



Skill 16: Electronics

HW Design Part 2 "Engineering" (Day 1)

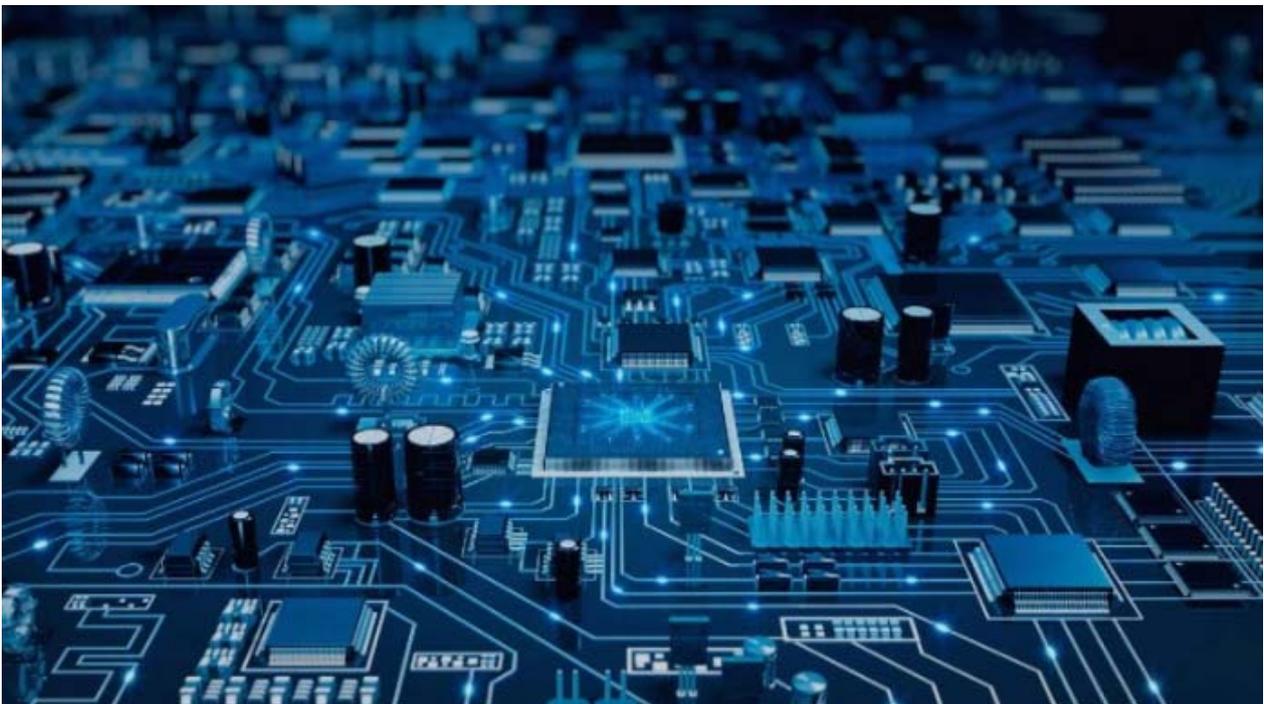


Fig. 1: Cover picture

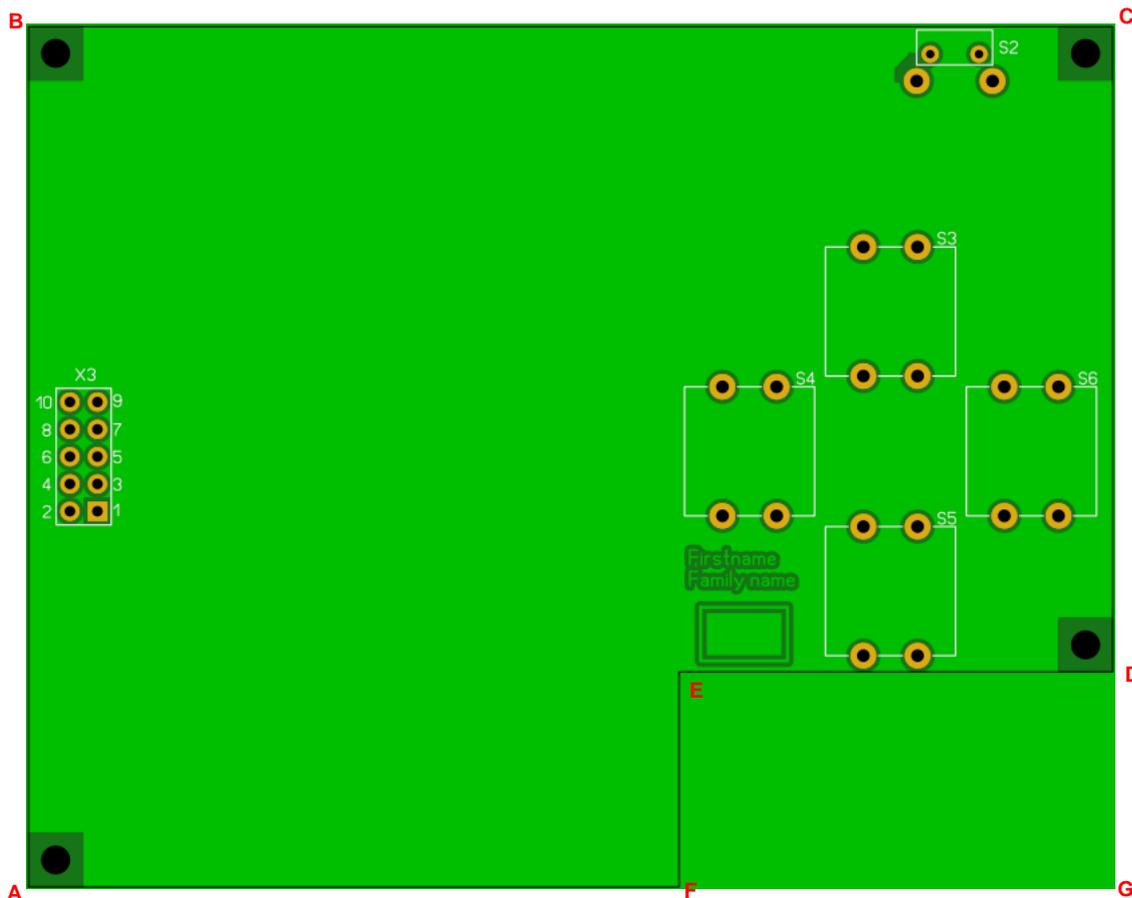
Table of Contents

Table of Contents	2
1 PCB specifications.....	3
1.1 PCB dimension.....	3
1.2 PCB fastening	4
1.3 Position specifications for components	4
1.4 Too boring?	5
1.5 Design Guidelines	5
1.6 Gerber Files.....	6
1.7 Documentation	6
2 Tools.....	6

1 PCB specifications

1.1 PCB dimension

The PCB must have the following shape:



The board layer for the base requirement is derived from the following vertices: (coordinates in mm):

Ref	X	Y
A	0	0
B	0	80
C	100	80
D	100	20
E	60	20
Q	60	0

The following information also applies to the printed circuit board:

- Maximum 2 Layers (Top & Bottom)
- No solder mask and no silkscreen

1.2 PCB fastening

The 4 mounting holes in corners A, B, C & D have an edge spacing of 2.5mm and a diameter of 2.7mm. The barrier area for the fastening screws has an edge spacing of 5mm. There must be nothing in this restricted area.

1.3 Position specifications for components

The following requirements for component placement must be met:

- The connector X3 must be placed so that the middle row (pin 5 and 6) is on the Y coordinate 40mm. The X coordinate should be selected in such a way that the provided add-on board can be securely plugged into the left side (A-B) and the distance between the two printed circuit boards is as small as possible.
- The following components must be placed on the back (side B-C):
 - USB Mini Jack X1 → The cable opening must be to the rear so that the USB cable can be securely plugged in and is not attached to the pcb edge or any other component
 - On/Off Switch S1 → The actuator should stick out of the PCB edge for easier operation.
 - Tactile Switch S2 → The actuator should stick out of the PCB edge for easier operation. The center of S2 should be 84.5mm on the X coordinate.
 - Infrared receiver U3 → The signal is received from behind
 - Infrared transmitter V3 → The signal is sent backwards
- The buttons S3-S6 should be arranged evenly according to the drawing and in the same area (bottom right).
- All display (V1-V9 & U2) and control elements (S3-S8 & P1) must be visible and operated from a 90° angle of view from above. Relevant is the effective display & control area and not the component surface (housing / pins / etc.)



- The 6 RGB LEDs (V4-V9) should be placed in a row and at equal distances.
- The components U1 and U2 are socketed. The area below can be used for components with a maximum height of 6mm.
- At the ESP32, you should take into account the fact that a USB cable must be plugged in for programming. The ESP32 is Wi-Fi capable therefore; there should be as little copper area as possible in the area of the antenna. The pin headers J2 and J3 are designed for jumper cables. The two pin headers must have a minimum distance of 2.54mm to the ESP32 development kit.
- The reading direction of the display is given according to the following image (SW version for commissioning). The text should be readable from the front (side A-F).



- First and last name should be included as text (font height at least 1.5mm) in the top layer. Consider the first name and last name location definition. Underneath, an area of 8x6mm (WxH) must be kept free.

- All parts on the default schematic (base part) must be placed on the top layer.

1.4 Too boring?

If you are done faster and you are bored, you can create a breakable PCB in the D-E-F-G (WxH = 40x20mm) area. The desired function is a gesture recognition with light and shadow. The following requirements apply:

- Use signals SDA, SCL, 3V3 and GND
- Use the multi-channel ADC "MCP3428"
- 4 Ambient Light sensors "TEMT6200FX01"
- Use 0805 resistors for current limitation, light sensor and I2C pull-ups
- Double-sided PCB assembly allowed, but the light sensors must be on the top layer.
- Create a cross on the top layer with SMD pads to solder a horizontal and vertical copper plate (thickness 1.6mm) on both sides. The crosses allow for a better signal when waiving with your hand or a light source over the light sensors.

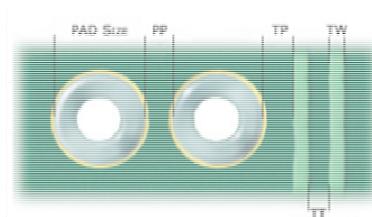


- Schematic is not mandatory.

1.5 Design Guidelines

You need to take care for the following listed requirements. The requirements are important, that your pcb can be produced.

- The minimum track width (TW) is 100µm. The minimum distance from tracks and pads (TT, TP & PP) is 100µm



- The smallest production hole diameter (PHD) is 200µm



- There should be a distance from 300µm to each edge of PCB (copper plane, pads, etc.)

1.6 Gerber Files

At the end of the task, the following gerber files must be stored in a ZIP file on a USB stick:

- Top Layer
- Bottom Layer
- Board Outline Layer
- Drill File

We must be able to check the ZIP file with an online gerber viewer.

Please indicate here which online gerber viewer you tested and works with your files:

1.7 Documentation

In addition, at the end of the task, the following PDF files are to be saved on a USB memory stick:

- Bill of Material → Only the material and quantity listed in the BOM will be provided for assembly. May also be an Excel file.
- Schematic
- Assembly plan of top and bottom side

2 Tools

The following tools are needed to solve the task:

- PC / notebook with a layout tool, as well as your own utilities
- Your own schematic from the morning