Toasted Power Board

TP3s6sSVC - 3s & 6s LiPo to Standerd Voltages' Converter

Independent board features:

MT3608-
• 2V to 24V Input Voltage
• Up to 28V Output Voltage
• Internal Compensation
• Up to 97% Efficiency
• Internal 4A Switch Current Limit

LM2576XX (HVT Version)-
• Up to 60V Input Voltage
• Up to 57V Output Voltage
• ±4% maximum overline and load conditions
• Up to 90% Efficiency
• 3A output current

Application:

TBH what do u really think a fricking power board means?
If u really don't know, you should... ahhh u thought I was gonna say "you don't deserve this sheet" or something like that?
Naah

So basically this board, (which took me fricking HOURS of rework and AHHHHGGGGGG.....) converts the standard 3S and 6S LiPo Batteries which have the 11.1V and 22.2V respectively to standard voltages (specifically 3.3v, 5v, 12v, 15v and 24v).

Description:

The Toasted Power Board has the versatility to be used in a broad range of applications. 3mm mounting holes permit them to be mounted to a chassis, cabinet wall, or bracket, or they may be used on a test bench or tabletop. Screw terminals provide easy connection without sockets or soldering and also Input/output isolation prevents ground loops.
It also has an individual SPST switch for each output terminal

Schematics:

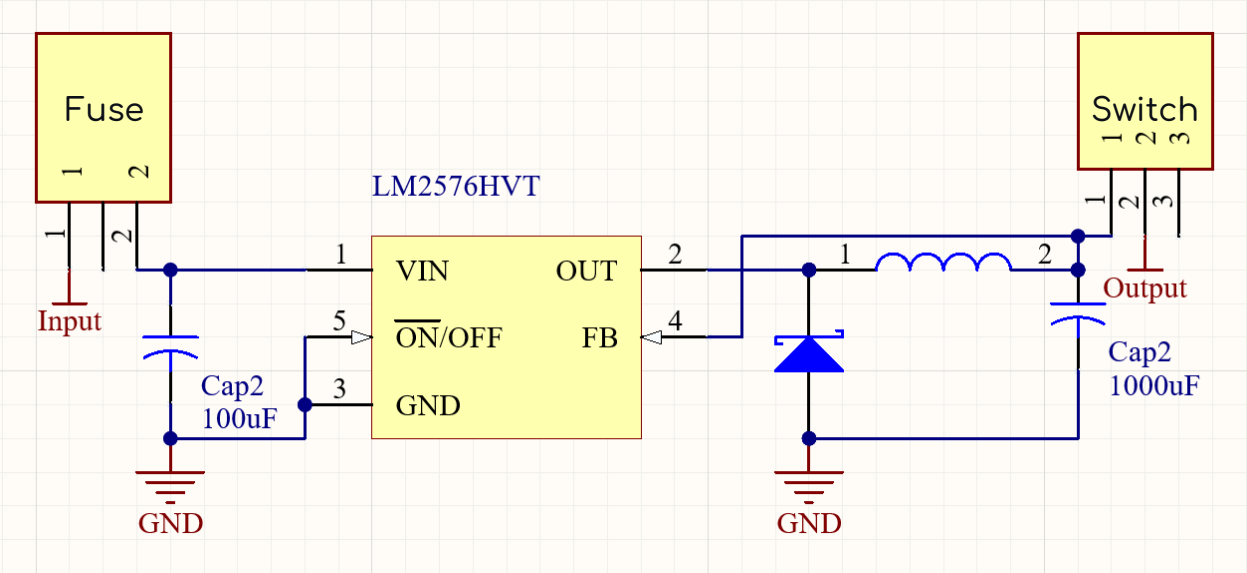

Fig 1.1 - LM2576HVT-XX Buck Circuit

Datasheet of LM2576XX -
https://www.ti.com/lit/ds/symlink/lm2576.pdf?ts=1644981846639&ref_url=https%253A%252F%252Fwww.google.com%252F

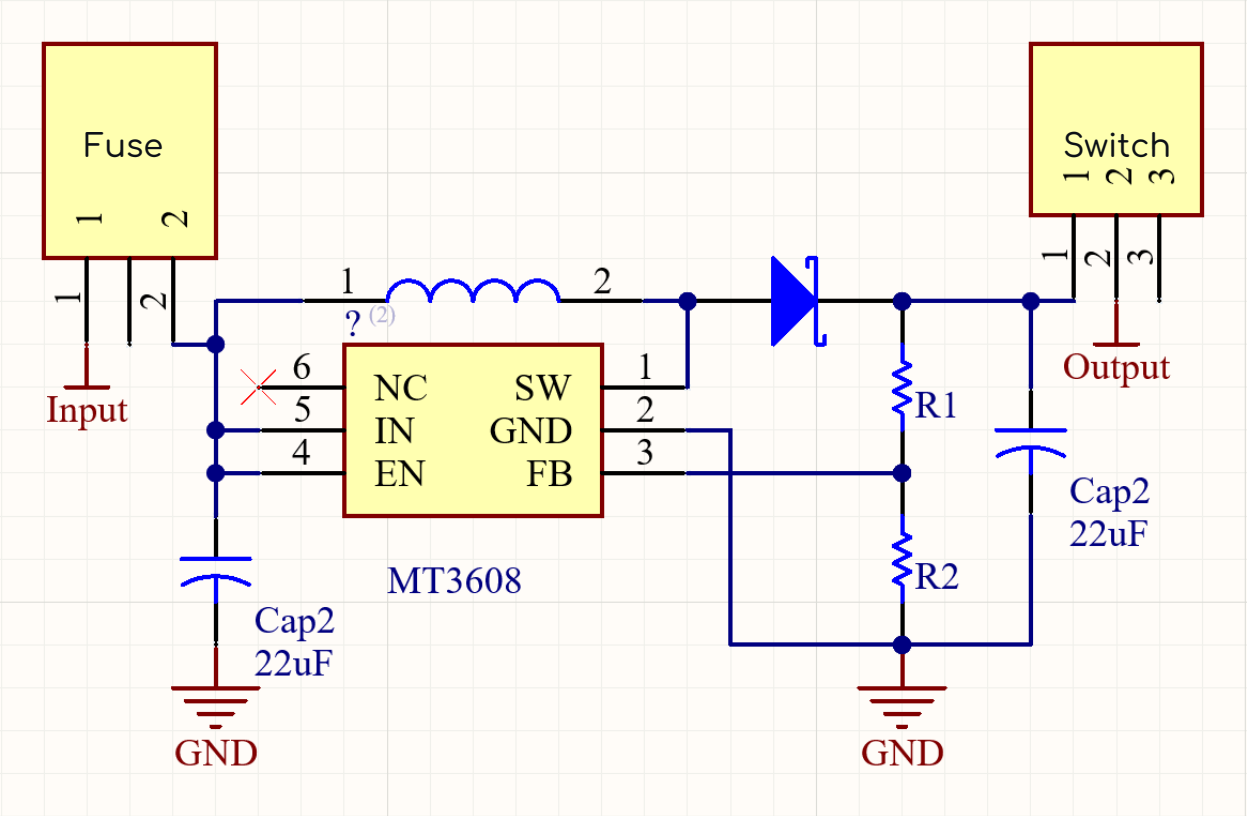

Fig 1.2 - MT3608 Boost Circuit

Datasheet of MT3608
https://www.olimex.com/Products/Breadboarding/BB-PWR-3608/resources/MT3608.pdf

My Brain:

Now, in this part, I will talk about how I came about using the circuit designs (Fig. 1.1 & 1.2). I'll keep it short and simple. I started out doing a bit of digging about what the hell is 3s and 6s batteries.
After understanding what 3s and 6s LiPo batteries are, their use, C rating, storage and charging, i/o pins, and other things, I went ahead and did some more research on some small details like the nominal voltage, etc.

Now I just needed to figure out how I will be controlling the current and voltage to the standard forms for the output.
I came across some notes I made during training about buck/boost converters. Went on youtube and did some more research and found a couple of ICs which can be used as buck/boost ICs but the reason I choose the above ICs are because of the availability, both on physical stores nearby in India and also the footprint for Altium which is the software I have used to make the board.

Now, let's come on to the components used. I have used the recommended values of resistors, capacitors, and inductors as given in the datasheet of the respective ICs.

For the LM2576,
the Input Capacitor is used to maintain stability, the regulator input pin must be bypassed with at least a 100-µF electrolytic capacitor as specified in the datasheet.
An output capacitor is required to filter the output voltage and is needed for loop stability.

$$\text{Output Ripple Voltage} = (\Delta I_{IND})\ (\text{ESR of } C_{OUT})$$

We also use a diode to provide a return path for the inductor current when the switch is off. This diode is placed close to the LM2576 using short leads and short printed-circuit traces. I have used Schottky diodes as it provides the best efficiency.

For the MT3608,
For the inductor selection, the recommended values of the inductor are 4.7 to 22µH and hence I have used the highest value just to be on the safer side.
Input and output ceramic capacitors of 22µF are recommended in the datasheet.
For the diode selection, I have used a Schottky diode because of its low forward voltage drop and fast reverses recovery.

$$I_D\ (\text{RMS}) \approx \surd I_{OUT} * \surd I_{PEAK}$$

The diode's reverse breakdown voltage should be larger than the output voltage.

Circuit Diagram/PCB Schimatic:

So the following is the circuit diagram/ schematic as a image so u guys can easly modifiy or repair the PCB when needed.
*all the files including the schematic, gerber and altium specific files will be on my git page (github.com/BakaToast/toastedpowerboard)

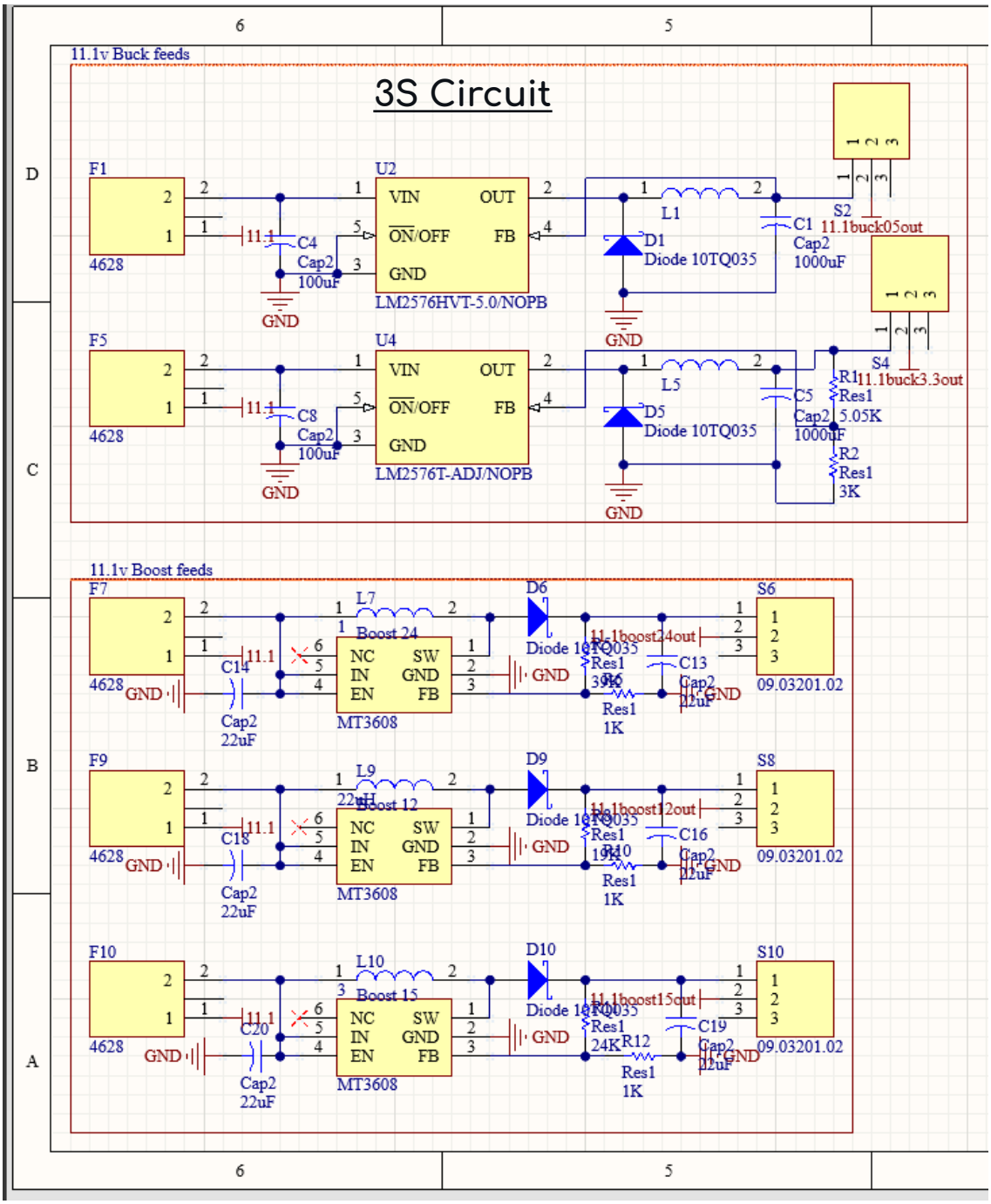

Fig 2.1 - 3S Circuit

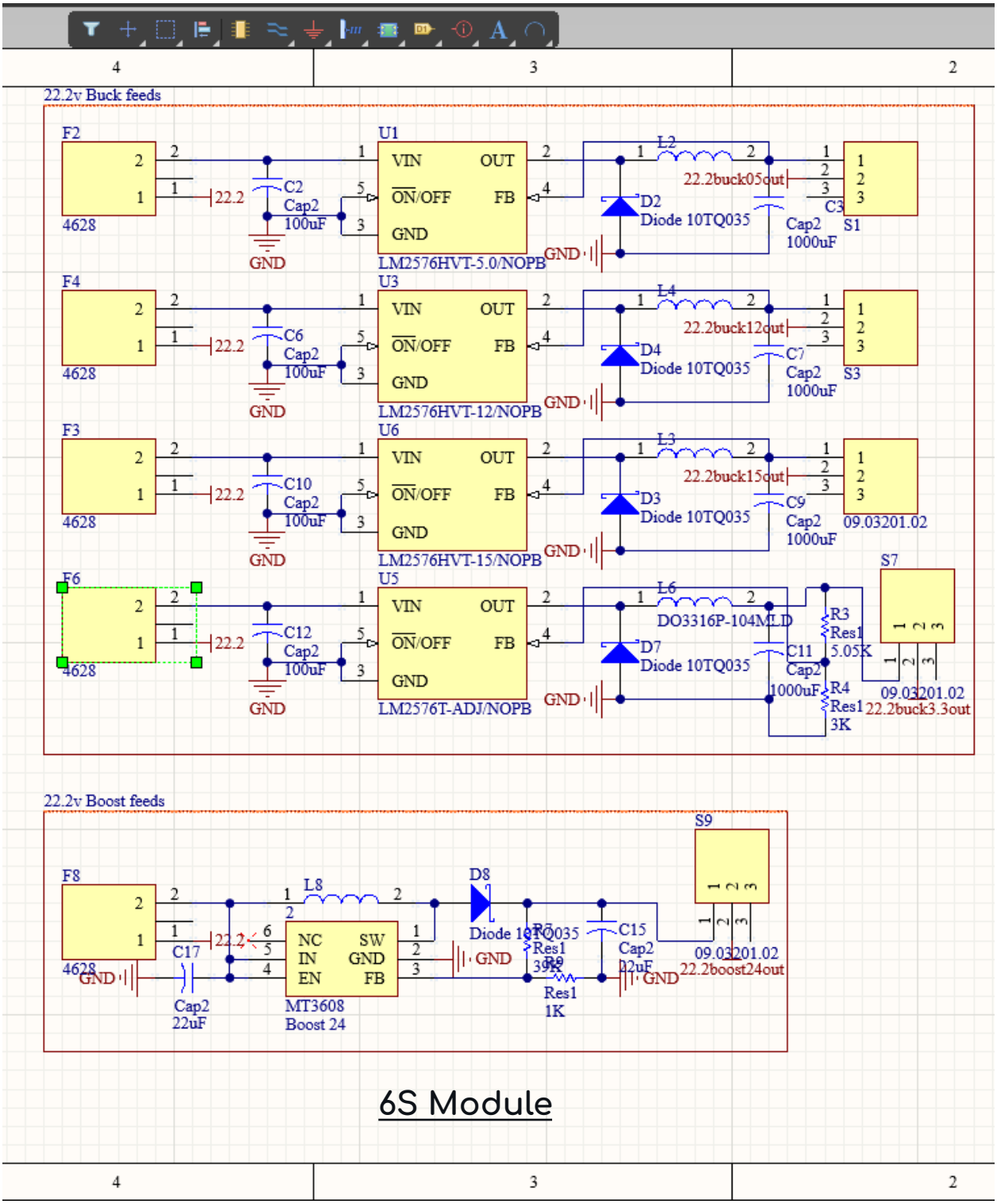

Fig 2.2 - 6S Circuit

The HD Images of the schematic of the whole board including the terminal connection and images of the 3D and 2D views of the printed circuit board will be available on the following github link
https://github.com/BakaToast/toastedpowerboard

Board Fabrication:

All the following information is available as “Board Fabrication Details.pdf” including board assmebly view, fabrication view, realistic view, dimentions of the board, layer stack legend, notes including fabrication and assembly rules for the full production of the PCB.

Some Sample Images are given below-

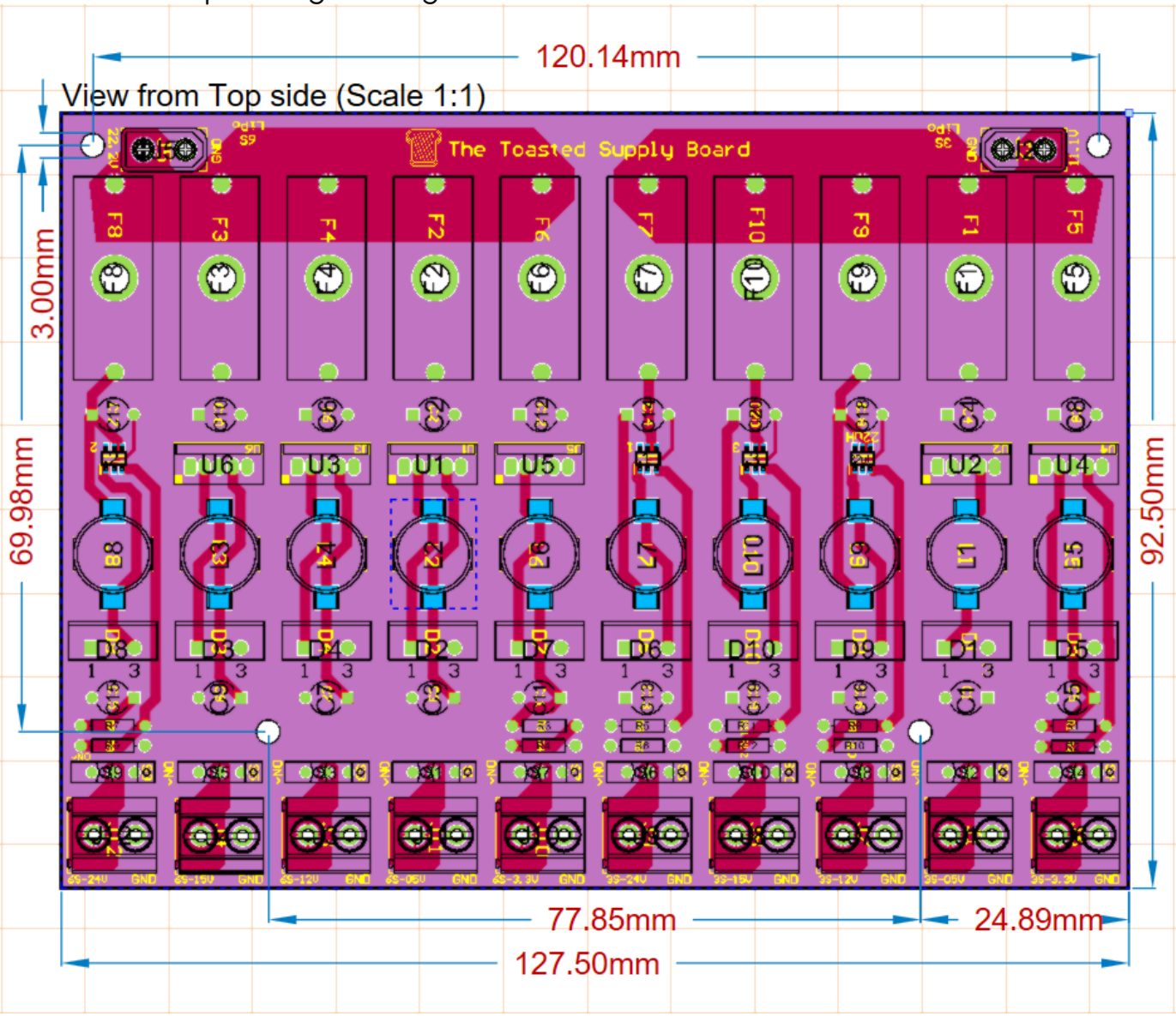

Fig 3.1 - Fabrication View

All the measurements are given on top and are even available on the PDF on my git page (/BakaToast/toastedpowerboard)

The following is the layer stack legend for the PCB which is an automated graphic composite of the project's internal board structure, which also includes the physical layer specifications as defined in the PCB.

Fig 3.2 - Layer Stack Legend

Board Assemble view:

Board Assembly View is an automated graphic composite of the active PCB project's board outline, cutouts, holes, and component graphics with additional notation.

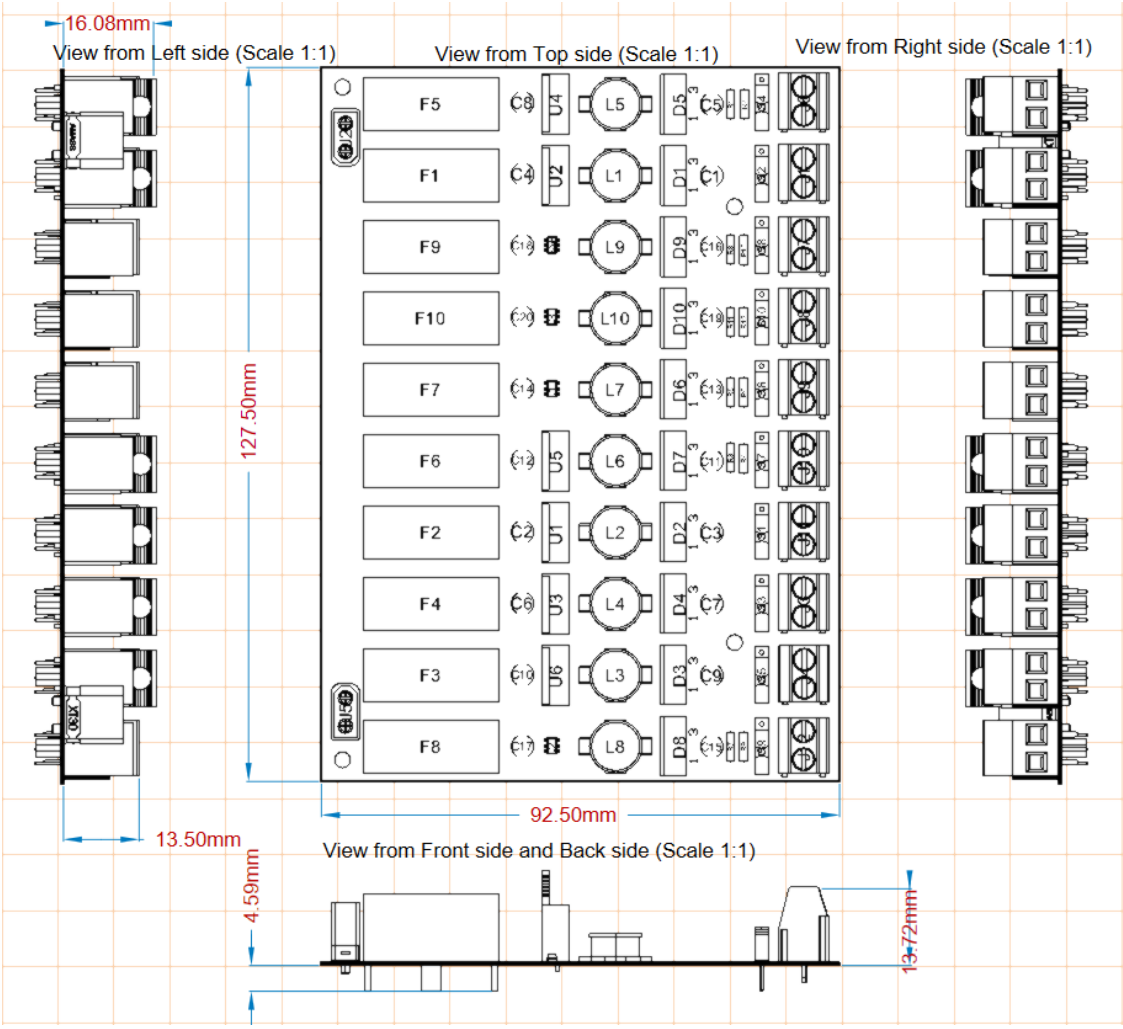

Fig 4.1 - Board Assembly View

Bill-Of-Materials (BOM):

The BOM is a list that describes the parts required to make the board. Usually, there are some basic notes associated with the BOM, for example "R1: do not fit this part". Such notes are also crucial for the manufacturer to know whether to install a certain part or not and that is why i have included it here, for all the notes, please check the pdf as it has detailed notes on fabrication and assembly.

Bill Of Materials

Line #	Designator	Comment	Quantity
1	1, 2, 3, 4	MT3608	4
2	C1, C3, C5, C7, C9, C11	1000uF	6
3	C2, C4, C6, C8, C10, C12	100uF	6
4	C13, C14, C15, C16, C17, C18, C19, C20	22uF	8
5	D1, D2, D3, D4, D5, D6, D7, D8, D9, D10	Diode	10
6	F1, F2, F3, F4, F5, F6, F7, F8, F9, F10	Fuse Holder	10
7	J1, J3, J4, J6, J7, J8, J9, J10, J11, J12	Screw Terminal	10
8	J2, J5	XT30PB	2
9	J6, J10	20020316-G021B01LF	2
10	L1, L2, L3, L4, L5, L6	100uH	6
11	L7, L8, L9, L10	22uH	4
12	R1, R3	5.05K	2
13	R2, R4	3K	2
14	R5, R7	39K	2
15	R6, R9, R10, R12	1K	4
16	R8	19K	1
17	R11	24K	1
18	S1, S2, S3, S4, S5, S6, S7, S8, S9, S10	SPST Switch	10
19	U1, U2	LM2576HVT-5.0/NOPB	2
20	U3	LM2576HVT-12/NOPB	1
21	U4, U5	LM2576T-ADJ/NOPB	2
22	U6	LM2576HVT-15/NOPB	1

PS - The Material in line 9 is not needed

Fig 5.1 - BOM

The following is the price and the link to buy the above components-

1. MT3608 Chip - 15rs - Click Here
2. 1000uF Capacitor - 8rs - Click Here
3. 100uF Capacitor - Click Here
4. 22uF Capacitor - Click Here
5. Diode - Click Here
6. Fuse Holder - Click Here
7. Screw terminal - Click Here
8. XT60 Terminal - Click Here
9. Nil
10. 100uF Inductor - Click Here
11. 22uF Inductor - Click Here
12. 5.05K Resistor - Shop
13. 3K Resistor - Shop
14. 39K Resistor - Shop
15. 1K Resistor - Shop
16. 19K Resistor - Shop
17. 24K Resistor - Shop
18. SPST Switch - Click Here
19. LM2576HVT 05 - Click Here
20. LM2576HVT 12 - Click Here
21. LM2576HVT ADJ - Click Here
22. LM2576HVT 15 - Click Here

Gerber Files:

The Gerbers are used to create the etched base laminate of the PCB. The PCB manufacturer can't make new PCBs without these and hence i have included all the Gerber files in the git link (/BakaToast/toastedpowerboard)

lg thats all for the documentation, if you have any issues regarding the board please contact the following email - bakatoast@beinghyper.in

buhbyeeee!