Soldering Station: Drawing, Designing, Manufacturing and Testing

|  |  |
| --- | --- |
| Author | Thibault Luyts |

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 explains all the steps for making a soldering station, this includes drawing the electrical schematics and PCB layout, designing a case for surrounding the finished PCB, and finally manufacturing and testing the soldering station.

The soldering station is designed to be a small and have interchangeable soldering irons. It even has an adjustable temperature range that can be seen on a 3-digit display. The project begins with an electrical schematic witch can be found in Elektor 5 from 2021. Most of the information has been provided and explained by a professional teacher.

# Material and methods

The primary program used for drawing the schematics and designing the PCB layout for this project is Altium Designer. The case design is fully designed with the use of Autodesk Fusion 360. These programmes are all used on the computer and are used by professionals around the world. The prototyping and final design for the case are 3D printed by a FDM or resin printer. The drawing of the electrical schematics is a time consuming process for someone with no Altium Designer experience. The designing of the PCB layout is quicker but with more practical and customisation options.

The components/hardware list is given in the Elektor. In this design the list is altered for better, newer and more available components/ hardware. This list is largely ordered from Mouser, some orders are from Würth Elektronik.



The components/hardware in the list above shows the used components, the manufacturer and where they were manufactured, the amount of each component used in the project and the total price of each component. The components chosen for this design where based on four factors: the price, the delivery date, the availability of the components and whether or not the footprints are easily available. The price is useful to show if a more expensive component is better or more or less the same on different websites. The delivery times are easily as important as the price, it may take a couple of weeks to deliver some components. The availability is something to keep an eye out for because some components may take up to half a year to restock. Footprints are a useful tool for de giving both the schematic and PCB layout a more simple and clearer look. Not all footprints were easily available on the manufacturer websites.

before PCB printing and testing the design there are five steps that needed to be checked. Firstly, there is the component selection. Components are chosen based on the four criteria previously listed. Once the components are selected the following step is finding footprints to match the components, so the schematic and PCB design phase can start. Next is filing libraries with components and their footprints. Once the libraries are completed the schematic design phase can start. For this step the base design from Elektor magazine is primarily used. But the design isn’t closely followed due to other components being chosen resulting in minor changes. Having a base design is a great utility for checking if the design was correct. 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 completed the PCB is now ready to be designed. This phase relies heavily on the schematic design to be in order, otherwise the final product can’t work. In this phase datasheets play an important role. They hold the information needed to know the dimensions of the components, so there can’t be an issue fitting them once the final product is ready. Then once the PCB is ready, and the specifications of this PCB design are known (width, height, length) the case design can begin. In this step a case is designed specifically to the specifications of the PCB . Due to the nature of the PCB design, there is a lot of freedom to where the 7-segment display and rotary encoder can be placed.

[Give an overview of the materials and the methods you used:

Materials: the components for the device

* Which materials (i.e. hardware and software) did you use and did you compare?
* Which materials were not useful and why not? Use proper, objective evaluation criteria.
* Add the Bill of Materials including an indication of the price, supplier name and delivery date (table in English!)

Methods: specific tools and procedures you use to collect and analyze data (for example, experiments, datasheets…)

* Include a schematic representation (i.e. flowdiagram) and explain this representation by providing a step by step overview of the design process, production process and testing process (including a description of the mechanical design).

**+/- 500 words**]

# 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.]