**Design of a soldering station**

|  |  |
| --- | --- |
| Author | Mauro De Bruyn |

Content

[1 Introduction 1](#_Toc104846504)

[2 Material and methods 2](#_Toc104846505)

[3 Results 3](#_Toc104846506)

[3.1 Schematics 4](#_Toc104846507)

[3.1.1 Explanation 6](#_Toc104846508)

[3.2 PCB design 6](#_Toc104846509)

[3.2.1 PCB 6](#_Toc104846510)

[3.2.2 Autorouting 6](#_Toc104846511)

[3.3 Mechanical design 7](#_Toc104846512)

[3.3.1 3D Drawing 7](#_Toc104846513)

[3.3.2 Knob 7](#_Toc104846514)

[4 Discussion 7](#_Toc104846515)

# Introduction

This is an application note on the design of a soldering station, the soldering station made in this note is a high end soldering station that can be used for soldering small components precisely.

The reason for the build is to see what can be achieved by making a project using skills learned in school, it’s a fun way of learning new techniques as well.

The soldering station will have an on/off -switch, a knob for setting the temperature of the soldering tip and a display showing the set temperature.

First electrical schematics were made and a PCB was designed, this all using the program “Altium”. The final step in designing the soldering station is making a case where the different components (PCB’s, Display, …) will be placed in, for designing the case “Autodesk Fusion 360” was used.

# Material and methods

To make this project, a lot of different techniques and tools are used.

Starting the project the tool “Altium Designer” was used to draw the electrical schematics. These schematics are based on an existing project that can be found in the Elector magazine.

After checking the schematics on possible important errors this step is completed and now the PCB design is ready to be made.

In Altium the PCB was designed based on the already drawn schematics. The components can be placed on the PCB in the program and the first design choice needed to be made being making 2 PCB’s for the project or 1. Why in this projects 2 PCB’s were used will be explained later in this application note.

When the design was finished the PCB could be ordered from JLCPCB.

Drawing the case was the next step, this was done by using Autodesk fusion 360. Autodesk is a easy to use program to design 3D models for beginners. You get fast and satisfising results in no time. The case made for this soldering station is designed to be easy to use for left hand people as the connector of the soldering iron sits on the left side of the design. The top of the design holds the display, this is a flat part as the PCB was to large to sit in a curved design.

The components of witch the price, supplier and delivery date are blank in the list below are components provided by PXL. The choice using those components was made because when ordering these components in small quantities the price can be higher than when buying in large batches.

When looking for the remaining components the price and soldering type where very important. Ideally SMD components are used because those are smaller than true hole components. The footprint available was also a big part in choosing the components. Because the design had to be made in a short notice the footprint of the component was needed quickly, as some of the part manufacturers did not provide the footprint the better option was to look for different components.

The footprint of a component is used to draw the schematic and PCB design, its more important for PCB design as it displays where the holes or soldering pads need to be on the PCB.

Choosing the components, the current and voltage was an important factor to account to. The schematics were used to check witch components had to have a specific current, voltage, wattage or accuracy.

The accuracy is also very important as some components have a big influence on the rest of the circuit and could damage it when it differs to much from the claimed values. The total price of the components and external power chord came down to 100€ this does not include the case, PCB’s and knob for the case.

Table

Description automatically generated

# Results

As the project is finished, the details will explained here.

There is a big knob on the frond of the device, this knob is used to set the preferred temperature of the soldering tip. Turning the knob clockwise will increase the temperature, turning it the other way will obviously decrease the temperature. When you press the button the power to the soldering iron is cut, this is a very handy feature to quickly turn the heat on or off.

The display module, also located on the front of the device, will display the set temperature of the soldering tip. It is important to know that this is not the physical temperature of the soldering tip but the set temperature.

On the left of the soldering station a connector can be found, this connector will be used to fit a soldering iron in.

On the back of the device there can be found a connector for the power supply, switch and a place to hold a fuse in.

The connector holds a 3 wire cable, this cable is used to power the whole device.

The integrated switch is used to cut off the power supply so turn off the device.

For safety a fuse needed to be fitted on the device, it sits behind the power supply and protects the device from to high of a current or voltage flow. The fuse will physically separate the power supply from the device. When the fuse is broken it can easily be replaced because there is a drawer like shutter on the outside of the device. For safety reasons the fused cannot be removed with the power cord still attached.

## Schematic: DisplaySchematicsDiagram, schematic Description automatically generated

A picture containing text, sky, map

Description automatically generatedDiagram

Description automatically generatedDiagram, schematic

Description automatically generated

### Explanation

A picture containing text, electronics, circuit

Description automatically generatedShown above is the schematic design that was used to make the PCB design. The choice was made to split up the design into 5 parts as its gives more structure to the project, searching for faults in the PCB is also easier with an uncluttered design.

## PCB design

### PCB

The image shown shows a 3D print of the PCB design.

### Autorouting

Most of the design was made using the auto routing function in Altium designer, however some connections needed to be modified as those connections are transporting higher currents than the rest of the board. We manually searched for the connections and made them thicker.

There were some problems doing this as my components as well as the other routes were so compact. By manually moving the other components and connections enough space was made to route the ticker tracks.

A picture containing white

Description automatically generated

## Mechanical design

### 3D Drawing

the image above you can see the 3D version of the case in Autodesk Fusion 360, this case is printed using a 3D printer and is later fitted with all of the electrical components.

### Knob

W searched for a button that could fit over the control rod of the encoder.

# Discussion

For the most part the project went smoothly but when starting out soldering the PCB some errors where found.

The drawing of schematics was easy but it took some time to get the library’s ready to use, after competing the library’s the schematics were finished in no time.

The same was for the PCB, it was mostly placing the components that took some time but it was not to bad.

Some problems were encountered while generating the PCB using auto routing. The footprint of the transistors was to small for JLCPCB to make so changes in the PCBlibrary needed to be made.

While drawing the PCB a wrong footprint for L1 was used, it was supposed to be an SMD component but instead it was true hole like the other coil L2. To fix the problem a true hole part was searched with the same Henri value so it would fit into the “wrong” footprint.

Also the 10 pin header box did not fit on the PCB as the footprint of the manufacturer turned out to be wrong, the outline of the component was drawn to small. This later turned out to be a problem as the header was placed close to another component.

Drawing the case was a bit hard at first as it was the first time using the program, after a while the design of the case went well and was finished in no time.

Soldering the PCB’s was not a hard task ether as there were some good preparations beforehand while making the schematics for example.

It was a fun project to work on as you have to make the hole device from A to Z, not do you get to see the designer side of things but also the physical part of soldering and combining the different parts.