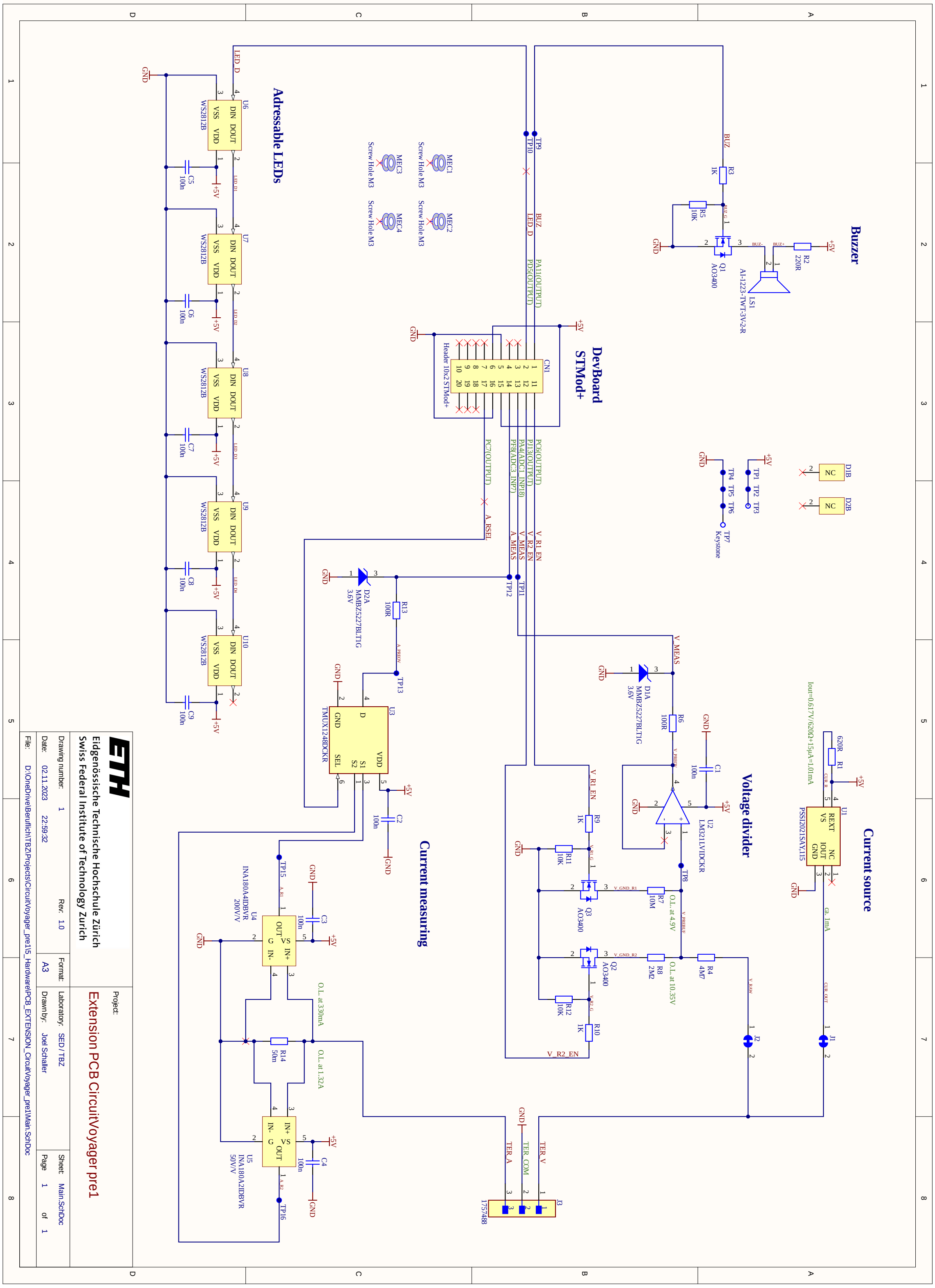


Figure 4.3: Extension PCB Schematics

5 Credits

Bibliography

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

[2] A. Walsh, “Protecting adc inputs.” Available at https://www.analog.com/en/technical-articles/protecting-adc-inputs.html.

[3] A. Garaipoom, “Digital multimeter circuit using icl7107.” Available at https://www.eleccircuit.com/digital-multimeter-circuit-using-icl7107/ (2022/08/16).

[4] T. Instruments, “An-202 a digital multimeter using the add3501.” Available at https://www.ti.com/lit/an/snoa592c/snoa592c.pdf?ts=1698230599203&ref_url=https%253A%252F%252Fwww.ti.com%252Fsitesearch%252Fde-de%252Fdocs%252Funiversalsearch.tsp%253FlangPref%253Dde-DE%2526searchTerm%253Dmultimeter%2526nr%253D5689 (2015/02).

List of Figures

List of Tables

Listings

Acronyms

CircuitVoyager The Name of the DMM I'm developing.

DevBoard main microcontroller developement board. (STM32H747I-Disco)

DMM digital multimeter

HW Hardware

SW Software

SPI Serial Peripheral Interface (low level protocol)

SDRAM Synchronous Dynamic Random Access Memory (external RAM)

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

TouchGFX Graphical UI designer for STM32 MCUs

QPSI Quad SPI

OL Overload

MUX Multiplexer