LM5146 BUCK CONVERTER PCB DESIGN

REPORT

ELECTRONIC PACKAGING, WS2024

Mahmoud Esameldin Osman

14.01.2025

FH JOANNEUM, Electronic Engineering Masters

Task Description:

This laboratory task involves the complete design of a prototype Printed Circuit Board (PCB) of a DC-DC Buck converter evaluation module based on the LM5146-Q1 controller IC. The task includes schematic and layout design. The specifications of the design are shown in table 1.

Table 1: Specifications

Specification	Description
Input voltage range	15V – 80V
Typical input voltage	48V
Output voltage	12V
Maximum output current	6A
Efficiency	96%
Switching frequency	400KHz

Calculations:

To correctly implement the schematic of my design, I used the application circuit 2 schematic (figure 1) from the datasheet [1] as a guide, and adjusted the values of different components as recommended. MATLAB is used to find some of the calculation results.

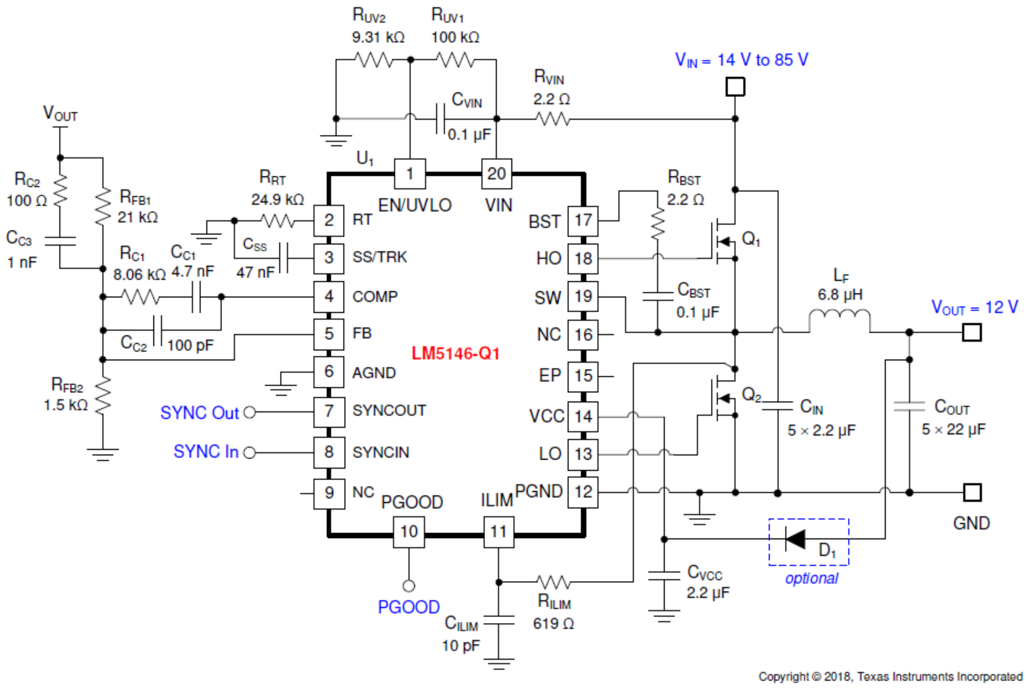

Figure 1: Application Circuit 2 from datasheet [1]

Inductor:

$$L_F = \frac{V_{OUT}}{V_{IN}} \cdot \left(\frac{V_{IN} - V_{OUT}}{\Delta I_L \cdot F_{SW}} \right)$$

Equation 1: Inductance formula [1]

```
VIN = 48;
VOUT = 12;
fsw = 400e3;
IMAX = 6;
dIL = 0.2 * IMAX;
LF = (VOUT/VIN)*((VIN-VOUT)/(fsw*dIL))
```

```
LF = 1.8750e-05
```

Therefore, the value of LF is 18uH.

Output Capacitor (C_{OUT}):

The output capacitor value is given by the following equation:

$$C_{OUT} \geq \frac{\Delta I_L}{8 \cdot F_{SW} \sqrt{\Delta V_{OUT}{}^2 - (R_{ESR} \cdot \Delta I_L)^2}}$$

Equation 2: Output Capacitor formula [1]

```
dVOUT =  0.01 * VOUT; % 1 % Tolerance
COUT = (dIL)/(8 * fsw * sqrt((dVOUT)^2));
```

```
COUT = 3.1250e-06
```

Therefore, the value of COUT is 3.12uF.

Based on the datasheet (pg.: 27), the ESR of ceramic capacitor is extremely low, so I neglected it in the calculation of output capacitor. Moreover, ignoring the ESR term in Equation 2 gives a quick estimation of the minimum ceramic capacitance necessary to meet the output ripple specification which is 2% of the maximum output current in my case. It is recommended to choose a larger output capacitance than 3.12uF, so I chose 3*22uF.

Input Capacitance (CIN):

$$C_{IN} \geq \frac{D \cdot (1-D) \cdot I_{OUT}}{F_{SW} \cdot (\Delta V_{IN} - R_{ESR} \cdot I_{OUT})}$$

Equation 3: Input Capacitor formula [1]

```
D = 0.5 % datasheet page: 29
```

```
D = 0.5000
```

```
dVIN = 0.01 * VIN; % 1 % Tolerance
CIN = (D *(1-D)* IMAX)/(fsw * (dVIN))
```

```
CIN = 7.8125e-06
```

I used 4*4.7uF in addition to 2*10nF and 2*100nF decoupling capacitors in parallel. My chosen capacitors are X7R rated to provide low impedance and high RMS current rating over a wide range of temperature as recommended by TI.

Other components:

For the other components, I used the Quickstart tool design calculator [2] to choose the values of R_{UV1} , R_{UV2}.

The frequency adjust resistor R_{RT} I chose is 24.9Kohm for the switching frequency of 400kHz based on table 8-1 from the datasheet shown below in figure 2.

Adjust the LM5146-Q1 free-running switching frequency by using a resistor from the RT pin to AGND. The switching frequency range is from 100 kHz to 1 MHz. The frequency set resistance, R_{RT}, is governed by Equation 3. E96 standard-value resistors for common switching frequencies are given in Table 8-1.

$$R_{RT}[k\Omega] = \frac{10^4}{F_{SW}[kHz]} \quad (3)$$

Table 8-1. Frequency Set Resistors

SWITCHING FREQUENCY (kHz)	FREQUENCY SET RESISTANCE (kΩ)
100	100
200	49.9
250	40.2
300	33.2
400	24.9
500	20
750	13.3
1000	10

Figure 2: Screenshot of Frequency set resistors values from the datasheet.

Placing a 2-ohm to 10-ohm resistor in series with the boot capacitor slows down the high-side MOSFET turn-on transition, serving to reduce the voltage ringing and peak amplitude at the SW node at the expense of increased MOSFET turn on power loss(pg. 48)[1], so I added a 2.2-ohm resistor.

Schematic design:

The figure below shows my schematic diagram. I used symbols as per the EN-60617 standard for circuit symbols [3].

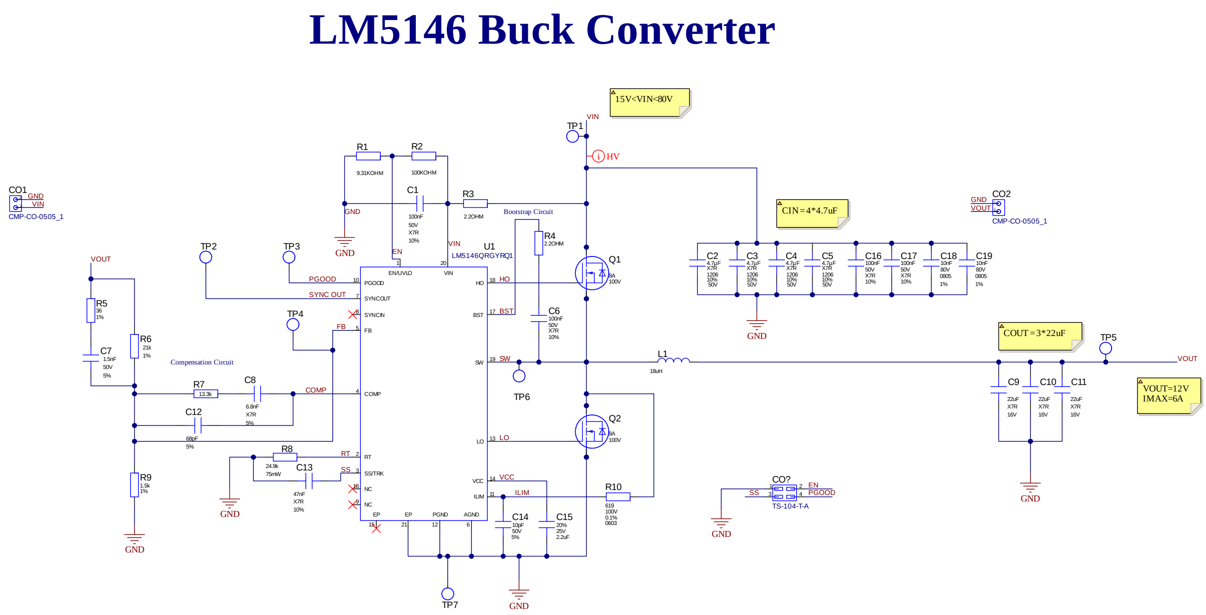

Figure 3: Schematic diagram

I have added 7 test points to take measurements of different signals and verify the functionality. Table 2 shows the test points and their net labels.

Table 2: Test points and their Net labels

Test point	Net label
TP1	VIN
TP2	SYNC OUT
TP3	PGOOD
TP4	FB
TP5	VOUT
TP6	SW
TP7	GND

Layout design:

My layout design is inspired by the layout example given in the datasheet of the LM5146-Q1, where I used copper filled polygon areas to route the components as much as possible. I mainly focused on connecting the power components with the polygons and then route the signals with tracks as shown in figure 4.

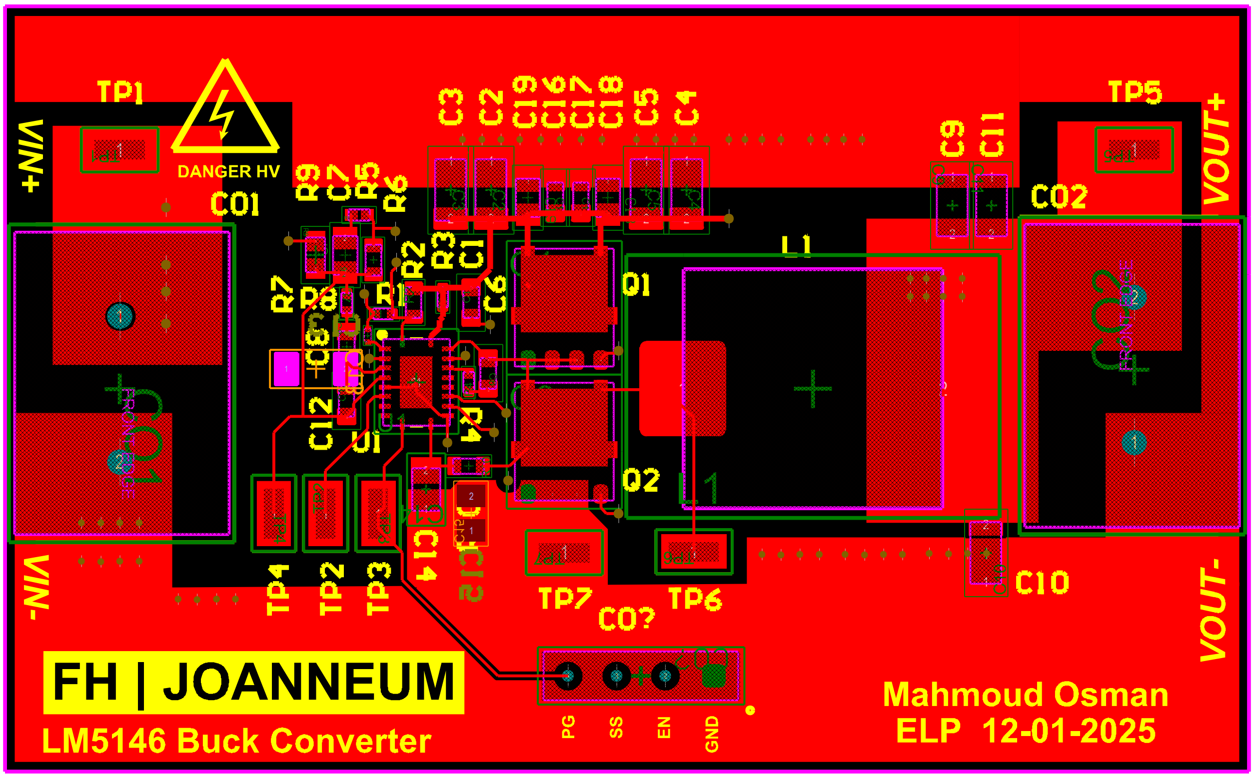

Figure 4: PCB 2D View with components- Top

In layout design rules, I chose the minimum clearance of 0.6mm for HV nets and 0.15mm according to the IPC2221 B conductor spacing requirements. [4]

Figure 5 shows my board layer stack up. It has an 8-layer stackup due to lower costs of production by eurocircuits.

#	Name	Material	Type	Weight	Thickness	Dk	Df	Orientation
	Top Overlay		Overlay					
	Top Solder	Solder Resist	Solder Mask		0.01mm	3.5		
1	Top Layer		Signal	1oz	0.035mm			Top
	Dielectric 6	PP-006	Prepreg		0.07112mm	4.1	0.02	
	Dielectric 7	PP-006	Prepreg		0.07112mm	4.1	0.02	
2	Layer 2	CF-004	Signal	1oz	0.035mm			Not allowed
	Dielectric 10	Core-009	Core		0.1016mm	4.5	0.02	
3	Layer 3	CF-004	Signal	1oz	0.035mm			Not allowed
	Dielectric 2	PP-006	Prepreg		0.07112mm	4.1	0.02	
	Dielectric 3	PP-006	Prepreg		0.07112mm	4.1	0.02	
4	Layer 4	CF-004	Signal	1oz	0.035mm			Not allowed
	Dielectric 1	FR-4	Core		0.9mm	4.2		
5	Layer 5	CF-004	Signal	1oz	0.035mm			Not allowed
	Dielectric 4	PP-006	Prepreg		0.07112mm	4.1	0.02	
	Dielectric 5	PP-006	Prepreg		0.07112mm	4.1	0.02	
6	Layer 6	CF-004	Signal	1oz	0.035mm			Not allowed
	Dielectric 11	Core-009	Core		0.1016mm	4.5	0.02	
7	Layer 7	CF-004	Signal	1oz	0.035mm			Not allowed
	Dielectric 8	PP-006	Prepreg		0.07112mm	4.1	0.02	
	Dielectric 9	PP-006	Prepreg		0.07112mm	4.1	0.02	
8	Bottom Layer		Signal	1oz	0.035mm			Bottom
	Bottom Solder	Solder Resist	Solder Mask		0.01mm	3.5		
	Bottom Overlay		Overlay					

Figure 5 Board layer stackup

Table 3 shows the polygon pours I implemented in each layer to ensure proper thermal management and current return paths. Hence I made the VIN and VOUT planes between ground layers.

Table 3: Polygon pours

Layer	Polygon Pour Net(s)
Layer 1	Signals and power (GND, VIN, VOUT)
Layer 2	Power (GND)
Layer 3	Power (VIN)
Layer 4	Power (GND)
Layer 5	Power (VOUT)
Layer 6	Power (GND)
Layer 7	Power (GND)
Layer 8	Signals and power (GND)

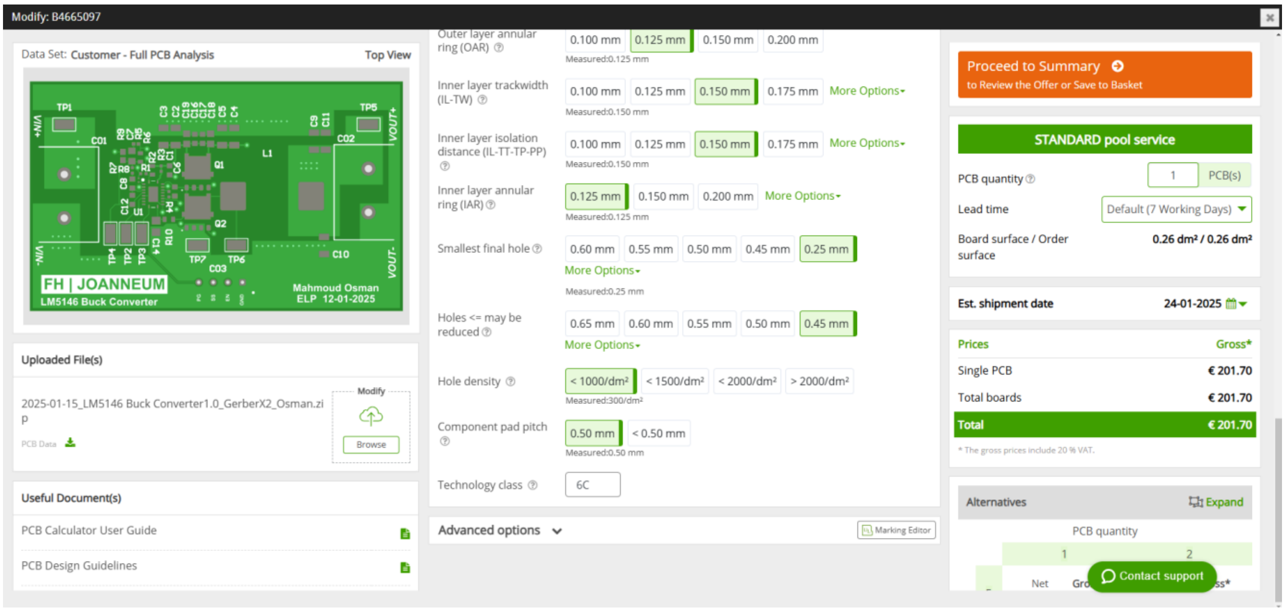

Figure 6: Price analysis

Figure 6 shows the price analysis of my board from eurocircuits PCB manufacturer[5]. It estimates the cost of my board to be €201.70 for one board of dimensions of 65mm X 40mm.

The figures below show the polygon pours of all the layers of my pcb followed by the 3D top, bottom and custom views.

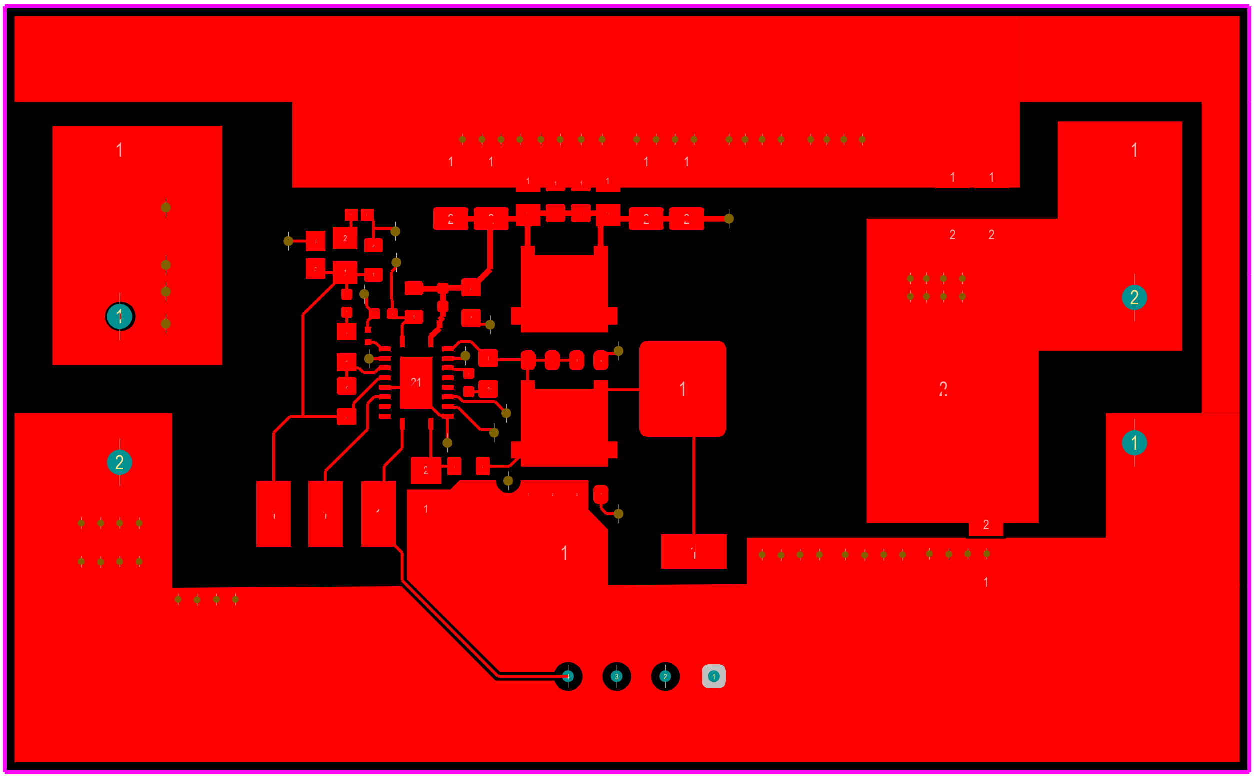

Figure 7: Layer 1 polygon pour

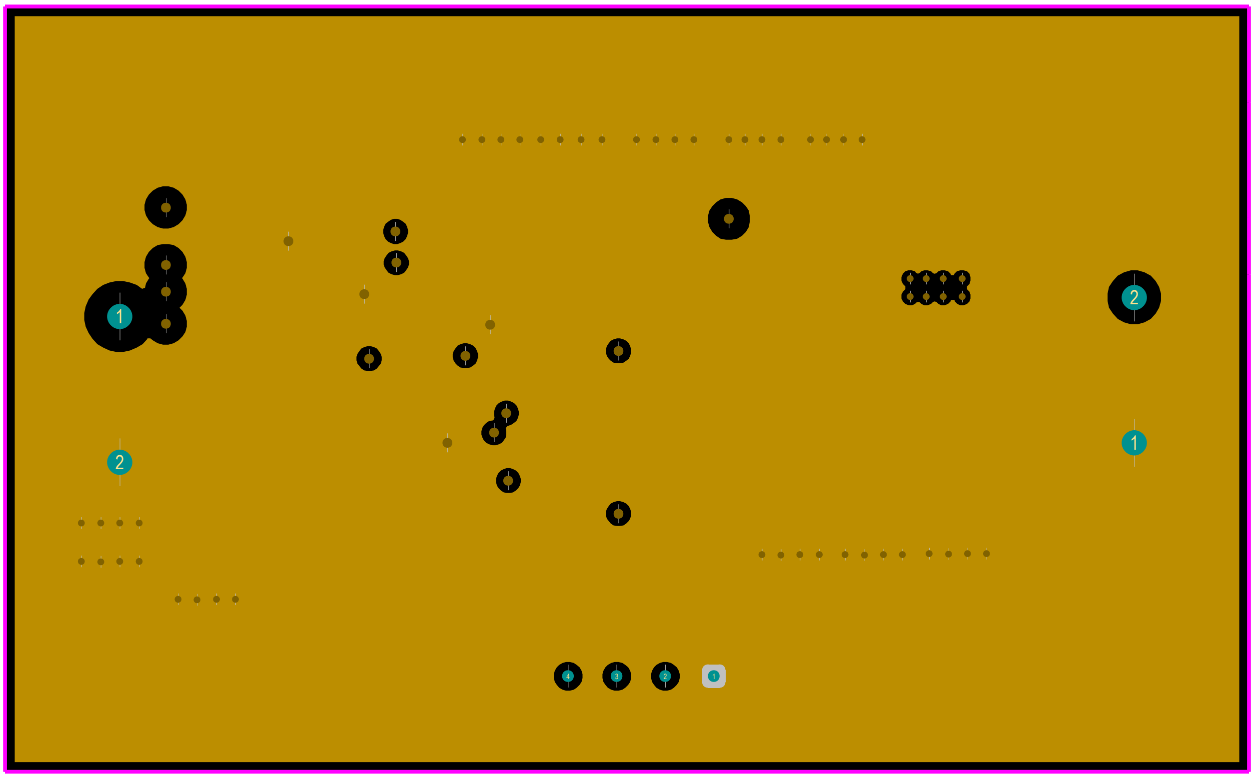

Figure 8: Layer 2 polygon pour

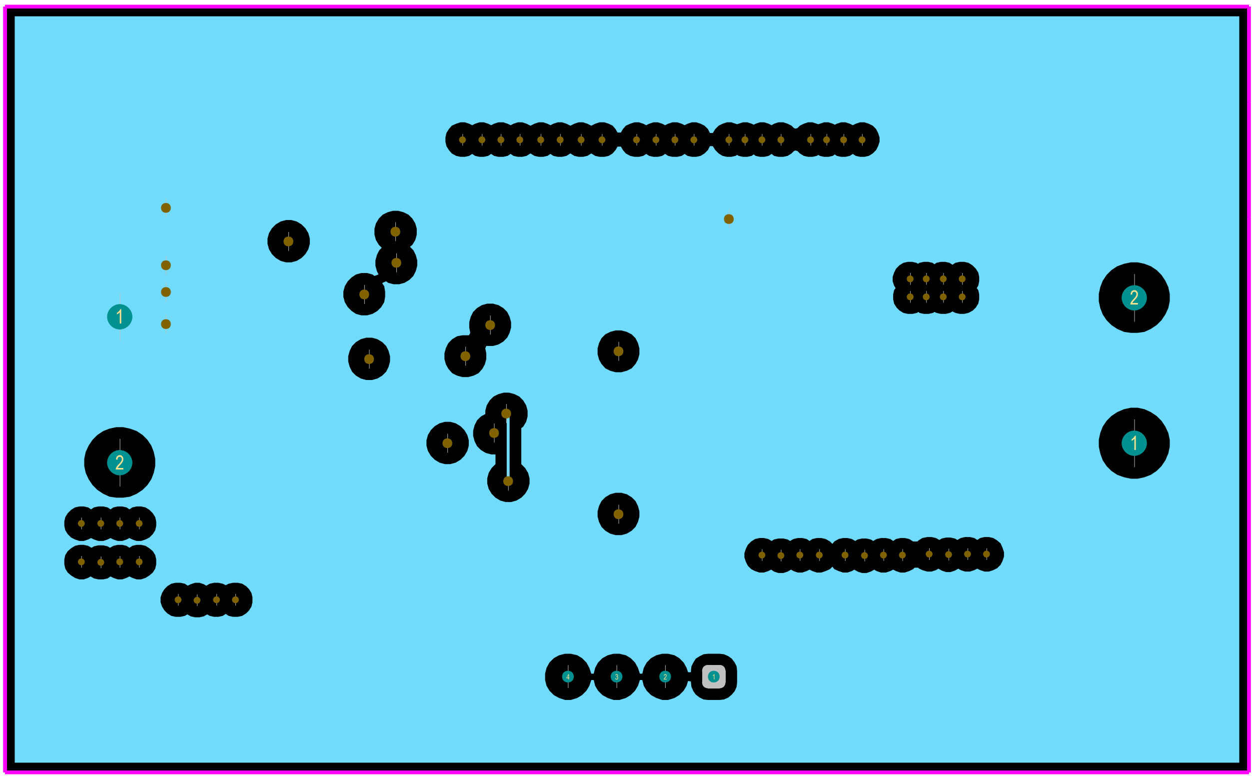

Figure 9: Layer 3 polygon pour

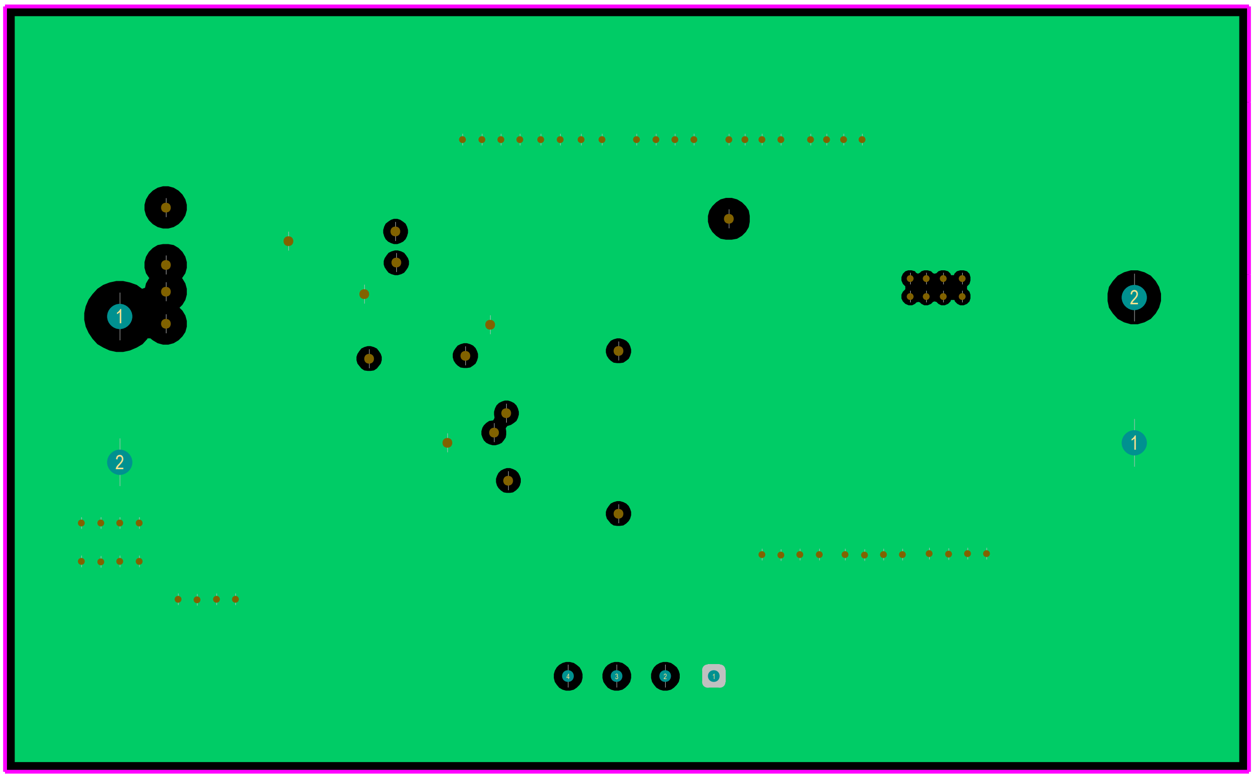

Figure 10: Layer 4 polygon pour

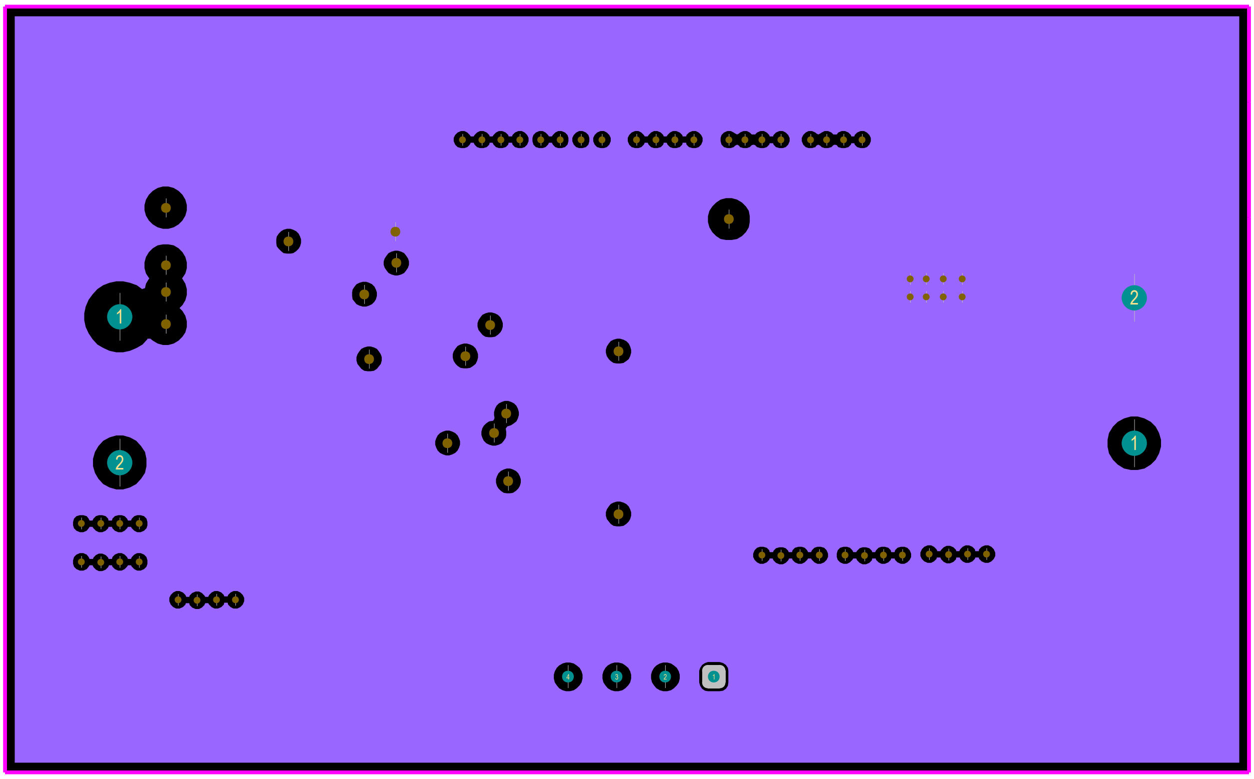

Figure 11: Layer 5 polygon pour

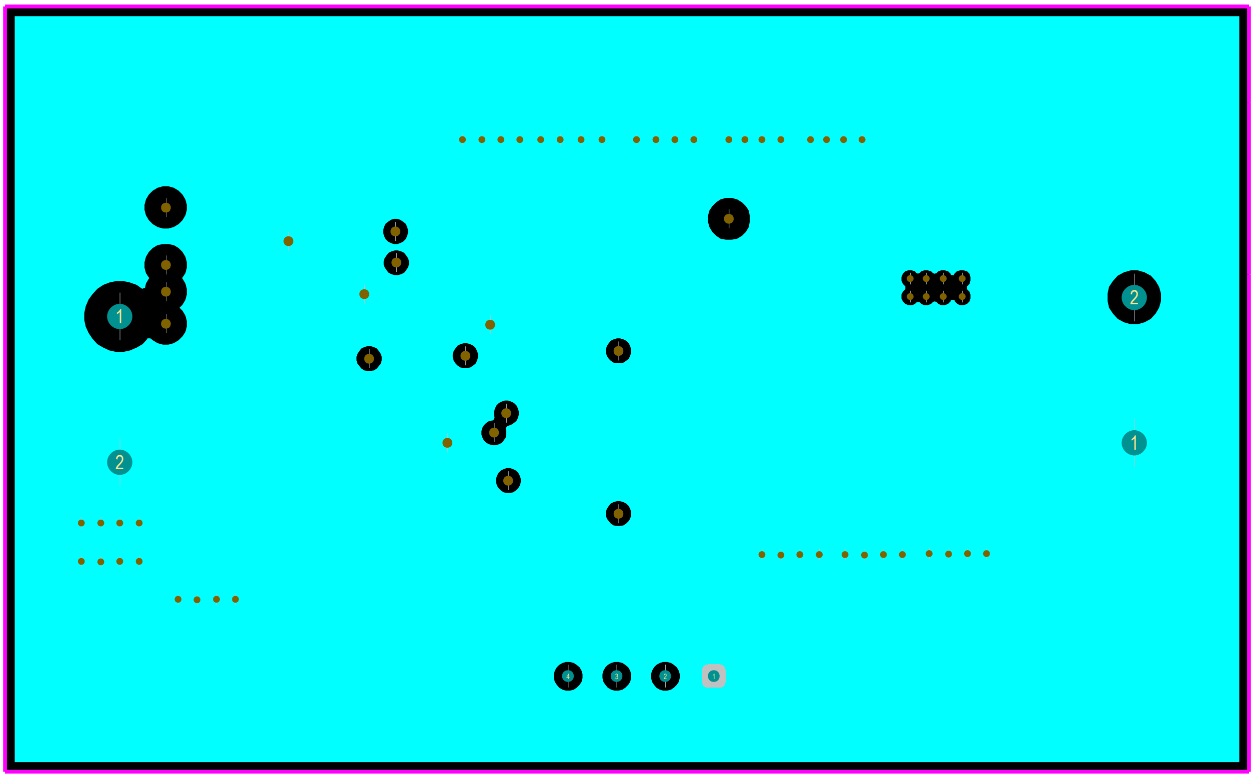

Figure 12: Layer 6 polygon pour

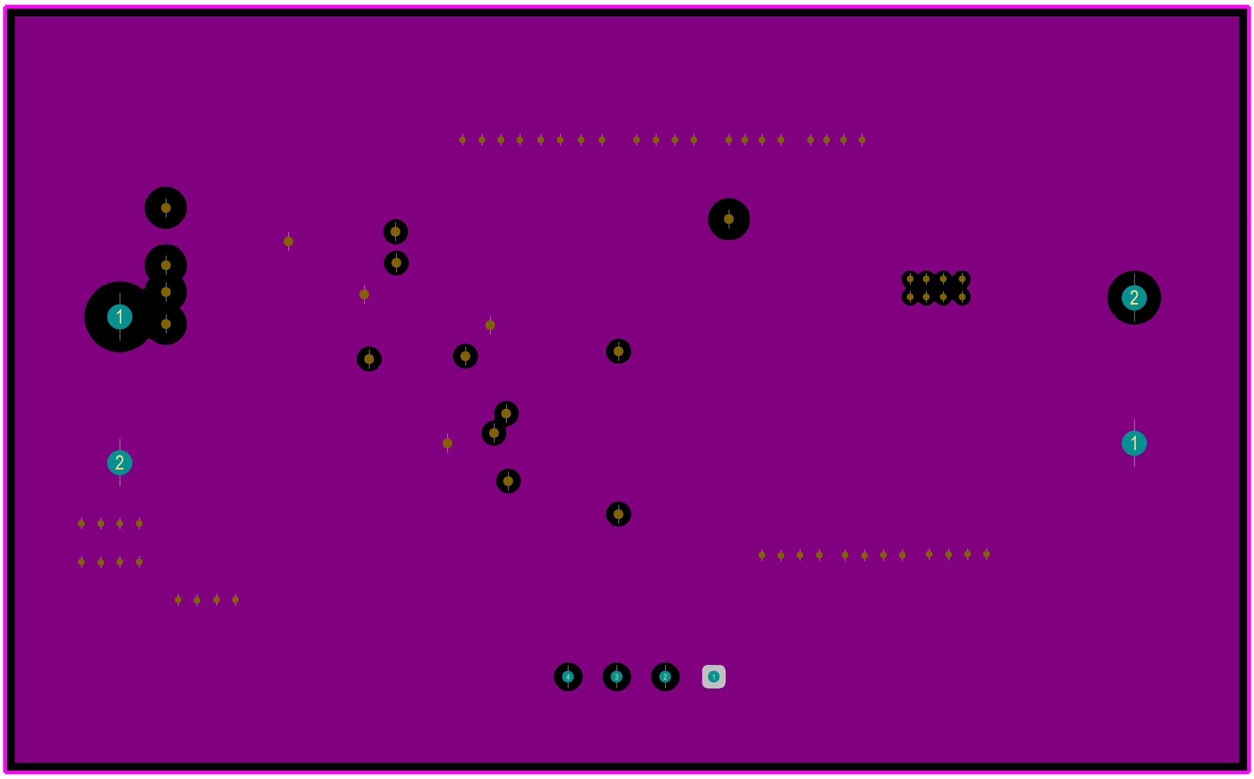

Figure 13: Layer 7 polygon pour

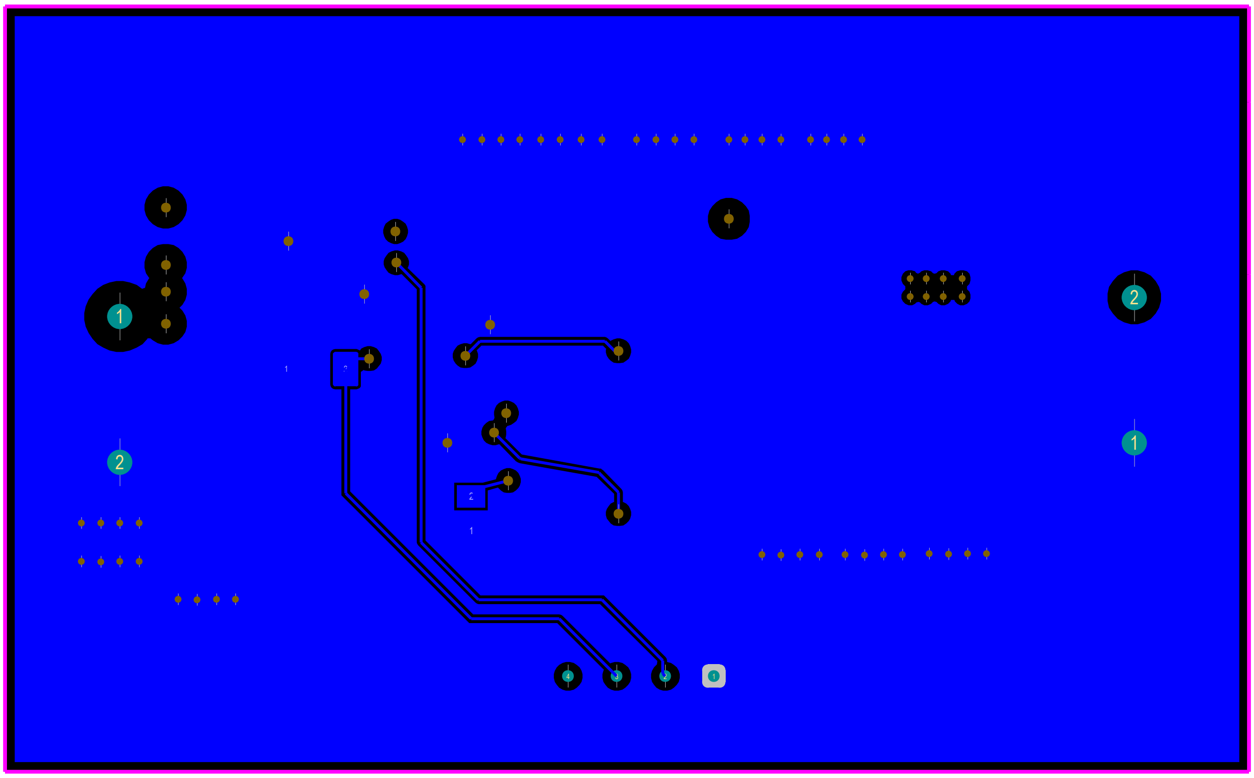

Figure 14: Layer 8 polygon pour

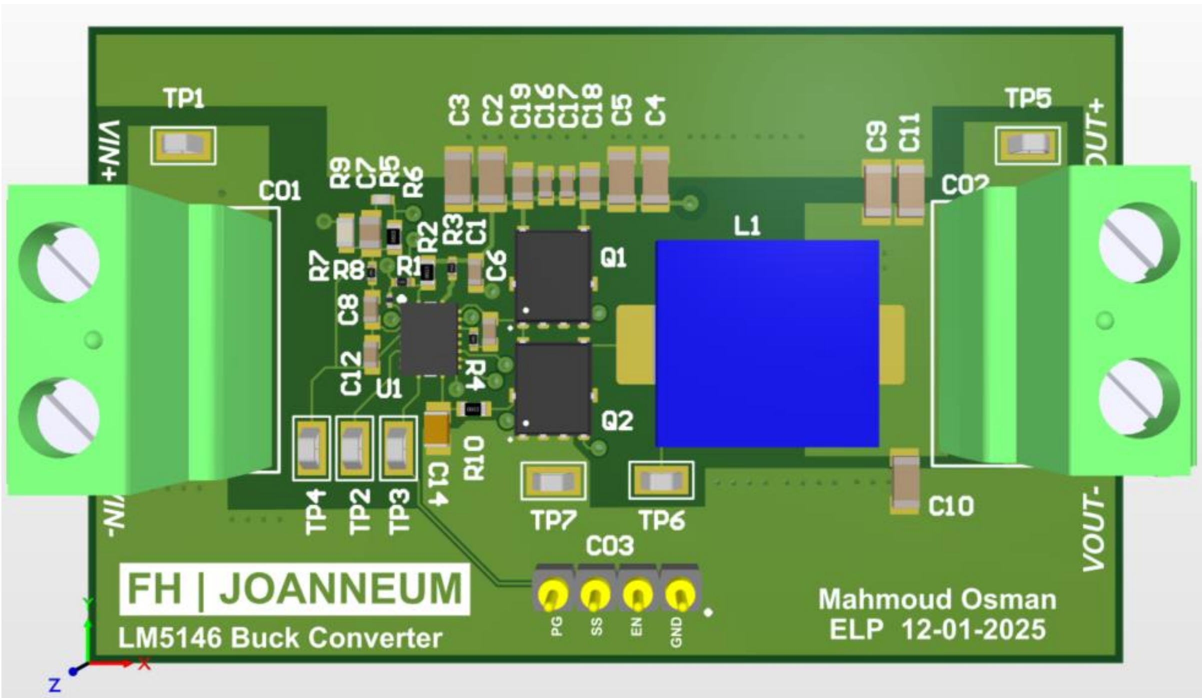

Figure 15: 3D Top View

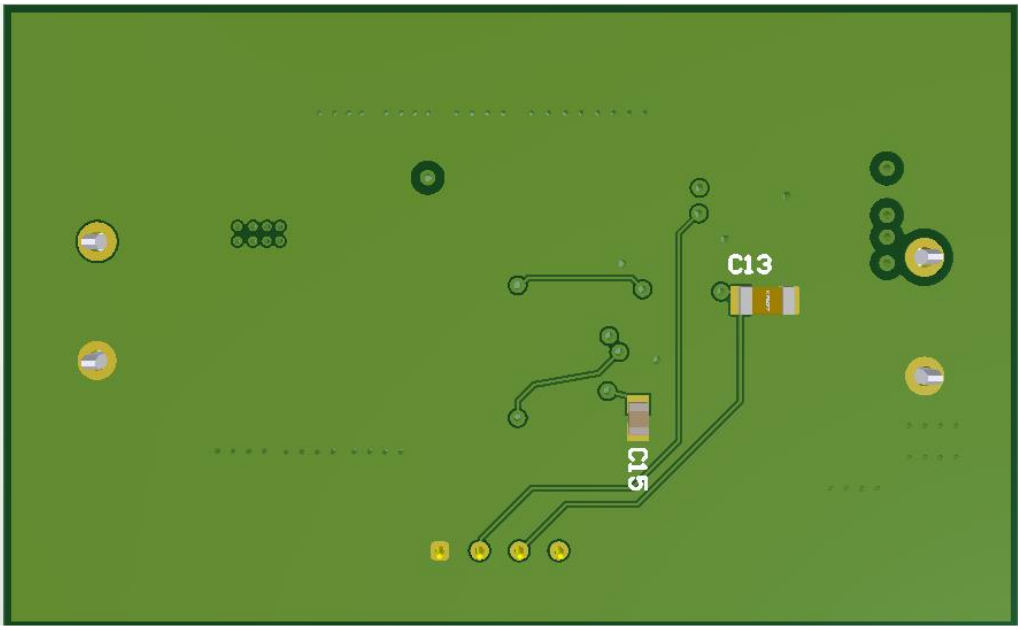

Figure 16: 3D Bottom View

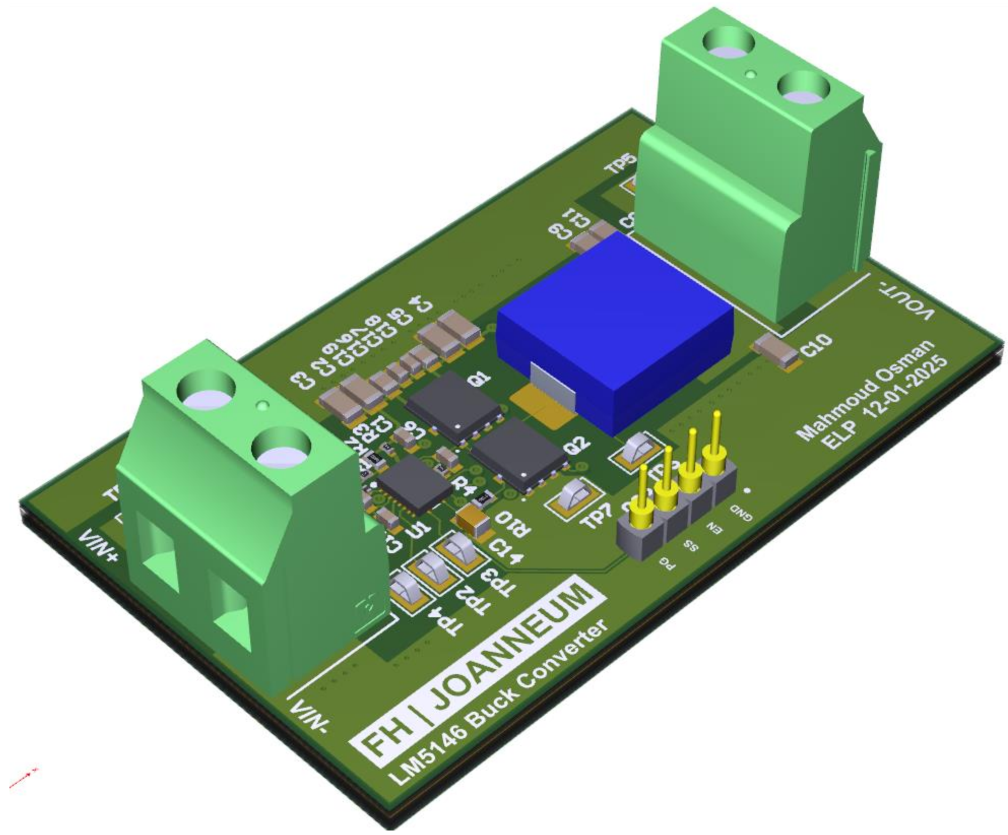

Figure 17: 3D Custom View

References

[1] ‘lm5146.pdf’. Accessed: Jan. 14, 2025. [Online]. Available: https://www.ti.com/lit/ds/symlink/lm5146.pdf

[2] ‘LM5146DESIGN-CALC Calculation tool | TI.com’. Accessed: Jan. 14, 2025. [Online]. Available: https://www.ti.com/tool/LM5146DESIGN-CALC

[3] ‘Liste der Schaltzeichen (Elektrik/Elektronik)’, *Wikipedia*. Apr. 03, 2024. Accessed: Jan. 14, 2025. [Online]. Available: https://de.wikipedia.org/w/index.php?title=Liste_der_Schaltzeichen_(Elektrik/Elektronik)&oldid=243697134

[4] ‘Using an IPC-2221 PCB Clearance Calculator for High Voltage Design’, Altium. Accessed: Dec. 29, 2024. [Online]. Available: https://resources.altium.com/p/using-an-ipc-2221-calculator-for-high-voltage-design

[5] ‘PCB & PCBA prototypes and small series “right first time”’, Eurocircuits. Accessed: Jan. 15, 2025. [Online]. Available: https://34.252.181.249/