**Soldeerstation ontwerpen**

|  |  |
| --- | --- |
| Author | Axel Vanherle |

Content

[1 Introduction 1](#_Toc63673997)

[2 Material and methods 2](#_Toc63673998)

[3 Results 2](#_Toc63673999)

[3.1 Subtitle 1 2](#_Toc63674000)

[3.1.1 Subtitle 2](#_Toc63674001)

[3.1.2 Subtitle 2](#_Toc63674002)

[3.2 Subtitle 2 2](#_Toc63674003)

[3.2.1 Subtitle 2](#_Toc63674004)

[3.2.2 Subtitle 2](#_Toc63674005)

[4 Discussion 2](#_Toc63674006)

[5 Reference list 2](#_Toc63674007)

# Introduction

This application note is about a solder station, that is designed by Axel Vanherle. The solder station has an adjustable heat output, and a 3-digit display to help closely monitor that temperature. This project is a suggestion by our teacher about the subject, and we got the schematics for the station from Elektor. There are some customizations to that schematic though, and the PCB is a fully custom design. The case and mechanical design is also fully custom, that are made to match the PCB and the rest of the hardware.

# Material and methods

To design the solder station the primary programs that were utilized are Altium Designer for the schematic and PCB design, and Autodesk Fusion 360 for the case design, which is 3D printed. The hardware which is used for the project is mainly ordered from Mouser, but small orders were also placed with TME and Conrad.



The list above shows the components that were chosen, the manufacturer where they were made, the amount ordered and the total price in euros. All components were chosen based on four criteria; do they work in the design, availability, price and whether or not the footprints are easily available. The delivery date on all of them is about the same. Components ordered from Mouser arrived in a few days. Conrad and TME followed up with a delivery time of about a week. Not all footprints were however easily available on the manufacturer websites, so other tools were used to get these. Third party ran websites were used for this, and those mainly include [componentsearchengine.com](componentsearchengine.com/) and [www.ultralibrarian.com](http://www.ultralibrarian.com.). To get the downloaded footprints into the Altium library the tool “Altium Library Loader” was used. This converted the downloaded ecad models to a usable Altium library files.

To get the project ready for PCB printing and soldering there were five major steps that had to be done. Firstly, there is the component selection. Components were chosen based on the four criteria previously listed above. Once the components were selected the next step is finding footprints to match the components, so the schematic and PCB design phase can start. This was done utilizing multiple tools. Mainly the manufacturer websites were used to get the footprints, but if this didn’t pan out already mentioned third party websites were used. Once the libraries were completed, filled with all my previously selected component footprints the schematic design phase could start. For this step the base design from Elektor magazine was primarily used. But the design wasn’t closely followed due to other components being chosen. Having the base design, however, is a great utility that was often used. To make sure the design would work with the newly chosen components the datasheets of said components were always used. Once the schematic design phase is complete the PCB is now ready to be designed. This phase relies heavily on the schematic design to be in order, otherwise the final product wouldn’t work. In this phase datasheets again played an important role. They hold the information needed to know the dimensions of the components, so there wouldn’t be an issue fitting them once the final product is ready. Then once the PCB was completed, and the specification of this were know (width, height, length) the case design could start. For this step a case was design specifically to my needs. Due to the nature of the PCB design, there is a lot of freedom to where the 7-segment display and rotary encoder are placed. The case is designed in fusion 360, and will be 3D printed.

# Results

[Describe the end result you accomplished.

* Describe every aspect of your device. How does it function?
* Add an image of the electrical schematic, PCB design, finalized mechanical design, and finalized product

Write a well-structured text using subtitles and paragraphs.

**+/-500**]

## Subtitle 1

### Subtitle

### Subtitle

## Subtitle 2

### Subtitle

### Subtitle

# Discussion

[Reflect on and discuss your project.

* Which difficulties did you encounter during the design process and why? How did you solve these issues?
* Reflect on the process: did things go as expected? Would you choose the same approach if you had to do the project all over again? Are there issues that still need to be fixed? How come?

**+/-300 words**]

# Reference list

[Insert your reference list here.]