Faculty of Engineering Technology, Campus Group T

Power Electronics Project

Nixie tube supply

Andrei Trafimau and Javad Nematli
21-12-2022

Contents

Chapter 1

1.1 Design Specification

Input:
9V
Output:
170V, up to 10 mA

Chapter 2

Design Procedure

The MAX1771CPA+ step-up switching controller from MAXIM INTEGRATED was used for this application.

2.1 MAX1771CPA+

The MAX1771 is a step-up switching controller that has a current-limited PFM (pulse-frequency-modulation) that gives it the benefits of PWM (pulse-width-modulation) converters while using less than 110µA of supply current. This value is very low, as it usually ranges from 2mA to 10mA for PWM converters.

The MAX1771 has a high switching frequency (up to 300 kHz). Moreover, it accepts input voltages between 2V and 16.5V.

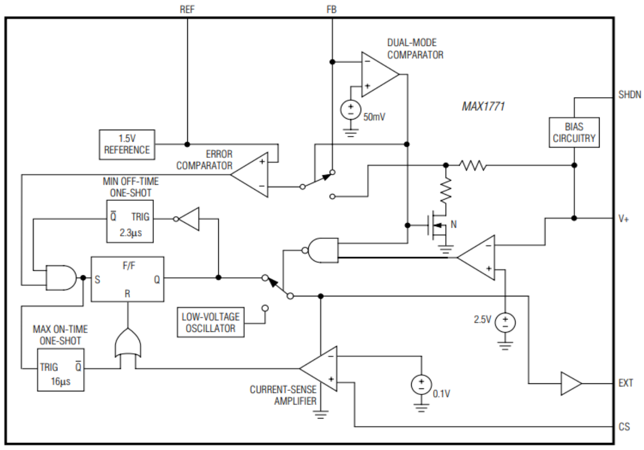


Figure 2.1: Block diagram of the MAX1771CPA+

This controller reduces noise using a single 100mV current limit threshold under all load conditions. Also, it drives an external N-channel MOSFET switch, allowing it to power loads up to 24W.

2.2 Schematic

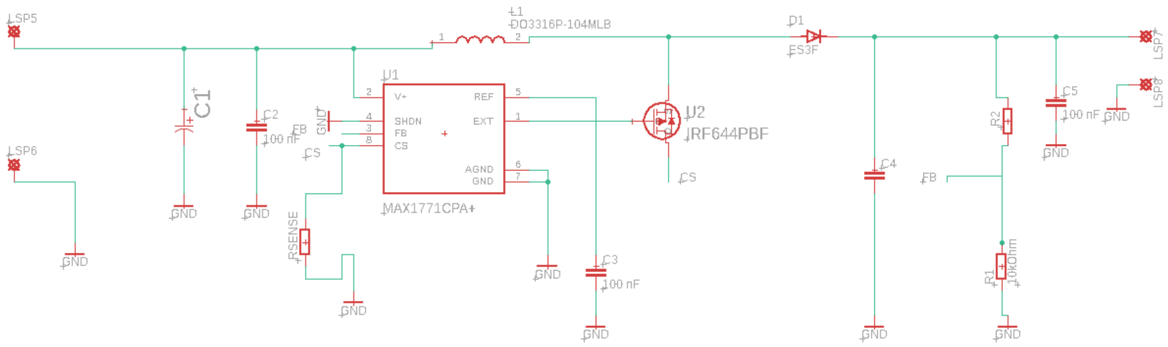


Figure 2.2: Schematic used for this project

In figure 2.2 it is possible to observe the schematic that was used to create this project. It is a boost converter. Here are the naming and the conventions that we have used for the components:

- Input capacitance: C1 and C2
- Sense resistance: Rsense
- Inductor: L1
- Reference capacitance: C3
- IC: U1
- MOSFET: U2
- Diode: D1
- Output capacitance: C4 and C5
- Feedback resistors: R1 and R2

Feedback resistors form a voltage divider circuit and are used to set up the output voltage. With the sense resistance, we control how much current will our circuit supply. C4 is the output filter capacitor. C5 is used with the same purpose as C4, but it is an optional one and it is not required by the manufacturer's guidelines.

2.3 Calculation of the components

2.3.1 Setting the output voltage

The MAX1771CPA+ can be configured for a wide range of outputs (starting from 3V and up to more than 200V). To select the output voltage a voltage divider network is used which consists of two feedback resistors. The manufacturer advises selecting R1 in the 10 KOhm to 500 KOhm range. We chose to work with 10 KOhm. After, we can find R2 using this formula:

$$R2 = (R1) * \left(\frac{Vout}{Vref} - 1\right)$$

Our Design Specification was to make a 170V output; therefore, Vout equals 170V. Vref is equal to 1.5V. So, if we plug in the formula of all values – we find that R2 equals 1.1 MOhm.

Important note: R2 has almost the full output voltage across it; therefore we chose a power resistor. R1 was chosen to be a generic 10 KOhm resistor from the 0805 SMD package.

2.3.2 Determining Rsense

Rsense is important to choose carefully because it will determine our output current. To choose Rsense correctly, we have used the datasheet of the MAX1771CPA+ and the guidelines from the manufacturer. According to our design specifications the output current cannot exceed 10 mA; therefore, Rsense was chosen to equal 100 mOhm.

2.3.3 Determining the Inductor

The manufacturer suggests using inductor values the in range of 10 µH to 300 µH. However, if the inductor value is too low, the current will ramp up to a high level before the current-limit comparator can turn off the switch. This is something that we want to avoid because our output current should not exceed 10 mA.

The value of an inductor can be calculated according to this formula:

$$L \geq \frac{Vin(maximal) * 2\mu s}{Ilim}$$

By using the above equation we have found that the inductance value should be greater or equal to 1.8 µH. This is a very low value because we are required to operate with low input voltage.

So, to choose the inductance value we used several statements from the manufacturer's website, namely:

1. If we choose a smaller inductance value – then it will allow the coil current to ramp up to higher levels before the switch turns off. This will increase the ripple at light loads.
2. If we choose a higher inductance value, it will increase the start-up time slightly.

Since we want to avoid a big ripple in our application, we decided to find a middle ground and chose the inductor with the inductance value of 100 µH with low DC resistance and high saturation current (1.2 A) to avoid any possible saturation.

2.3.4 Power transistor selection

The power MOSFET is an N-type one, as recommended in the datasheet of a step-up converter. Moreover, three important parameters need to be considered, which are the total gate charge (Q_g), on-resistance ($R_{ds(on)}$), and reverse transfer capacitance. The datasheet of MAX1771 says that the chosen N-type switch should have low $R_{ds(on)}$. The IRF644 MOSFET fulfills this requirement by having low $R_{ds(on)}$ (0.28 Ohm at V_{gs} = 10V). The total gate charge of this MOSFET is 68nC max, which is considered low.

According to the datasheet of MAX1771, the chosen N-FET's most significant losses are switching and I^2R losses. To minimize these losses, the chosen MOSFET has a low reverse transfer capacitance (85 pF).

At the start-up, the maximum switching frequency of MAX1771 can reach 500 kHz (even though the maximum allowed switching frequency is 300 kHz). For this reason, the maximum current required to charge the N-FET's gate is:

$$Igate(max) = f(max) \text{ x } Qg(typ) = (500\ kHz)\ (17\ nC) = 8.5\ mA$$

Where $Q_{g(typ)}$ is the typical gate charge of the N-FET from its datasheet.

Moreover, according to the datasheet of MAX1771, the bypass capacitor on V+ must furnish the gate charge instantly to limit the amount of voltage drop, (<200mV for the best performance).

For the chosen bypass capacitor C2 of 0.1 µF, the voltage drop at this point is:

$$\Delta V+ = \frac{Qg}{C2} = \frac{17nC}{0.1\mu F} = 170\ mV$$

2.3.5 Diode selection

To maintain MAX1771's high switching frequency we need to choose a high-speed rectifier. The main requirements for the diode selection are that its average current rating should exceed the peak current limit set by Rsense and that its break-down voltage exceeds Vout. Having this in mind we chose an ultrafast rectifier (ES3F).

ES3F features a very small reverse recovery time – 35 ns. Moreover, the average forward current is 3A, meaning we will never have problems with exceeding its average current rating and the maximum break-down voltage is 300V. This diode not only guarantees safety but also allows room for experiments with higher voltage.

2.3.6 Capacitor selection

In our design, we have 5 capacitors, so in this section, we will explain how those were selected.

2.3.6.1 Output Filter Capacitor

We have two output filter capacitors in our design. C4 is required by the manufacturer and C5 is optional.

The primary criterion for selecting the output filter capacitor (C4) is low effective series resistance (ESR). Low ESR is needed because it affects the efficiency of the circuit. So, to achieve the best performance, it is better to use low ESR capacitors. The product of the peak inductor current and the output filter capacitor's ESR determines the amplitude of the ripple seen on the output voltage.

Moreover, the output filter capacitor should be rated at more than the output DC voltage to prevent any kind of damage to it.

Our selected capacitor is the aluminum electrolytic one and it features a very low ESR – less than 3Ohm, it is rated to work at 250V DC at maximum and its capacitance value is 10 µF.

Furthermore, we added the optional 100 nF capacitor (C5). The main requirement is that it should be rated at more than the output DC voltage. So we chose a ceramic capacitor that is rated to work 200V max DC.

2.3.6.2 Input Bypass Capacitors

The input bypass capacitor (C1) reduces peak currents drawn from the voltage source and also reduces noise at the voltage source caused by the switching action of the MAX1771. The manufacturer recommends using a low ESR capacitor.

We decided to choose a 100 µF Tantalum capacitor with a 20 V DC rating. Moreover, it features a low ESR – 35 mOhm.

The second capacitor (C2) is a generic 100 nF ceramic capacitor. There is nothing special with this one.

2.3.6.3 Reference Capacitor

According to the manufacturer's guidelines – C3 should be also a 100 nF ceramic capacitor. Therefore, we decided to use the same capacitor as we use for C2. One thing is that the REF pin of the IC can source up to 100 µA of current for external loads, but there will not be any problems if we use the same capacitor as C2.

2.4 PCB layout

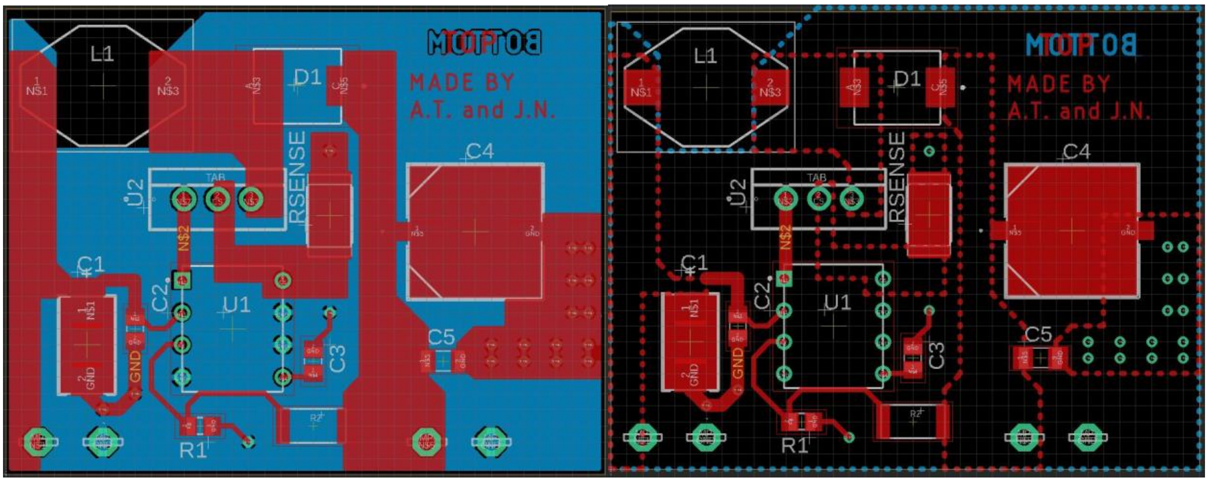


Figure 2.3. PCB layout.

Figure 2.4 shows the PCB layout. This is a 2-layer PCB. The guidelines from the IC manufacturer were followed during the design phase of the PCB, for instance, capacitors C1 and C2 should be placed very close to the IC.

The trace between the MAX 1771 and the FET needs to be short and FAT, i.e. low resistance and inductance. This allows the gate charge to be given to the FET in the shortest possible time; the speed at the FET switches directly affects efficiency.

The FB pin is very sensitive to stray EMI - the voltage feedback network should be kept away from high current paths and the connection to the FB pin should be as short as possible.

The lack of a ground plane under the inductor prevents energy loss due to induced currents in the ground plane.

Red polygons are current paths. Using polygons instead of routes allowed us not to worry about currents going to and from the inductor; therefore, it is safer for our board. Moreover, there are holes in the red polygons, for example, on the right. This is not the current plane, this is a polygon on the top layer that connects the grounds of capacitors and with the help of via's connects the top plane with the bottom one. By using this technique we can dissipate more heat on the PCB, so our capacitors do not get hot.

Chapter 3

Design Verification

3.1 Setup

To test and validate our circuit we assembled the PCB and carried out tests on fixed loads and a nixie tube. All measurements were done in the lab room of campus “Group T” in Leuven.

Equipment that was used for the tests:

- Power supply – Gw INSTEK GPS-3303
- Digital Oscilloscope Tektronix TBS 1052B-EDU. Any scope images in this report are from this Tektronix Digital Oscilloscope.
- Power resistors with values of 10 KOhm, 6.8 KOhm, and 2.2 KOhm
- Probe with 10x attenuation from our electronic kit.

3.2 Working

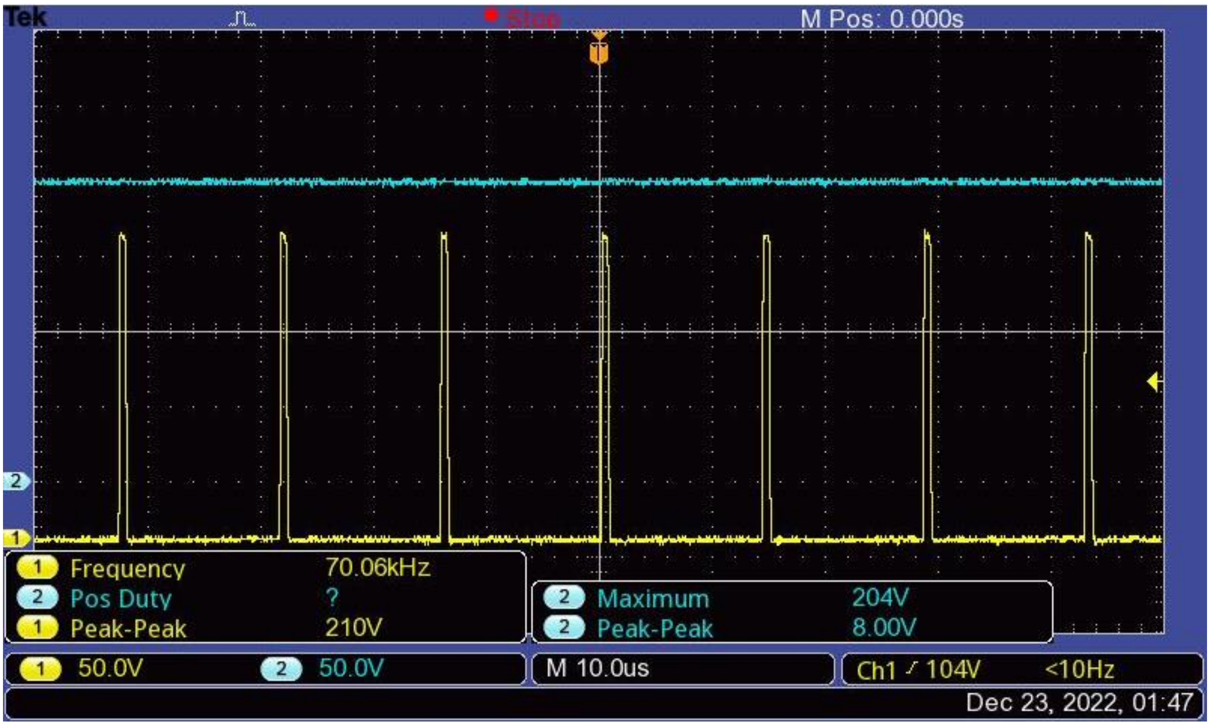


Figure 3.2a: Output voltage (blue line) and switching voltage (yellow line) with 16.8 KOhm load

In figure 3.2a it is possible to observe a working setup. We have connected a 16.8 KOhm load to simulate the required output current that was proposed in the design specification part.

However, as can be seen from the figure above we see that the output voltage is 200V. During the testing phase, we were not able to identify what was the problem, so we proceeded to make the test with this issue. But we have identified the problem – the issue is that we made a mistake when we were ordering components and instead of ordering a 1.1 MOhm resistor (R2) we ordered a 1.3 MOhm resistor. If we verify the calculation with 1.3 MOhm resistance – we see that the output voltage is 200V.

So, going back to the figure we can observe the switching voltage does not have a lot of noise. The frequency is around 70 kHz and the duty cycle is around 5%.

The output current can be calculated using the formula below:

$$Iout = \frac{Vout}{Rload} = \frac{200V}{16.8\ KOhm} = 11.9\ mA$$

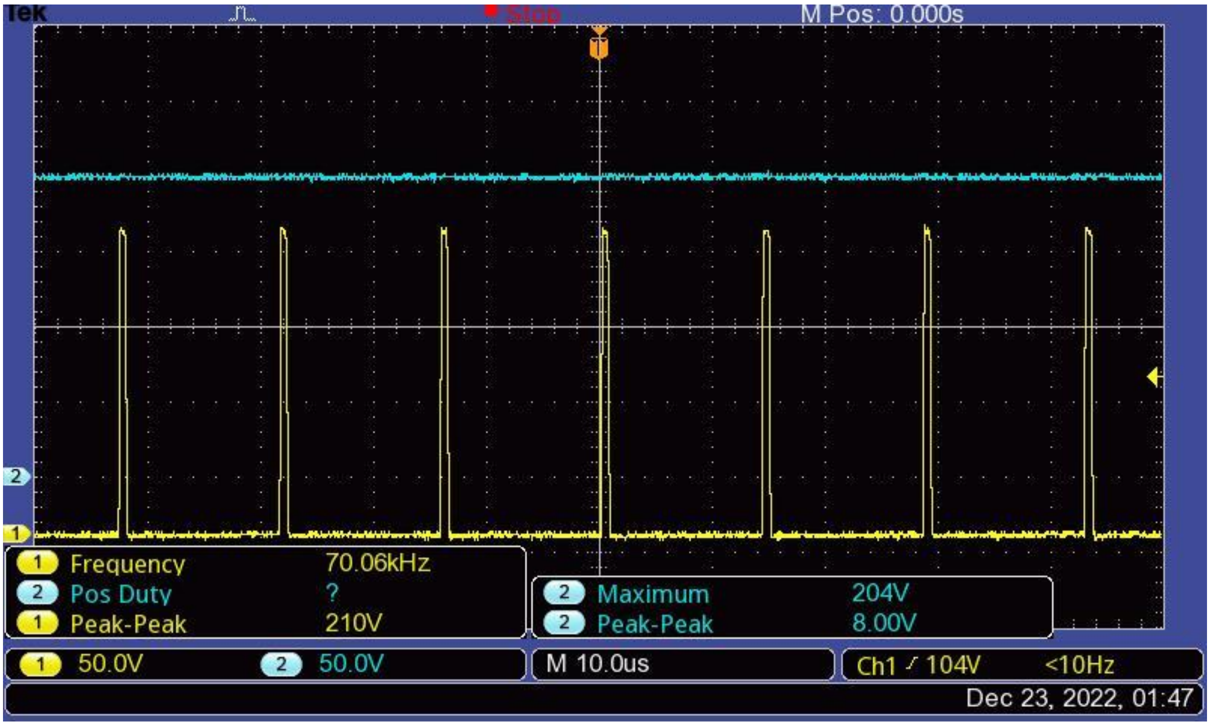
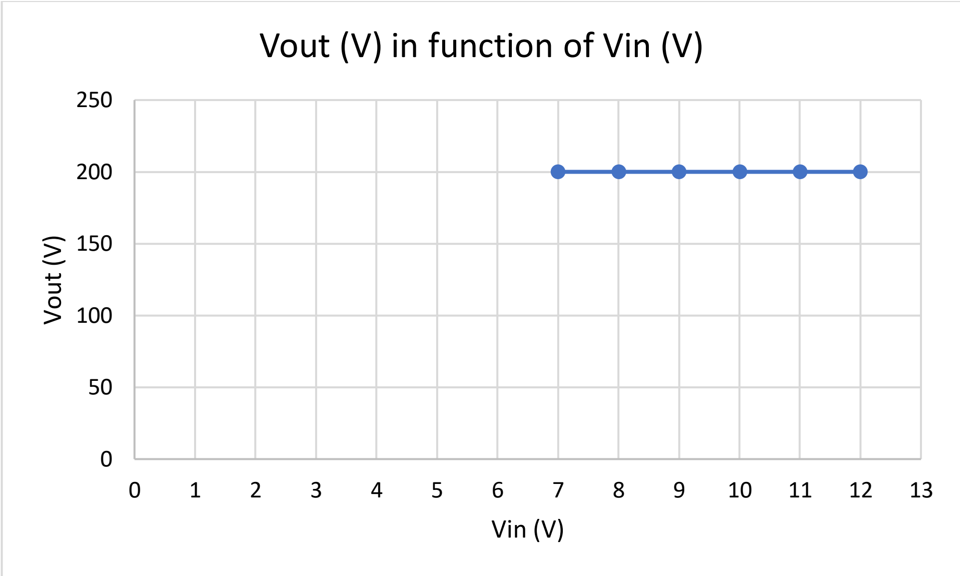


Figure 3.2b: Output voltage (blue line) and switching voltage (yellow line) with no load connected

As can be seen in figure 3.2b we achieved very similar results for the output and switching voltage, even though we did not connect any load to our circuit.

3.2.1 Input regulation

To test the stability of output with different inputs we connected a load of 16.8 KOhm to simulate the required current output. Then, we measured the output voltage (directly at load) with different input voltages: 7V to 12V.



Graph 3.2.1. Vout in the function of Vin

As can be seen from graph 3.2.1 the output voltage remained unchanged with different inputs. The results are very good and show that our circuit is stable and is not affected if a user will change its input.

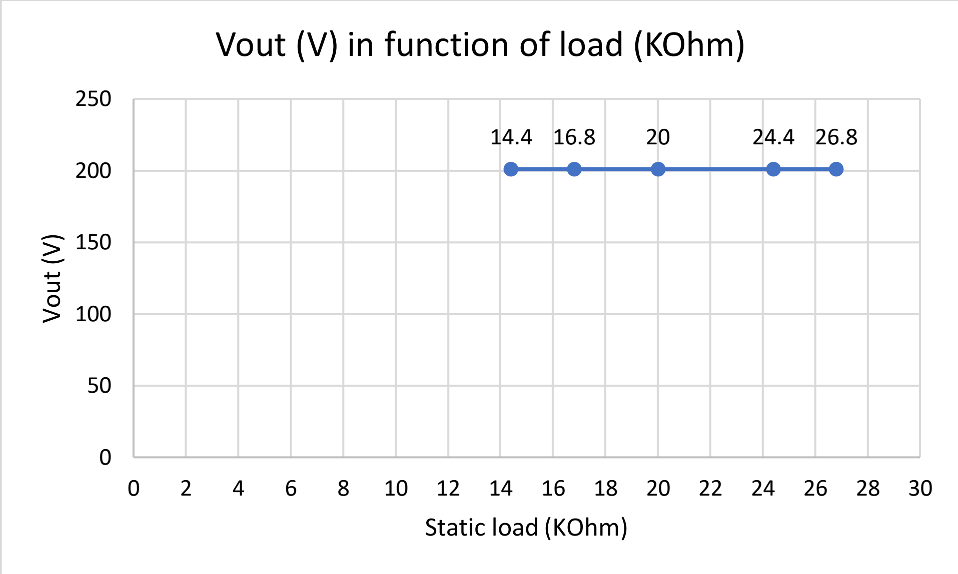
3.2.2 Static load regulation

To make a test for the static load regulation we measured Vout on different loads with a fixed input voltage (9 V).

Loads that we used during the test:

- No load
- 14.4 KOhm
- 16.8 KOhm
- 20 KOhm
- 24.4 KOhm
- 26.8 KOhm
- Nixie tube

To measure Vout we attached a probe to the PCB terminals to prevent any voltage drop over the wires to the load.



Graph 3.2.2. Vout in the function of load, values on the graph are load values

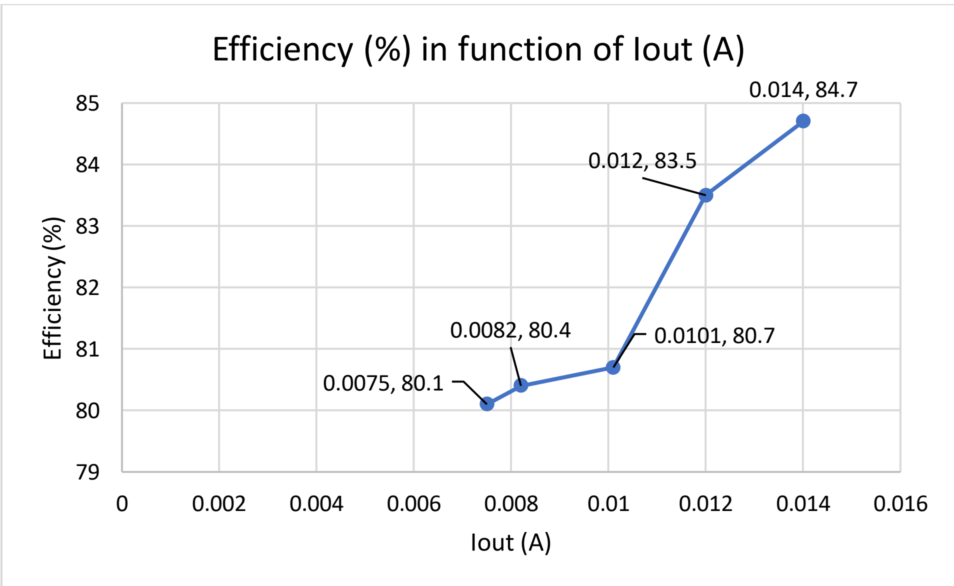
As can be seen from graph 3.2.2 we achieved good results and our output voltage does not change with the different loads connected to our circuit.

Since the nixie tube has dynamic resistance we did not include this measurement in the graph, but the output voltage was the same – 201V.

3.2.3 Efficiency

To measure the efficiency of our circuit we measured Vin, Vout, Iin (from the power supply), and Iout (was calculated out of the Vout measurement using this formula: Iout = Vout/Rload). The loads used to calculate efficiency are the same as in section 3.2.2.

Vin was measured directly on the terminals of the PCB and Vout on the terminals of the load resistor.



Graph 3.2.3. Efficiency (%) in the function of Iout (A)

As we see from Graph 3.2.3 we have a rise in efficiency with the rising output current. This is expected. The efficiency was calculated using the formula below:

$$\eta\ (\%) = \frac{Vout * Iout}{Vin * Iin} * 100\%$$

$$where\ Iout = \frac{Vout}{Rload}$$

Moreover, according to the datasheet, we should expect an efficiency of up to 90% for 30 mA to 2 A load currents. Since our maximum load current is 14 mA we cannot expect an efficiency of 90%. Nevertheless, our results are very good, and we can say that our circuit is efficient.

3.2.4 Ripple and noise

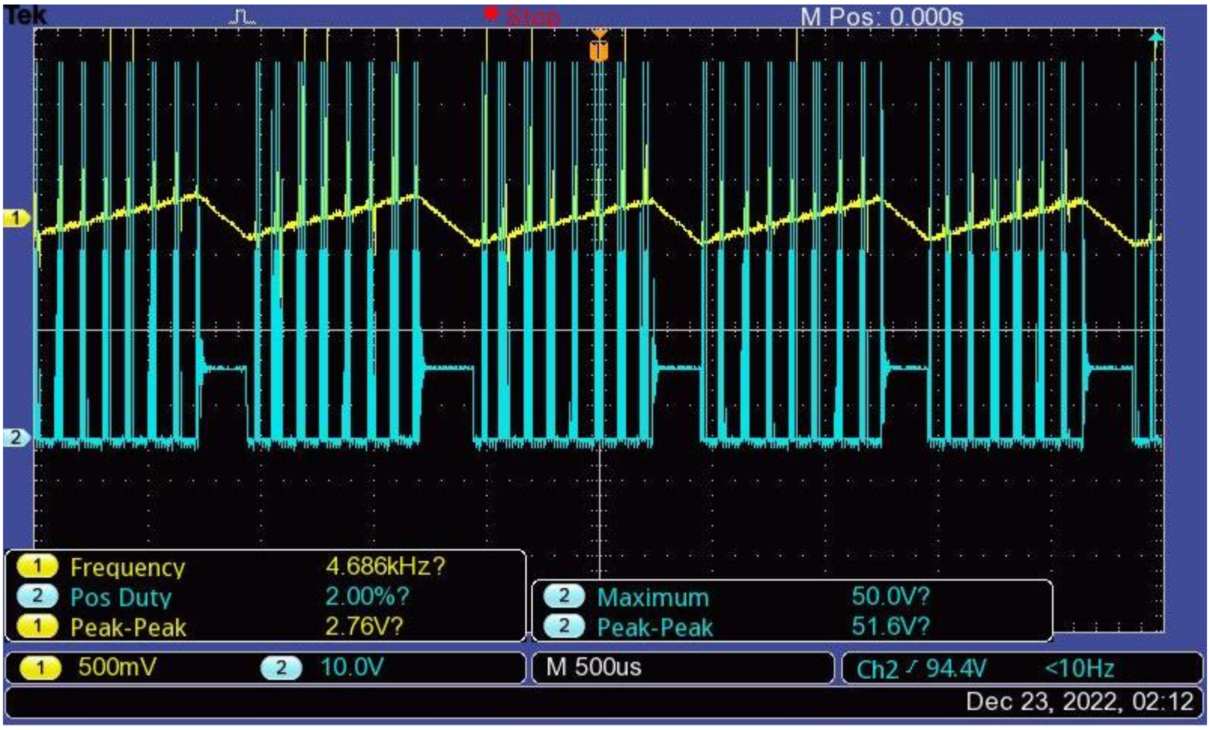


Figure 3.2.4a: Switching voltage (blue line) and output voltage (yellow line) ripple with 16.8 kOhm load connected

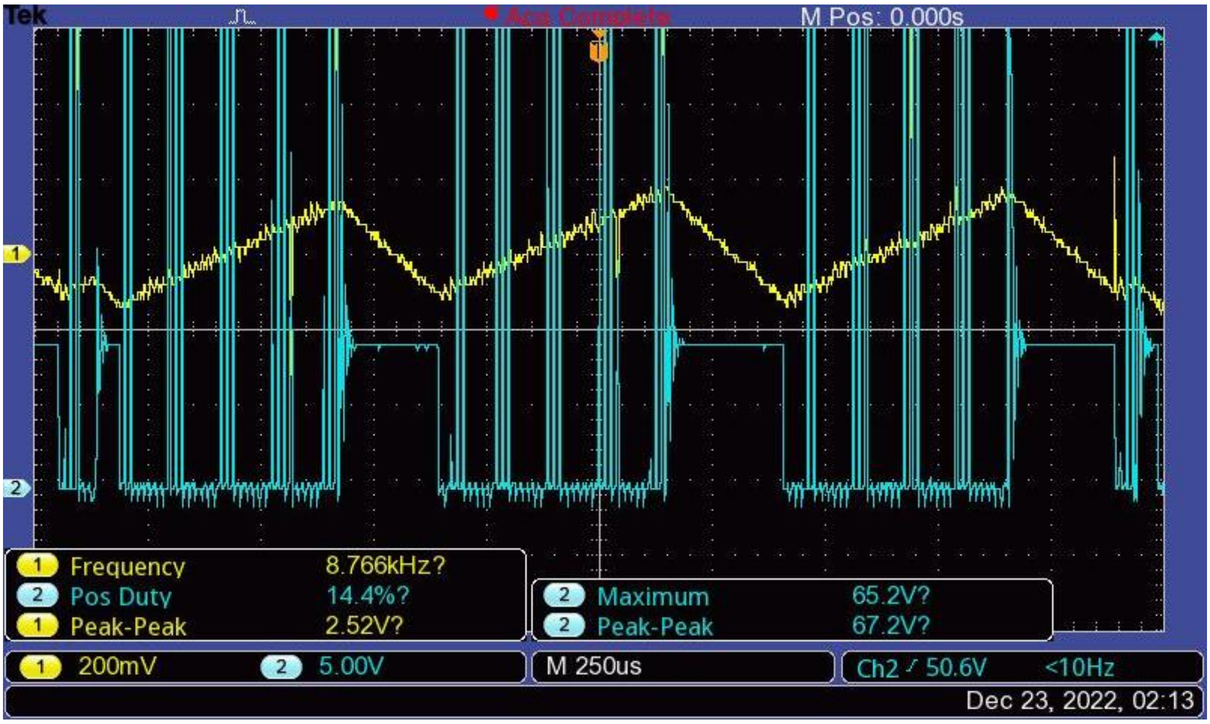


Figure 3.2.4b: Switching voltage (blue line) and output voltage(yellow line) ripple with 20 kOhm load connected

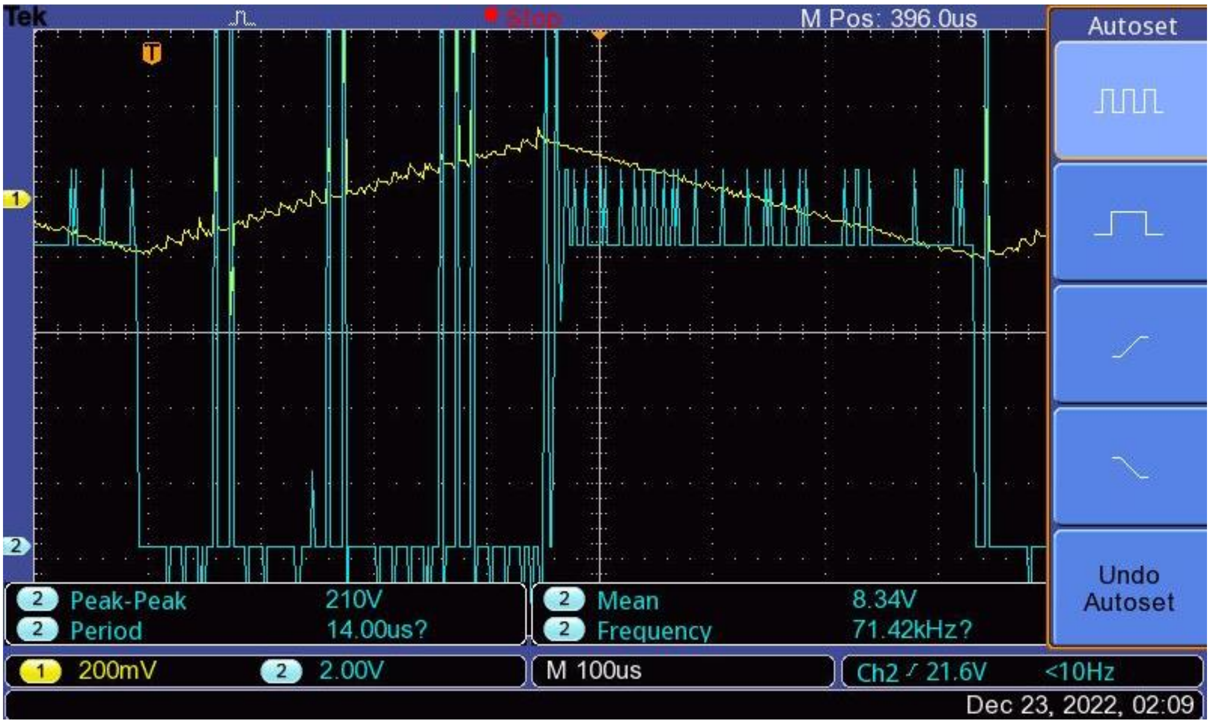


Figure 3.2.4c: Switching voltage (blue line) and output voltage (yellow line) ripple with 26.8 kOhm load connected

As can be seen in figures 3.2.4a – c, the output voltage ripple is in the range of 350 mV. This is very small compared to the output voltage value, which is 201V. From these values we can calculate the percentage ripple:

$$\%r = \frac{350mV}{201V} \times 100\% = 0.17\%$$

From this calculation we can see that the voltage ripple at the output is less than 1%, which means it is negligible.

We can also see from the figures that when the different loads are connected at the output, the output voltage ripple does not change in magnitude. However, it does change in shape. In Figure 3.2.4a, we can see that when the load of 16.8 kOhm is connected, the output voltage increases for around 700 µs and then decreases for approximately 250 µs. However, when the 20 kOhm load is connected, the output increases for a shorter period and increases for a longer period. For an output load of 26.8 kOhm, the rise and decay times are almost equal. This happens because of the switching. For a smaller load, the period of the switching voltage is bigger, and the MOSFET is ON for a longer period.

Moreover, from Figures 3.2.4a – c, it can be seen that the switching voltage is unstable. The pulses of approximately 200V are visible. This is mainly due to the imperfections in our measurement. The switching voltage was measured by holding the probe by hand without attaching it to the board.

3.2.5 Temperature

For the temperature measurement, the input voltage was set to 9V, and the nixie tube was connected to the output. Figure 3.2.5 shows an IR probe image of the PCB, it was taken after 10 minutes of leaving it on. As can be seen from the figure, the most heat-dissipating component on our PCB is the inductor. It has reached a temperature of 27.4 °C. According to the datasheet of the chosen inductor, the temperature rise should be about 10 °C for our application (10 mA output current). The actual temperature rise is, however, higher than expected, but is still acceptable.

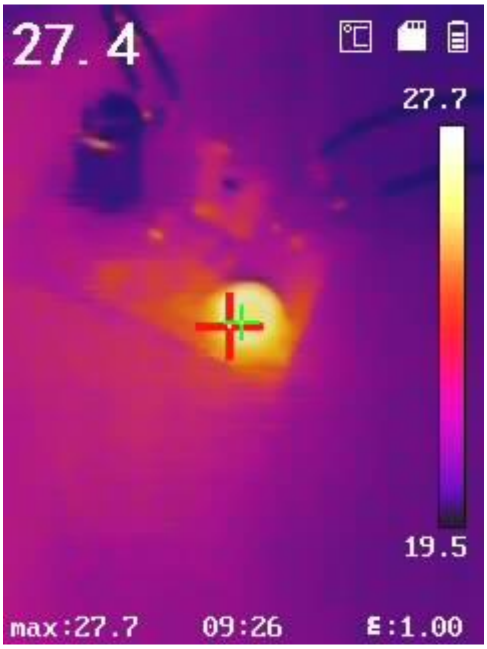


Figure 3.2.5: IR probe image of the PCB

Chapter 4

4.1 Conclusion

To conclude, we would like to say that our project met our expectations. The measurement results were acceptably close to the predicted ones, except for the issue with the choice of the resistor (R2). The output is very stable and the ripple at the output is negligible, which makes the design very efficient. Moreover, the components, especially capacitors and resistors were chosen to be with a safe margin of tolerance. This allowed us to carry out different tests with multiple loads and not worry that something will break. The heat dissipation is not much, which makes the components and the PCB more long-term. The components on the PCB were placed in a way to optimize the size so that it can be compact and portable. Overall, we can say that our design was successful.

In the future, it would be very interesting to further extend the design and connect a board with a microcontroller to program the nixie tubes to show different digits. In this way, the series of nixie tubes can be connected and by programming the microcontroller, the so-called nixie clock can be created.