ELEKTRONICA-ICT

Project Ontwerpen

# Application note- Soldeerstation

*Author* *Kobe Dieryck*

## Contents

# 1 Introduction

For this assignment, a soldering station was chosen. The project aimed to achieve a minimalist, small, and sleek design while maintaining a high level of power. The preferred connector for this project was the *Hirose RPC1* 6 pin connector, commonly found in JBC soldering stations. This choice allowed for easy swapping of the iron without the need for any modifications.

Furthermore, a single modification was made to the design by incorporating SMD (Surface-Mount Device) Components. This decision aimed to simplify the assembly process and achieve a more compact overall design.
The foundation of this project is based on the Elektor magazine release 507 (June 2021) [1], which provides detailed instructions for constructing a soldering station. The design utilizes the *Atmega4809* microcontroller and a 2 x 12V transformer with a power rating of 60VA.

# 2 Material and methods

In the original project, a 60VA transformer was used, which proved to be sufficient for the *JBC-C210* soldering iron. However, in order to accommodate general-purpose soldering, a *C245* iron would be more suitable. Consequently, a larger transformer was required, and a 150VA transformer was selected for powering the *JBC-C245* [2]. The power consumption of this iron is approximately 130W. Since the iron operates solely as a resistive load, no conversion to VA is necessary. Thus, without factoring in any losses, we can expect to have approximately 20W of excess power. These additional 20W will be allocated for the 5V power supply.

For the connectors the Hirose RPC1 was chosen, simply because these are used in the JBC stations. A C14 connector is used to power the station.

Mostly, components were chosen by Elektor [1]. A choice was made to mostly use SMD components instead of the THT (through hole technology) This was because the size was something that needed to be addressed, because the size of the PCB was quite large.

## 2.1 Design

For the design small sleek, and modern was chosen. A simple 3D-printable design would be a nice addition to this design, this way no complicated manufacturing needed to be done. Every design choice for the case was made using this decision.

## 2.2 Altium designer

The schematic was subdivided into four parts:

- Thermocouple Figure 2
- power supply Figure 3
- microcontroller Figure 4
- output stage Figure 6

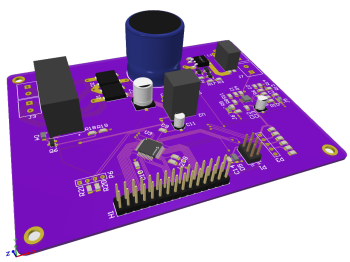

Figure 1: PCB

In the process of PCB design, a schematic was created for each item, which allowed for grouping them into rooms [3]. These rooms serve the purpose of organizing the PCB layout. By arranging the items within each room, they could be sorted, compacted, and individually routed before being moved around to achieve a compact overall design. The final render of the PCB can be seen in Figure 1 and in Figure 5
To begin the PCB design, the footprints of the components were added to the pcblib file. The pinout of these components was cross-referenced with the manufacturers' datasheets. Subsequently, the PCBlib components were linked to the SCHlib file. Next, the schematic design was imported into the PCBdoc file, which included the imported rooms [3]. With this setup, the components could be placed within each room. Once the placement was done, the interactive routing

tool was used to route each room individually. After outlining the PCB, the rooms were organized on the board to complete the design.

## 2.3 Soldering

At the soldering stage, the first components to solder where the SMD components. This is because the through-hole components cannot handle the temperatures needed by the oven wherein these are soldered.

### 2.3.1 SMD

Firstly, the components were knolled in separate bags, categorized by component type and value. Then, the SMD stencil was carefully placed on the PCB, ensuring correct alignment. Unfortunately, a minor error occurred during this step. The stencil ended up slightly misaligned, resulting in a smear of solder paste on the package of the *Atmega4809* microcontroller. Consequently, when the PCB was placed in the reflow oven, shorts occurred at the MCU(microcontroller unit). Though these were easily fixed with a soldering iron under a microscope.

### 2.3.2 THT

After that components that were left over needed to be soldered, such as the big electrolytic capacitor and the relay. This was easily done with a clean soldering iron.

# 3 Results

This section explains what happened throughout the project and while evaluating it.

## 3.1 Schematic

The schematic went pretty fast and easy. The connections were made with offsheet connectors [4], thus making nets easy to read and to connect the rooms to each other. You can see this in Figure 2.

## 3.2 PCB

In conclusion, the PCB has turned out very well, although there are aspects that could be improved in a future board revision. One example of such improvements includes incorporating additional test points and solder bridges for more points of isolation. Furthermore, during the design process, I aimed to create highly compact spaces, yet upon reviewing the final outcome, I noticed a few unnecessary empty areas that could have been avoided. It seems that I could have achieved an even more condensed design, although the current iteration already represents a significant enhancement over the original. Although ultimately unnecessary, I decided to leave the high current traces exposed without solder mask. Additionally, I now realize that I made an oversight by placing the connectors too close to the components, resulting in the inability to utilize the JST-XH connector in one particular location.

## 3.3 Assembly of PCB

The ease of assembly of the PCB pleasantly surprised me, as the entire process unfolded smoothly. However, I did encounter a minor setback with the ATmega, as its pins were slightly smeared with solder paste. Nevertheless, there were no significant issues beyond that.

## 3.4 Case

The design created showcases a personal appreciation, with recognition of potential areas for improvement. The feet, characterized by a slow and gentle curve leading to the screen, are particularly pleasing. Furthermore, the modular nature of the design allows for the seamless interchangeability of each panel, adding to its versatility.

# 4 Discussion

In conclusion, despite encountering challenges in soldering, testing, CAD design, and PCB design, the project achieved an overall successful outcome. The PCB, although not fully functional, served as a notable achievement, complemented by an appealing case. Mistakes were addressed and resolved throughout the process, showcasing adaptability and problem-solving skills. The utilization of tools such as a microscope, tweezers, and a desoldering wick effectively resolved soldering issues. The replacement of the original Atmega4809 after an ESD incident demonstrated the ability to overcome setbacks. Additionally, the lack of test points in the PCB design was ingeniously resolved through resourceful methods like soldering wires or employing probes. These solutions exemplify the dedication and perseverance involved in the project's completion.

## References


[1] L. Lemmens and M. Claussen, "Soldeerstation," vol. 507, pp. 30–36, 2021.

[2] "Cdb soldering station." https://www.jbctools.com/cd-b-general-purpose-product-1605.html, 2023.

[3] "Working with rooms on a pcb in altium designer." https://www.altium.com/documentation/altium-designer/working-with-rooms, 2022.

[4] "Working with an off sheet connector object." https://www.altium.com/documentation/altium-designer/schematic-off-sheet-connector, 2017.

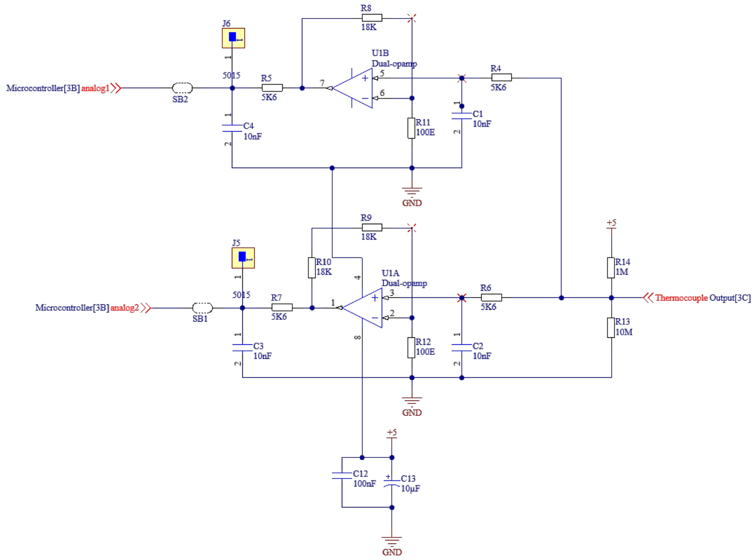


Figure 2: Thermocouple schematic

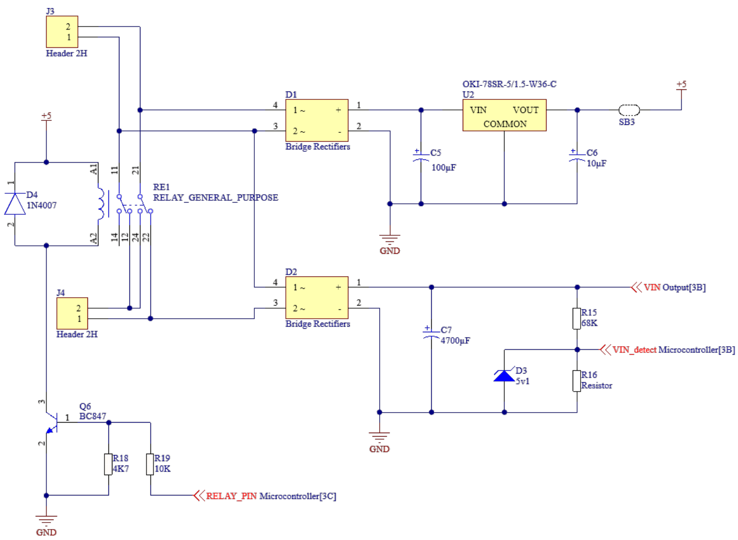


Figure 3: Power Supply schematic

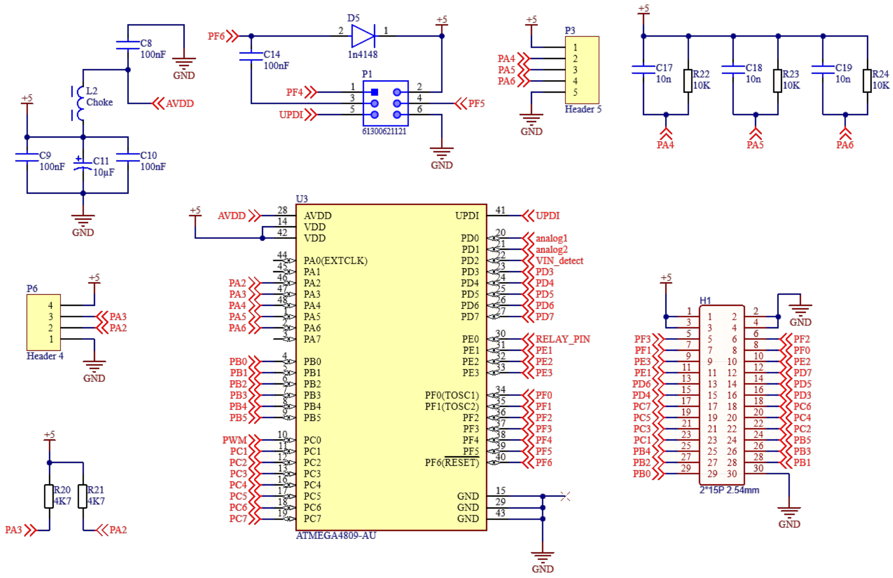

Figure 4: Microcontroller schematic

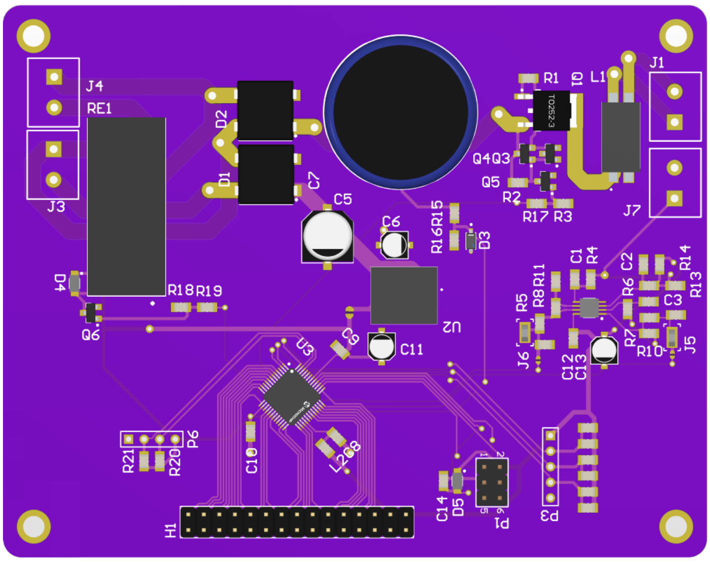

Figure 5: Top orthographic view of the pcb

Figure 6: 3D Render of the case without the knob