**The development of a soldering station**

|  |  |
| --- | --- |
| Author | Jasper Maes |
|  |  |

Content

[1 Introduction 1](#_Toc136616187)

[2 Material and methods 2](#_Toc136616188)

[3 Results 2](#_Toc136616189)

[3.1 Schematic 2](#_Toc136616190)

[3.1.1 Power supply 2](#_Toc136616191)

[3.1.2 The main IC schematic 3](#_Toc136616192)

[3.1.3 The display board 3](#_Toc136616193)

[3.2 PCB 3](#_Toc136616194)

[3.2.1 Design 3](#_Toc136616195)

[3.2.2 Order and mounting 3](#_Toc136616196)

[3.3 Finishing stage 3](#_Toc136616197)

[3.3.1 3D-printed case 3](#_Toc136616198)

[3.3.2 Final test 3](#_Toc136616199)

[4 Discussion 4](#_Toc136616200)

[5 Reference list 4](#_Toc136616201)

# Introduction

This document describes the complete assembly of the soldering station. Both technical details and instructions for use of the finished product are described. The design/build process is as well discussed.

The finished product will be able to drive a soldering iron based on user's selected value on the display. The fully self-designed PCB (Printed Circuit Board) can read the temperature sensor located in the soldering iron. This information will be used to control the thermocouple based on the user's setting. As a result, the soldering iron will maintain the desired temperature.

This project includes some basic circuitry, such as transistor circuits. However, it is not an overly complex circuit, which makes it an ideal way to get familiar with PCB design. Moreover, the final product is also useful, so it isn't a waste of money.

The project is based on an example from the 'Elektor' magazine. They already provided a schematic that was tested and worked without error. Based on this schematic, the soldering station being described here was made.

# Material and methods

To get started on the design of the project, Altium Designer was used. This software was used to create both the schematic and the PCB design.

The BOM (Bill of Materials), which can be found here, describes which components were ordered to realize this project.

[no bill of materials has been created for now, it will be added here].

The resistors and capacitors used were already available as SMD (Surface Mounted Device). This is why a conscious decision was made to use SMD for these components.

[More may be told about other components, once they have all arrived]

[More will be written about PCB testing in a later stage. The method for this is described here.]

# Results

The project can be divided into three main parts. Firstly, the schematic, secondly, the PCB design. As a finishing touch, a case will be designed and 3D-printed. As the very last step, the finished product is tested.

## Schematic

The schematic design consists of 5 parts:

* The soldering iron controller
* The power supply
* The temperature measurement
* The main IC schematic
* The display board

Three of them are highlighted below.

### Power supply

The power supply is where all the voltages enter. It is provided by the toroidal transformer whose input voltage is 230VAC. This is supplied by the euro connector, which contains a 250V fuse. The transformer has a power of 60VA, and the output voltage is 2 X 12VAC. So the input voltage of our PCB will be 2 X 12 V. That voltage is connected to K1 and K2. Here they form the 2 important circuits.

The first circuit is the one for +5VDC. The input voltage is rectified by the bridge rectifier (in this case B2) and goes to the DC/DC converter. From this point we can tap a constant voltage of 5V for our electronic components (e.g. ATMEGA,...).

The other input voltage is used, sometimes in combination with the previous one, to power the bolt. Software programming can determine how much voltage the bolt needs for proper operation. If the bolt needs only a lower voltage, the relay will be switched off by software. This allows the voltage to the bolt to be software-controlled.

The PWM signal driving the bolt is sent to K8 and K3 via the MOSFET to ensure that the bolt receives the right, amplified signal. The common mode choke (L2) located in front of the output (K8) serves to suppress RF noise on the cable of the bolt.

### The main IC schematic

The schematic of the main IC shows the following things.

The VDD pins. This is where the voltage at which the processor operates comes from. This is 5VDC, and is simply tapped from the +5V circuit discussed in the section above. The ground pins are obviously equally important.

At the place where the voltage comes in (at the level of VDD) is an LC circuit that provides a noise filter. On the PCB, these were therefore placed as close as possible to the ATMEGA, to ensure the highest quality signal.

Connected to the PA pins there is the connector (K12) which is used to connect to the displayprint.

The other circuits, such as temperature measurement, bolt control and power supply, were also connected to the ATMEGA.

Additional pin headers were provided on this schematic, in case the user would want to connect some extras. During soldering, these pin headers were not placed, as they are unnecessary, but each pin of the ATMEG has been brought to a header.

### The display board

On this diagram, the other side of the connector is provided. This is connected to [connector number] with a flat cable. In this way, you can flexibly go from the main board to the display board. This PCB contains the rotary encoder with internal pushbutton. This serves to set the temperature.

Additionally, the TM1637 can also be found on this circuit. This chip controls the 4-digit 7-segment display. The advantage of this chip is that it can be controlled with only 2 ports. This allows us to transfer all information from the display board to the main IC board (and vice versa) using only a few ports and cables.

This design will be placed in a separate project to make it a stand-alone PCB. This in order to place the PCB separately in the 3D design of the case.

## PCB

The PCB is where the magic happens. The above schematic was created entirely in Altium Designer. After carefully checking the pcb, it was ordered and assembled.

### Design

Altium designer was used to design the PCB. That is a software package used to design schematics and associated PCBs.

Before starting the project, component libraries were created. These consist of the schematic representation of the components, and their footprints. Once these are made, design can start smoothly.

The design has two main parts. The schematic part, and the pcb part. These are, at their turn, split into two separate projects. That's because in that way we have no physical connection between the main PCB and the display PCB.

The main PCB consists of 4 schematics. Bolt control, power supply, temperature measurement and the main IC sheet. These are subdivided and linked with 'ports'. This function in Altium Designer allows us to divide the schematic into several sheets. That way, everything stays clear.

On the displayprint, you can find everything that was descriped above. (see 3.1.3 The display board)

Based on the schematic design, the PCB design was generated. After some manual adjustments to the structure, Altium's autorouting tool was used to properly lay out the lanes. After several people had carefully checked the design, the PCB was ready for order.

### Order and mounting

## Finishing stage

### 3D-printed case

The case design depends on the final dimensions of the PCB. By next week the PCB design will be completed, as of then I will start the final PCB design.

### Final test

[to be completed later]

# 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.]